Agilent
PNA Series Network Analyzer
Printed Version of PNA Help User’s and Programming Guide

Supports Firmware A.09.42
Sept. 16, 2011
Warranty Statement

THE MATERIAL CONTAINED IN THIS DOCUMENT IS PROVIDED “AS IS,” AND IS SUBJECT TO BEING CHANGED, WITHOUT NOTICE, IN FUTURE EDITIONS. FURTHER, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, AGILENT DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED WITH REGARD TO THIS MANUAL AND ANY INFORMATION CONTAINED HEREIN, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. AGILENT SHALL NOT BE LIABLE FOR ERRORS OR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, USE, OR PERFORMANCE OF THIS DOCUMENT OR ANY INFORMATION CONTAINED HEREIN. SHOULD AGILENT AND THE USER HAVE A SEPARATE WRITTEN AGREEMENT WITH WARRANTY TERMS COVERING THE MATERIAL IN THIS DOCUMENT THAT CONFLICT WITH THESE TERMS, THE WARRANTY TERMS IN THE SEPARATE AGREEMENT WILL CONTROL.

DFARS/Restricted Rights Notice

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as “Commercial computer software” as defined in DFAR 252.227-7014 (June 1995), or as a “commercial item” as defined in FAR 2.101(a) or as “Restricted computer software” as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies’ standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Certification

Agilent Technologies, Inc. certifies that this product met its published specifications at the time of shipment from the factory. Agilent Technologies, Inc. further certifies that its calibration measurements are traceable to the United States National Institute of Standards and Technology, to the extent allowed by the Institute's calibration facility, and to the calibration facilities of other International Standards Organization members.

Contacting Agilent

Assistance with test and measurements needs and information on finding a local Agilent office are available on the Web at: http://www.agilent.com/find/assist

If you do not have access to the Internet, please contact your Agilent field engineer.

In any correspondence or telephone conversation, refer to the Agilent product by its model number and full serial number. With this information, the Agilent representative can determine whether your product is still within its warranty period.
This page intentionally blank for pagination.
This page intentionally blank for pagination.
This page intentionally blank for pagination.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Whats New</td>
<td>53</td>
</tr>
<tr>
<td>Administrative Tasks</td>
<td></td>
</tr>
<tr>
<td>Microsoft EULA</td>
<td>60</td>
</tr>
<tr>
<td>PNA User Accounts and Passwords</td>
<td>68</td>
</tr>
<tr>
<td>Computer Properties</td>
<td>72</td>
</tr>
<tr>
<td>Error-check and Disk Defragmenter</td>
<td>76</td>
</tr>
<tr>
<td>Operating System Recovery</td>
<td>77</td>
</tr>
<tr>
<td>Wake PNA at Specified Time</td>
<td>78</td>
</tr>
<tr>
<td>Windows Considerations</td>
<td>79</td>
</tr>
<tr>
<td>Quick Start</td>
<td></td>
</tr>
<tr>
<td>Front Panel Tour</td>
<td>81</td>
</tr>
<tr>
<td>Rear Panel Tour (E836x and PNA-L)</td>
<td>90</td>
</tr>
<tr>
<td>Rear Panel Tour (PNA-X)</td>
<td>92</td>
</tr>
<tr>
<td>Powering the PNA ON and OFF</td>
<td>97</td>
</tr>
<tr>
<td>Traces, Channels, and Windows on the PNA</td>
<td>100</td>
</tr>
<tr>
<td>Basic Measurement Sequence</td>
<td>107</td>
</tr>
<tr>
<td>Frequency Blanking</td>
<td>108</td>
</tr>
<tr>
<td>Internal Second Source</td>
<td>111</td>
</tr>
<tr>
<td>Connectivity Guide</td>
<td>113</td>
</tr>
<tr>
<td>PNA Preferences</td>
<td>114</td>
</tr>
<tr>
<td>VNC</td>
<td>120</td>
</tr>
<tr>
<td>LXI Compliance</td>
<td>121</td>
</tr>
<tr>
<td>Using Help</td>
<td>124</td>
</tr>
<tr>
<td>Help About</td>
<td>131</td>
</tr>
<tr>
<td>1. Set Up a Measurement</td>
<td></td>
</tr>
<tr>
<td>Preset the PNA</td>
<td>133</td>
</tr>
<tr>
<td>Measurement Parameters</td>
<td>141</td>
</tr>
<tr>
<td>Measurement Classes</td>
<td>149</td>
</tr>
<tr>
<td>Frequency Range</td>
<td>152</td>
</tr>
<tr>
<td>Power Level</td>
<td>161</td>
</tr>
<tr>
<td>Receiver Leveling</td>
<td>171</td>
</tr>
<tr>
<td>Sweep Settings</td>
<td>176</td>
</tr>
<tr>
<td>Trigger Setup</td>
<td>187</td>
</tr>
<tr>
<td>External Triggering</td>
<td>191</td>
</tr>
</tbody>
</table>
2. Optimize a Measurement

- Dynamic Range
- Number of Data Points
- Phase Accuracy
- Electrically Long Devices
- Reflection Accuracy
- Measurement Stability
- Noise Reduction Techniques
- Crosstalk
- Effects of Accessories
- Fastest Sweep
- Multiple State Measurements
- Fastest Data Transfer
- Using Macros

3. Calibrate a Measurement

- Calibration Overview
- Calibration Standards
- Calibration Wizard
- Select a Calibration
- Using Cal Sets
- Error Correction and Interpolation
- Calibration Thru Methods
- Accurate Calibrations
- Validity of a Cal
- ECal
- ECal User Characterization
- TRL Calibration
- Calibration Preferences
Measurement Errors 368
Modify Cal Kits 380
Power Calibration 399
Fixture Compensation 416
Port Extensions 430
Characterize Adaptor Macro 437
Calpod 446
Delta Match Calibration 452

4. Analyze Data
  Locate Data Using Markers 455
  Math & Memory Operations 470
  Equation Editor 475
  Use Limits to Test Devices 488

5. Output Data
  Save and Recall Data 493
  Drive Mapping 512
  Print 514

Programming 520

COM
  Commands
    Objects
      The Analyzer Object Model 521
      Application Object 524
      AuxTrigger Object 531
      BalancedMeasurement Object 533
      BalancedStimulus Object 535
      BalancedTopology Object 537
      CalFactorSegments Collection 539
      CalFactorSegmentsPMAR Collection 540
      Calibrator Object 541
      CalKit Object 546
      CalManager Object 548
      CalSet Object 551
      CalSets Collection 557
      CalStandard Object 559
      Capabilities Object 561
      Channel Object 564
      Channels Collection 572
SweptIMD Object 745
SweptIMDCal Object 749
TestSetControl Object 751
Trace Object 753
Traces Collection 755
Transform Object 756
TriggerSetup Object 758
VMC Type Object 760

Properties
AcceptTriggerBeforeArmed 763
AcquisitionDirection 764
AcquisitionMode 765
Active Cal Kit 766
Active Channel 767
Active ExtDev Property 768
Active Marker 769
Active Measurement 770
Active NAWindow 771
Active Trace 772
ActiveBackground Property 773
ActiveLabels Property 774
ActiveXAxisRange 775
ActiveXAxisRange(Conv) 776
ADCCaptureMode 778
ALCLevelingMode 779
AllowArbitrarySegments 781
Alternate Sweep 782
AmbientTemperature 783
AnalysisCWFreq Property 784
AnalysisEnable Property 785
AnalysisIsDiscreteFreq Property 786
AnalysisXAxis Property 787
Application 788
Arrange Windows 789
Attenuator Mode 790
Attenuator 791
AutoDetection Property 793
AutoIFBandWidth Property 794
AutoIFBWAdjustment
AutoIFGain Property
AutoOptimizePRF Property
AutoOrient
AutoOrientTuner Property
AutoPortExtConfig
AutoPortExtDCOffset
AutoPortExtLoss
AutoPortExtSearchStart
AutoPortExtSearchStop
AutoPortExtState
AutoPulseTiming Property
AutoSelectPulseGen Property
AuxiliaryTriggerCount
AuxTriggerScopelsGlobal
AvailableMeasurementClasses Property
AverageMode
Averaging Count
Averaging Factor
Averaging ON/OFF
AvoidSpurs
Background Property
BackOff Property
BackOffGain Property
BackOffP In Property
BackOffP out Property
BalancedMode
BalPort1PhaseOffset
BalPort1PowerOffset
BalPort1StartPhase
BalPort1StopPhase
BalPort2PhaseOffset
BalPort2PowerOffset
BalPort2StartPhase
BalPort2StopPhase
Bandwidth Target
Bandwidth Tracking
BB_BalPort1Negative
<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>BB_BalPort1Positive</td>
<td>834</td>
</tr>
<tr>
<td>BB_BalPort2Negative</td>
<td>835</td>
</tr>
<tr>
<td>BB_BalPort2Positive</td>
<td>836</td>
</tr>
<tr>
<td>BBalMeasurement Property</td>
<td>837</td>
</tr>
<tr>
<td>Begin Response</td>
<td>838</td>
</tr>
<tr>
<td>Begin Stimulus</td>
<td>839</td>
</tr>
<tr>
<td>BroadbandTuningSpan</td>
<td>840</td>
</tr>
<tr>
<td>Bucket Number</td>
<td>841</td>
</tr>
<tr>
<td>C0</td>
<td>842</td>
</tr>
<tr>
<td>C1</td>
<td>843</td>
</tr>
<tr>
<td>C2</td>
<td>844</td>
</tr>
<tr>
<td>C3</td>
<td>845</td>
</tr>
<tr>
<td>Cal Factor</td>
<td>846</td>
</tr>
<tr>
<td>Cal Type (applied)</td>
<td>847</td>
</tr>
<tr>
<td>Calibration Name</td>
<td>849</td>
</tr>
<tr>
<td>Calibration Port</td>
<td>850</td>
</tr>
<tr>
<td>Calibration TypeID</td>
<td>851</td>
</tr>
<tr>
<td>CalibrationFrequencies</td>
<td>853</td>
</tr>
<tr>
<td>CalKitType</td>
<td>854</td>
</tr>
<tr>
<td>CalKitType (FCA)</td>
<td>856</td>
</tr>
<tr>
<td>CalKitType</td>
<td>858</td>
</tr>
<tr>
<td>CalKitTypes Property</td>
<td>859</td>
</tr>
<tr>
<td>CalMethod</td>
<td>860</td>
</tr>
<tr>
<td>CalMethod (imd)</td>
<td>861</td>
</tr>
<tr>
<td>Cal Power</td>
<td>862</td>
</tr>
<tr>
<td>Center</td>
<td>863</td>
</tr>
<tr>
<td>Center (Meas)</td>
<td>864</td>
</tr>
<tr>
<td>Center Frequency</td>
<td>865</td>
</tr>
<tr>
<td>Channel Number</td>
<td>866</td>
</tr>
<tr>
<td>CharacterizationNumber</td>
<td>867</td>
</tr>
<tr>
<td>CharacterizeMixerOnly</td>
<td>868</td>
</tr>
<tr>
<td>CharFileName</td>
<td>869</td>
</tr>
<tr>
<td>CharMixerReverse</td>
<td>870</td>
</tr>
<tr>
<td>CitiContents</td>
<td>871</td>
</tr>
<tr>
<td>CitiFormat</td>
<td>872</td>
</tr>
<tr>
<td>CmnModeZConvPortImag</td>
<td>873</td>
</tr>
<tr>
<td>CmnModeZConvPortReal</td>
<td>874</td>
</tr>
<tr>
<td>CmnModeZConvPortZ0</td>
<td>875</td>
</tr>
</tbody>
</table>
DistanceMarkerUnit  957
Divisor  958
Do1PortEcal  959
Do2PortEcal  960
Domain  961
Driver Property  962
DspFpgaRevision Property  963
DspRevision Property  964
DUTTopology  965
Dwell Time  966
DwellPerPoint Property  967
ECALCharacterization (smc)  968
ECALCharacterization (vmc)  969
ECALCharacterizationEx  970
EcalID  971
ECALIsolation  972
ECALModuleNumberList  973
EcalOrientation  974
EcalOrientation1Port  976
EcalOrientation2Port  977
ECALPortMapEx  979
ElecDelay Medium  981
ElecDistanceDelay  982
ElecDistanceDelayUnit  983
Electrical Delay  984
Element  985
Elements  986
Embed4PortA  987
Embed4PortB  988
Embed4PortC  989
Embed4PortD  990
Embed4PortList  991
Embed4PortNetworkFilename  993
Embed4PortNetworkMode  994
Embed4PortState  995
Embed4PortTopology  996
Enable  997
Enabled  998
<table>
<thead>
<tr>
<th>Property Name</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ForceDeEmbedSensorAdapter Property</td>
<td>1039</td>
</tr>
<tr>
<td>Format (Marker)</td>
<td>1040</td>
</tr>
<tr>
<td>Format</td>
<td>950</td>
</tr>
<tr>
<td>FormatUnit</td>
<td>1042</td>
</tr>
<tr>
<td>Frequency GD Property</td>
<td>1043</td>
</tr>
<tr>
<td>Frequency</td>
<td>1044</td>
</tr>
<tr>
<td>FrequencyCenter</td>
<td>1045</td>
</tr>
<tr>
<td>FrequencyCenterCenter</td>
<td>1046</td>
</tr>
<tr>
<td>FrequencyCenterspan</td>
<td>1047</td>
</tr>
<tr>
<td>FrequencyCenterStart</td>
<td>1048</td>
</tr>
<tr>
<td>FrequencyCenterStop</td>
<td>1049</td>
</tr>
<tr>
<td>FrequencyList</td>
<td>1050</td>
</tr>
<tr>
<td>Frequency Offset Divisor</td>
<td>1051</td>
</tr>
<tr>
<td>Frequency Offset Frequency</td>
<td>1052</td>
</tr>
<tr>
<td>Frequency Offset Multiplier</td>
<td>1053</td>
</tr>
<tr>
<td>FrequencyOffsetRangeForCalComputations Property</td>
<td>1054</td>
</tr>
<tr>
<td>Frequency Offset Override To CW</td>
<td>1055</td>
</tr>
<tr>
<td>Frequency Offset State</td>
<td>1056</td>
</tr>
<tr>
<td>Gain Property</td>
<td>1057</td>
</tr>
<tr>
<td>GainLinear Property</td>
<td>1058</td>
</tr>
<tr>
<td>GainMax Property</td>
<td>1059</td>
</tr>
<tr>
<td>GainSaturation Property</td>
<td>1060</td>
</tr>
<tr>
<td>FrequencySpan</td>
<td>1061</td>
</tr>
<tr>
<td>Gate Shape</td>
<td>1062</td>
</tr>
<tr>
<td>Gate Type</td>
<td>1063</td>
</tr>
<tr>
<td>GPIBAddress</td>
<td>1064</td>
</tr>
<tr>
<td>GPIB Mode</td>
<td>1065</td>
</tr>
<tr>
<td>GPIBPortCount</td>
<td>1066</td>
</tr>
<tr>
<td>Grid Property</td>
<td>1067</td>
</tr>
<tr>
<td>GridLineType Property</td>
<td>1068</td>
</tr>
<tr>
<td>HandshakeEnable</td>
<td>1069</td>
</tr>
<tr>
<td>HasItem Property</td>
<td>1070</td>
</tr>
<tr>
<td>HighestOrderProduct</td>
<td>1071</td>
</tr>
<tr>
<td>ID</td>
<td>1072</td>
</tr>
<tr>
<td>IDString</td>
<td>1073</td>
</tr>
<tr>
<td>IF Bandwidth Option</td>
<td>1074</td>
</tr>
<tr>
<td>IF Bandwidth</td>
<td>1075</td>
</tr>
</tbody>
</table>
Interpolate Correction 1114
Interpolated 1115
InterpolateNormalization 1116
Interrupt 1117
Invert Property 1118
IOCConfiguration Property 1119
IOEnable Property 1120
IsContinuous 1121
IsECALModuleFoundEx 1122
IsFrequencyOffsetPresent 1123
IsHold 1124
IsMarkerOn 1125
IsolationAveragingIncrement 1126
IsOn 1127
IsReceiverStepAttenuatorPresent 1128
IsReferenceBypassSwitchPresent 1129
IsSParameter 1130
Item Property 1131
Items Property 1133
IterationNumber 1134
IterationsTolerance 1135
Kaiser Beta 1136
L0 1137
L1 1138
L2 1139
L3 1140
Label 1141
Label Testset 1142
LANConfiguration 1143
LastCalPassedTolerance 1144
LastLevelingAsSPC 1145
LastModified 1146
LevelingIFBW 1147
Limit Property 1148
LimitFrequency Property 1149
LimitTestFailed 1150
Limit Line Begin Stimulus 839
Limit Line End Stimulus 1151
Limit Line Begin Response 838
Limit Line End Response 1152
Limit Type 1153
Line Display 1154
LoadCharFromfile 1155
LoadPort 1156
LocalLockoutState 1157
Locator 1158
Lock Property 1159
LODeltaFound 1160
LODenominator 1161
LOFixedFrequency 1162
LOFrequencyDelta 1163
OBS_LogMagnitudeOffset 1164
LOName 1165
LONumerator 1167
LOPower 1168
LORangeMode 1169
Loss 1170
Loss (sourceCal) 1171
LOStage 1172
LOStartFrequency 1173
LOStartPower 1174
LOStopFrequency 1175
LOStopPower 1176
LXIDeviceIDState 1177
MagnitudeOffset 1178
MagnitudeSlopeOffset 1179
MainToneIFBandwidth 1180
Marker Annotation 1181
Marker Bucket Number 841
Marker Format (all) 1182
Marker Format (indiv) 1040
Marker Interpolate(all) 1184
Marker Interpolate (indiv) 1115
Marker Number 1185
Marker Position 1186
Marker Readout 1187
<table>
<thead>
<tr>
<th>Property</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum IF Frequency Property</td>
<td>1227</td>
</tr>
<tr>
<td>Minimum Number of Points</td>
<td>1228</td>
</tr>
<tr>
<td>Minimum Receiver Step Attenuator</td>
<td>1229</td>
</tr>
<tr>
<td>Minimum Source ALC Power</td>
<td>1230</td>
</tr>
<tr>
<td>Mixer Characterization File Property</td>
<td>1231</td>
</tr>
<tr>
<td>Mode</td>
<td>1232</td>
</tr>
<tr>
<td>Mode-TMSA</td>
<td>1233</td>
</tr>
<tr>
<td>Multiplier</td>
<td>1234</td>
</tr>
<tr>
<td>Name (Calset)</td>
<td>1235</td>
</tr>
<tr>
<td>Name (CalKit object)</td>
<td>1236</td>
</tr>
<tr>
<td>Name config</td>
<td>1237</td>
</tr>
<tr>
<td>Name element</td>
<td>1238</td>
</tr>
<tr>
<td>Name ExtDev Property</td>
<td>1239</td>
</tr>
<tr>
<td>Name FOM Range</td>
<td>1240</td>
</tr>
<tr>
<td>Name (meas)</td>
<td>1241</td>
</tr>
<tr>
<td>Name (PowerSensor)</td>
<td>1242</td>
</tr>
<tr>
<td>Name (trace)</td>
<td>1243</td>
</tr>
<tr>
<td>Network Filename</td>
<td>1244</td>
</tr>
<tr>
<td>Network Mode</td>
<td>1245</td>
</tr>
<tr>
<td>Network Port Map Property</td>
<td>1246</td>
</tr>
<tr>
<td>Noise Average Factor</td>
<td>1247</td>
</tr>
<tr>
<td>Noise Average State</td>
<td>1248</td>
</tr>
<tr>
<td>Noise Bandwidth</td>
<td>1249</td>
</tr>
<tr>
<td>Noise Receiver Property</td>
<td>1250</td>
</tr>
<tr>
<td>Noise Receiver Sweep Time Property</td>
<td>1251</td>
</tr>
<tr>
<td>Noise Gain</td>
<td>1253</td>
</tr>
<tr>
<td>Noise Source CalKit Type</td>
<td>1254</td>
</tr>
<tr>
<td>Noise Source Cold</td>
<td>1255</td>
</tr>
<tr>
<td>Noise Source Connector Type</td>
<td>1256</td>
</tr>
<tr>
<td>Noise Source State</td>
<td>1257</td>
</tr>
<tr>
<td>Noise Tuner</td>
<td>1258</td>
</tr>
<tr>
<td>Noise Tuner In</td>
<td>1259</td>
</tr>
<tr>
<td>Noise Tuner Out</td>
<td>1260</td>
</tr>
<tr>
<td>Nominal Incident Power State</td>
<td>1261</td>
</tr>
<tr>
<td>Normalize Point Property</td>
<td>1262</td>
</tr>
<tr>
<td>Normalize Point SMC Property</td>
<td>1263</td>
</tr>
<tr>
<td>Number (meas)</td>
<td>1264</td>
</tr>
<tr>
<td>Number</td>
<td>1265</td>
</tr>
<tr>
<td>Property</td>
<td>Page</td>
</tr>
<tr>
<td>---------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>PerformPowerCalibration</td>
<td>1310</td>
</tr>
<tr>
<td>Period</td>
<td>1311</td>
</tr>
<tr>
<td>Phase Offset</td>
<td>1312</td>
</tr>
<tr>
<td>PhaseAsFixture</td>
<td>1313</td>
</tr>
<tr>
<td>PhaseControlMode Property</td>
<td>1314</td>
</tr>
<tr>
<td>PhaseCorrectionData Property</td>
<td>1315</td>
</tr>
<tr>
<td>PhaseCorrectionEnabled Property</td>
<td>1316</td>
</tr>
<tr>
<td>PhaseIterationNumber Property</td>
<td>1317</td>
</tr>
<tr>
<td>PhaseParameter Property</td>
<td>1318</td>
</tr>
<tr>
<td>PhaseParameterModes Property</td>
<td>1319</td>
</tr>
<tr>
<td>PhaseReferencePort Property</td>
<td>1320</td>
</tr>
<tr>
<td>PhaseSwpAsFixture</td>
<td>1321</td>
</tr>
<tr>
<td>PhaseSwpState</td>
<td>1322</td>
</tr>
<tr>
<td>PhaseTolerance Property</td>
<td>1323</td>
</tr>
<tr>
<td>Pin Property</td>
<td>1324</td>
</tr>
<tr>
<td>PinOffset Property</td>
<td>1325</td>
</tr>
<tr>
<td>PMaxBackOff Property</td>
<td>1326</td>
</tr>
<tr>
<td>PMaxIn Property</td>
<td>1327</td>
</tr>
<tr>
<td>PMaxOut Property</td>
<td>1328</td>
</tr>
<tr>
<td>PointAveragingState</td>
<td>1329</td>
</tr>
<tr>
<td>Points Property</td>
<td>1330</td>
</tr>
<tr>
<td>PointSweepState</td>
<td>1331</td>
</tr>
<tr>
<td>Port 1</td>
<td>1332</td>
</tr>
<tr>
<td>Port 2</td>
<td>1333</td>
</tr>
<tr>
<td>Port 3</td>
<td>1334</td>
</tr>
<tr>
<td>Port1NoiseTunerSwitchPresetsToExternal</td>
<td>1335</td>
</tr>
<tr>
<td>Port2PdeembedCktModel</td>
<td>1336</td>
</tr>
<tr>
<td>Port2PdeembedState</td>
<td>1337</td>
</tr>
<tr>
<td>PortArbzImag</td>
<td>1338</td>
</tr>
<tr>
<td>PortArbzReal</td>
<td>1339</td>
</tr>
<tr>
<td>PortArbzState</td>
<td>1340</td>
</tr>
<tr>
<td>PortArbzZ0</td>
<td>1341</td>
</tr>
<tr>
<td>PortCatalog</td>
<td>1342</td>
</tr>
<tr>
<td>PortCLogic</td>
<td>1343</td>
</tr>
<tr>
<td>PortCMode</td>
<td>1344</td>
</tr>
<tr>
<td>PortCoupleToSystemMedia</td>
<td>1345</td>
</tr>
<tr>
<td>PortCoupleToSystemVelocity</td>
<td>1346</td>
</tr>
<tr>
<td>PortDelay</td>
<td>1347</td>
</tr>
</tbody>
</table>
PowerOnDuringRetraceMode
PowerSensorCalKitType Property
PowerSensorCalKitType
PowerSensorConnectorType
PowerSensorConnectorType Property
PowerStep
PowerSweepRetracePowerMode
PreciseTuningTolerance
PreferInternalTriggerOnChannelSingle
PreferInternalTriggerOnUnguidedCal
ResetPowerState Property
PulseMeasMode Property
RangeCount
rangeNumber
RatioedPowerCorrectionData Property
RatioedPowerCorrectionEnabled Property
R1 Input Path
Readings Per Point
ReadingsTolerance
ReadyForTriggerPolarity
ReadyForTriggerState
Receiver Attenuator
ReceiverCount
ReceiverRatio Property
ReceiverStepAttenuatorStepSize
Receive Port
RedTraceOnFail Property
ReduceIFBandwidth
ReferenceCalFactor
Reference Marker State
ReferenceReceiver
Reference Level
Reference Position
RemoteCalStoragePreference
ReportReceiverOverload
ResolutionBW
Reverse2PortAdapter Property
ReverseLinearPowerLevel
RFOffOnReceiverOverload 1426
SafeMode 1427
SafeSweepCoarsePowerAdjustment 1428
SafeSweepEnable 1429
SafeSweepFinePowerAdjustment 1430
SafeSweepFineThreshold 1431
SafeSweepMaximumLimit Property 1432
SaturationLevel Property 1433
SB_BalPortNegative 1434
SB_BalPortPositive 1435
SB_SEPort 1436
SBalMeasurement 1437
ScaleCouplingMethod Property 1438
ScaleCouplingState Property 1439
Scope 1440
Search Function 1441
SearchFailures Property 1442
SearchSummary Property 1443
SecurityLevel 1444
Segment Number 1445
SegmentCount Property 1446
SegmentFixedFrequency Property 1447
SegmentFixedPower Property 1449
SegmentIFBandwidth Property 1451
SegmentInputGreaterThanLO Property 1452
SegmentMixingMode Property 1454
SegmentPoints Property 1456
SegmentRangeMode Property 1457
SegmentStartFrequency Property 1459
SegmentState Property 1461
SegmentStopFrequency Property 1462
SelectPort 1464
SensorIndex Property 1465
SeparatePowerCal Property 1466
Show Statistics 1467
ShowProperties 1468
SICL 1469
SICLAddress 1470
Stage2Coefficients 1510
Stage2MaximumCoefficient 1511
Stage2MaximumCoefficientCount 1512
Stage2MaximumCoefficientSum 1513
Stage2MinimumCoefficientCount 1514
Stage3FilterType Property 1515
Stage3FilterTypes Property 1516
Stage3Parameter Property 1517
Stage3ParameterMaximum Property 1518
Stage3ParameterMinimum Property 1519
Stage3Parameters Property 1520
Standard Deviation 1521
Standard For Class 1522
StartFrequency_CS 1524
Start Frequency 1525
Start Power 1527
Start 1528
Start (Meas) 1529
StartPhase Property 1530
StartPowerEx 1531
StartRatioedPower Property 1532
State GPL Property 1533
State 1534
State rx 1536
State pulse 1537
Statistics Range 1538
StatusAsString 1539
Step Rise Time 1540
StepData 1541
StepTitle 1542
StimulusValues 1543
Stop Frequency 1544
Stop Frequency_CS 1546
Stop Power 1547
Stop 1548
Stop (Meas) 1549
StopPhase Property 1550
StopPowerEx 1551
<table>
<thead>
<tr>
<th>Variable</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tracking</td>
<td>1595</td>
</tr>
<tr>
<td>TrackingChannel</td>
<td>1596</td>
</tr>
<tr>
<td>TrackingEnable</td>
<td>1597</td>
</tr>
<tr>
<td>TrackingManualStepEnable</td>
<td>1598</td>
</tr>
<tr>
<td>TrackingStepIndex</td>
<td>1599</td>
</tr>
<tr>
<td>Transform Mode</td>
<td>1232</td>
</tr>
<tr>
<td>Trigger Delay</td>
<td>1600</td>
</tr>
<tr>
<td>TriggerInPolarity</td>
<td>1601</td>
</tr>
<tr>
<td>TriggerInType</td>
<td>1603</td>
</tr>
<tr>
<td>TriggerMode ExtDev Property</td>
<td>1605</td>
</tr>
<tr>
<td>TriggerOutDuration</td>
<td>1606</td>
</tr>
<tr>
<td>TriggerOutInterval</td>
<td>1607</td>
</tr>
<tr>
<td>TriggerOutPolarity</td>
<td>1608</td>
</tr>
<tr>
<td>TriggerOutPosition</td>
<td>1609</td>
</tr>
<tr>
<td>TriggerOutputEnabled</td>
<td>1610</td>
</tr>
<tr>
<td>TriggerPort Property</td>
<td>1611</td>
</tr>
<tr>
<td>TuningIF BW</td>
<td>1612</td>
</tr>
<tr>
<td>TuningMode</td>
<td>1613</td>
</tr>
<tr>
<td>TuningSweepInterval</td>
<td>1614</td>
</tr>
<tr>
<td>TwoPointGroupDelayAperture Property</td>
<td>1615</td>
</tr>
<tr>
<td>Trigger Mode</td>
<td>1616</td>
</tr>
<tr>
<td>Trigger Signal</td>
<td>1617</td>
</tr>
<tr>
<td>Trigger Type</td>
<td>1618</td>
</tr>
<tr>
<td>Type (calstd)</td>
<td>1619</td>
</tr>
<tr>
<td>Type_ts</td>
<td>1620</td>
</tr>
<tr>
<td>TZImag Property</td>
<td>1621</td>
</tr>
<tr>
<td>TZReal Property</td>
<td>1622</td>
</tr>
<tr>
<td>UnusedChannelNumbers</td>
<td>1623</td>
</tr>
<tr>
<td>USBPowerMeterCatalog</td>
<td>1624</td>
</tr>
<tr>
<td>UseCalWindow</td>
<td>1625</td>
</tr>
<tr>
<td>UsedChannelNumbers</td>
<td>1626</td>
</tr>
<tr>
<td>UseMultipleSensors Property</td>
<td>1627</td>
</tr>
<tr>
<td>Use Power Loss Segments</td>
<td>1628</td>
</tr>
<tr>
<td>Use Power Sensor Frequency Limits</td>
<td>1629</td>
</tr>
<tr>
<td>User Range</td>
<td>1630</td>
</tr>
<tr>
<td>User Range Max</td>
<td>1631</td>
</tr>
<tr>
<td>User Range Min</td>
<td>1633</td>
</tr>
<tr>
<td>UserDescriptionOfPNA</td>
<td>1635</td>
</tr>
</tbody>
</table>
UserName 1636
UserPresetEnable 1637
Valid 1638
ValidConnectorType 1639
Value element 1641
Values 1642
Velocity Factor 1643
View 1644
Visible 1645
WGCutoffFreq 1646
WideBandDectionState Property 1647
Width 1648
Window Number 1649
Window State 1650
XAxisAnnotation 1651
XAxis Point Spacing 1652
XAxisStart 1653
XAxisStop 1654
YAxisAnnotation 1655
YScale 1656
Z0 1657

Methods
Abort 1658
AbortPowerAcquisition 1659
Acquire Cal Standard 1660
Acquire Cal Standard2 1662
AcquireCalConfidenceCheckECALEx 1664
AcquirePowerReadingsEx 1665
AcquireStep 1667
Activate 1668
Activate Marker 1669
Activate Window 1670
Add (channels) 1671
Add (measurement) 1672
Add (naWindows) 1675
Add (PowerLossSegment) 1676
Add (PowerSensorCalFactorSegment) 1677
Add (segments) 1678
Add External Device Method 1679
Add GuidedPowerSensors Method 1680
Add Testset 1681
AddSegment Method 1682
Allow All Events 1683
AllowChannelToSweepDuringCalAcquisition 1684
Allow Event Category 1686
Allow Event Message 1687
Allow Event Severity 1688
Apply 1689
ApplyDeltaMatchFromCalSet 1690
ApplyPowerCorrectionValuesEx 1691
ApplySourcePowerCorrectionTo 1692
AssignSourceToRole 1693
AutoOrient Method 1694
AutoPortExtMeasure 1696
AutoPortExtReset 1697
Autoscale 1698
Averaging Restart 1699
Build Hybrid Kit 1700
Calculate Error Coefficients 1701
Calculate 1702
Change Parameter 1704
CheckPower 1707
Clear 1708
Clear fifo 1709
Close CalSet 1710
ComputeErrorTerms 1711
ConfigEnhancedNB2 1712
ConfigEnhancedNBIFAtten 1714
ConfigNarrowBand3 1715
ConfigurationFile 1717
Configurations 1718
Configure 1719
Continuous Sweep 1720
Copy 1721
CopyToChannel 1723
OBS_Create SParameter Method 1724
CreateCalSet 1725
CreateCustomCal 1726
CreateCustomCalEx 1727
CreateCustomMeasurementEx 1729
CustomCalConfiguration Method 914
Create Measurement 1734
DataToMemory 1738
Deembed Method 1739
Delete 1741
Delete Marker 1742
Delete All Markers 1743
DeleteAllSegments Method 1744
DeleteCalSet 1745
DeleteConfiguration 1746
DeleteSegment Method 1747
Delete ShortCut 1748
Delta Marker 930
Disallow All Events 1749
DiscardChanges 1750
DisplayNAWindowDuringCalAcquisition 1751
DisplayOnlyCalWindowDuringCalAcquisition 1752
Do Print 1753
DoECAL1PortEx 1754
DoECAL2PortEx 1755
DoneCalConfidenceCheckECAL 1757
DoReceiverPowerCal 1758
DoResponseCal Method 1760
Embed Method 1761
EnumerateCalSets 1763
EnumerateItems 1764
Execute 1765
Execute Shortcut 1766
GenerateGlobalDeltaMatchSequence 1767
GenerateErrorTerms 1768
GenerateSteps 1769
GetAllSegments 1770
GetCalKitTypeString Method 1772
GetCompatibleCalKits Method 1773
Get Paired Data
Get Port
Get PortC Data
Get PortNumber
GetRaw2DData
GetRaw2DDataIm
GetRaw2DDataRe
GetRxLevelingConfiguration
GetSourceByRole
GetSourceRoles
Get Reference Marker
Get Required Eterm Names
Get Scalar
Get Shortcut
Get SnPData
Get SnpDataWithSpecifiedPorts
GetSourcePowerCalDataEx
GetSourcePowerCalDataScalarEx
Get Standard
Get Standard By String
Get Standard Complex
Get Standard Complex By String
Get StandardsList
Get Standard List2
Get StandardsForClass
Get StepDescription
GetSupportedALCModes
Get Test Result
Get Trace Statistics
Get X-Axis Values
Get XAxisValues (Meas)
Get X-axis Values Variant
Has CalType
Hold
Hold (All Chans)
ImportDataSet Method
ImportLibrary Method
Initialize
Show Table 2021
Show Title Bars 2022
Show Toolbar 2023
Single 2024
Store 2026
StoreConfiguration 2027
StoreTheme Method 2028
StringToNACalClass 2029
StringtoNAErrorTerm2 2031
SweepOnlyCalChannelDuringCalAcquisition 2032
TestsetCatalog 2033
UserPreset 2034
UserPresetLoadFile 2035
UserPresetSaveState 2036
Write Data 2037
Write Raw 2038
WriteSnpFileWithSpecifiedPorts 2040

Events
OnCalEvent 2042
OnChannelEvent 2044
OnDisplayEvent 2046
OnHardwareEvent 2048
OnMeasurementEvent 2050
OnSCPIEvent 2052
OnSystemEvent 2054
OnUserEvent 2056

Examples
C Example 2057
CalSet_Examples 2062
Getting Trace Data from the Analyzer 2064
Perform a Guided Cal using COM 2067
Perform a Source Power Cal 2071
Perform an Unguided Cal using COM 2074
Perform an Unknown Thru or TRL Cal 2077
Perform Global Delta Match Cal 2079
Perform a Guided Cal with CSharp 2080
Perform an ECal 2084
Perform an ECal User Characterization 2087
<table>
<thead>
<tr>
<th>Task</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Perform a Comprehensive Guided 2-Port Cal</td>
<td>2092</td>
</tr>
<tr>
<td>Perform an ECAL Confidence Check</td>
<td>2097</td>
</tr>
<tr>
<td>Writing Cal Set Data using COM</td>
<td>2099</td>
</tr>
<tr>
<td>Upload a Source Power Cal</td>
<td>2101</td>
</tr>
<tr>
<td>Upload Segment Table</td>
<td>2103</td>
</tr>
<tr>
<td>Create and Cal an SMC Measurement</td>
<td>2107</td>
</tr>
<tr>
<td>Create and Cal a VMC Measurement</td>
<td>2110</td>
</tr>
<tr>
<td>Create an SMC Fixed Output Measurement</td>
<td>2114</td>
</tr>
<tr>
<td>Create a Segmented Sweep for Mixers</td>
<td>2116</td>
</tr>
<tr>
<td>Use Existing Power Cal for SMC</td>
<td>2119</td>
</tr>
<tr>
<td>Create a Pulsed Measurement</td>
<td>2122</td>
</tr>
<tr>
<td>Create a Balanced Measurement</td>
<td>2124</td>
</tr>
<tr>
<td>Create a PMAR Device and Measurement</td>
<td>2128</td>
</tr>
<tr>
<td>Create a Wideband Pulsed Measurement using the PNA-X</td>
<td>2130</td>
</tr>
<tr>
<td>Create an IM Spectrum Measurement</td>
<td>2132</td>
</tr>
<tr>
<td>Create an iTMSA Measurement</td>
<td>2134</td>
</tr>
<tr>
<td>Create_and_Cal_a_Gain_Compression_Measurement</td>
<td>2135</td>
</tr>
<tr>
<td>Create and Cal a GCX Measurement</td>
<td>2140</td>
</tr>
<tr>
<td>Create and Cal a NoiseFigure Measurement</td>
<td>2144</td>
</tr>
<tr>
<td>Create and Cal an NFX Measurement</td>
<td>2148</td>
</tr>
<tr>
<td>Create and Cal an IMD Measurement</td>
<td>2153</td>
</tr>
<tr>
<td>ENR File Management Example</td>
<td>2157</td>
</tr>
<tr>
<td>Events with C</td>
<td>2159</td>
</tr>
<tr>
<td>FOM Examples</td>
<td>2161</td>
</tr>
<tr>
<td>Limit Line Testing Example with COM</td>
<td>2163</td>
</tr>
<tr>
<td>Modify Display Colors</td>
<td>2164</td>
</tr>
<tr>
<td>E5091Testset Control</td>
<td>2165</td>
</tr>
<tr>
<td>Errors and the SCPIStringParser Object</td>
<td>2166</td>
</tr>
<tr>
<td>External Testset Control</td>
<td>2168</td>
</tr>
<tr>
<td>PathConfiguration Example</td>
<td>2170</td>
</tr>
<tr>
<td>PNA-X Create a Pulsed Measurement</td>
<td>2172</td>
</tr>
<tr>
<td>Setup Basic Measurements</td>
<td>2176</td>
</tr>
<tr>
<td>Setup Compression Marker</td>
<td>2178</td>
</tr>
<tr>
<td>SetUp Embedded LO Measurement</td>
<td>2179</td>
</tr>
<tr>
<td>Setup FastCW and FIFO</td>
<td>2181</td>
</tr>
<tr>
<td>Setup Noise Figure Port Mapping</td>
<td>2183</td>
</tr>
<tr>
<td>Setup Phase Control</td>
<td>2185</td>
</tr>
</tbody>
</table>
Setup PNOP and PSAT Marker Search 2187
Setup Receiver Leving 2189
Show Custom Window during Calibration 2191
Events Example 2193
Upload_Segment_Table_in_C 2194
Using C# 2201

Concepts
Configure for COM-DCOM Programming 2203
COM Fundamentals 2208
Getting a Handle to an Object 2212
Collections in the Analyzer 2215
COM Data Types 2216
PNA Automation Interfaces 2218
Working with the Analyzer's Events 2220
Read and Write Calibration Data using COM 2224
C and the COM Interface 2226
Using .NET 2229

SCPI
Commands
SCPI Command Tree 2231
Common Commands 2234
Abort 2237
Calculate
Correction 2238
Custom 2246
Data 2252
Equation 2260
Filter 2265
Format 2271
FSimulator 2274
Function 2276
GCData 2281
GCM meas 2286
GroupDelay 2290
Limit 2293
Marker 2301
Math 2321
Mixer 2323
Normalize 2324
Offset  2327
Parameter  2330
RData  2340
Smoothing  2342
Transform  2345
X Values  2354
Control  2355
CSET  2367
Display  2371
Format  2395
Hardcopy  2398
Initiate  2411
Memory  2414
Output  2427
Route  2429
Sense
  Average  2430
  Bandwidth  2433
  Class  2435
  Correction  2436
    Cal Kit  2465
    Cal Stds  2478
    Cal Sets  2500
    Extensions  2515
    Guided Cal  2529
    IMD  2555
    Session  2561
    SMC  2567
    VMC  2579
  Couple  2590
  FOM  2592
  FOMSegment  2602
  Frequency  2613
  Gain Compression  2617
  IF  2633
  IF (PNA-X)  2639
  IMD  2649
Create and Cal a VMC Measurement 2959
Create and Cal an IMD Measurement 2963
Create and Cal an NFX Measurement 2967
Create and Cal an SMC Measurement 2972
Create and Cal Multiple SMC Channels 2976
Create New Cal Kit using SCPI 2979
Custom Power Meter Driver 2985
ECALConfidence Check using SCPI 2989
Establish a VISA Session 2992
External Test Set using SCPI wLink 2994
Getting and Putting Data using SCPI 2996
GPIB Pass Through 2998
GPIB using Visual C 2999
Load Eterms into Cal Sequence 3003
Modify a Calibration Kit using SCPI 3004
Guided 2-Port or 4-Port Cal 3006
Perform a Simple Source Power Cal 3010
Perform a Source and Receiver Power Cal using SCPI 3012
Perform a Source Power Cal with TWO Sensors 3021
Perform an ECal User Characterization 3023
Perform an Unguided Cal on a 4-Port PNA using SCPI 3030
Perform Global Delta Match Cal 3039
Perform Guided 1Port 3040
Perform Guided 2-Port Comprehensive Cal 3042
Perform Guided ECal 3049
Perform Guided Mechanical Cal 3051
Perform Guided TRL Calibration 3053
Perform an Unguided 1-Port Cal on Port 2 3055
Perform Unguided 2-Port Mech Cal 3057
Perform Unguided ECAL 3059
Perform Unknown Thru or TRL Cal 3061
Setup FastCW and FIFO 3063
Setup Markers 3066
Setup Noise Figure Port Mapping 3068
Setup Phase Control 3070
Setup PNOP and PSAT Markers 3072
Setup RxLeveling 3074
Shut Down the PNA Remotely  3223
LXI Compliance  121
Using Macros  279

Applications

Frequency Converter Application (Opts 082 / 083)
  FCA Overview  3224
  SMC Measurements  3238
  VMC Measurements  3255
  How to make a VMC Fixed Output Measurement  3270
  How to make an SMC Fixed Output Measurement  3278
  Embedded LO (Opt 084)  3285
  Characterize Adaptor Macro  437
  SMC with a Booster Amp  3291

Frequency Offset (Opt 080)
  Frequency Offset (Option 080)  3294
  Frequency Converting Device Measurements  3304
  Frequency Offset Calibration  3305
  Conversion Loss  3307
  Conversion Compression  3311
  Isolation  3313
  Harmonic Distortion  3316
  Return Loss and VSWR  3318

Antenna Features  3320
Gain Compression Application (Opt 086)  3325
Gain Compression for Converters  3350
Gain Compression Cal  3358
iTMSA (Opt 460)  3362
Noise Figure (Opt 029)  3371
Noise Figure on Converters  3392
Noise Figure Cal  3407

Swept IMD and IM Spectrum (Opt 087)
  Swept IMD and IM Spectrum Concepts  3415
  Swept IMD  3424
  IM Spectrum  3438
  Swept IMDx for Converters  3447
  IMx Spectrum for Converters  3459

Time Domain (Opt 010)  3467

Integrated Pulse Application (Opt 008)  3483
Pulsed Application (Opt H08) 3491
WideBand Pulsed App 3505

Networking the PNA
  Drive Mapping 512
  Connecting the PNA to a PC 3506
  Easy vs Secure Configuration 3509
  Changing Network Client 3510

Product Support
  Troubleshoot the PNA 3511
  List of Error Messages 3515
  About Error Messages 3577
  Accessories 3581
  USB to GPIB Adapter 3588
  Firmware Update 3591
  PNA Configurations and Options 3597
  Option Enable 3611
  Instrument Calibration 3615
  Other Resources 3617
  SCPI Errors 3618
  Technical Support 3629

Diagnostic Tools and Adjustments
  3.8 GHz Frequency Adjust 3634
  10 MHz Reference Adjust 3636
  Display Test 3638
  LO Power Adjust 3639
  Offset LO Power Adjustment 3641
  Operators Check 3642
  Option H11 Verification 3645
  Phase-Lock IF Gain Adjust 3648
  System Verification 3650
  Source Adjustment 3661
  Receiver Cal 3664
  Receiver Display 3667

IF Access
  IF Access User Interface Settings 3668
  External Test Head Configuration 3676
  PNA-X IF Path Configuration 3668
  N5264A Measurement Receiver 3688
System Settings
Configure an External Device 3692
Configure a Power Meter As Receiver 3700
E5091 Test Set Control 3710
Error Messages 3577
External Testset Control 3714
Display Colors 3726
Frequency Blanking 108
Interface Control 3154
Mechanical Devices 3730
PNA Preferences 114
Power Limit and Power Offset 3733
System Impedance 3736
Synchronize an External PSG Source 3738
Rear Panel IO Connectors
   Interface Control 3154
   Auxiliary IO connector 3160
   External TestSet IO Connector 3166
   Material Handler IO Connector 3172
   Pulse IO (PNA-X) 3185
   Power IO (PNA-X) 3187
Tutorials
App Notes 3743
Network Analyzer Basics 3745
Connector Care 3746
ESD Protection 3757
Measurements
   Absolute Output Power 3758
   AM-PM Conversion 3760
   Amplifier Measurements 3765
   Antenna Measurements 3768
   Balanced Measurements 3771
   Complex Impedance 3777
   Comparing the PNA Delay Functions 3780
   Deviation from Linear Phase 3782
   External Source Control 3738
   Gain and Flatness 3785
   Gain Compression 3788
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group Delay</td>
<td>3793</td>
</tr>
<tr>
<td>High-Gain Amplifier Measurements</td>
<td>3800</td>
</tr>
<tr>
<td>High-Power Measurements using a PNA</td>
<td>3802</td>
</tr>
<tr>
<td>High Power Measurements using a PNA-X</td>
<td>3803</td>
</tr>
<tr>
<td>Phase Measurements</td>
<td>3805</td>
</tr>
<tr>
<td>Reverse Isolation</td>
<td>3808</td>
</tr>
<tr>
<td>Reflection Measurements</td>
<td>3811</td>
</tr>
<tr>
<td>Time Domain Measurements</td>
<td>3467</td>
</tr>
<tr>
<td>Specifications</td>
<td>3815</td>
</tr>
<tr>
<td>Glossary</td>
<td>3817</td>
</tr>
</tbody>
</table>
What's New in PNA Code Version A.09.42

- New N522x Models

**Standard Measurements** - available on all Models / Options

- Drag a trace to another window

**Application Enhancements**

- Noise Figure Cal using a Power Meter
- Copy Channels on all Applications
- **"Src 2 out Port 2"** factory configuration on PNA-X Opt 423.
- IMD f2 Tone using External Source / Combiner
- IMD and IM Spectrum Tone Power Leveling settings
- IMD, IMDx, and IM Spectrum "Min" and "Max" parameters
- Use a Power Table with mmWave SMC Measurements
- mmWave; Mixer mode - 2-port test set on 4-port PNA
- Guided Power Cal for SMC
- ESG and PSG Sources for Phase Control

*See New 9.42 Programming Commands*

This version includes features from all versions listed below.

To check your PNA code version, click **Help**, then **About Network Analyzer**

---

What's New in PNA Code Version A.09.33

**New Options**

- Source Phase Control - Opt 088

**Application Enhancements**

- FCA Update - Opt 082, and 083

**Standard Measurements** - available on all Models / Options
- Security for External Sources
- 2-Port and 4-Port Fixture Extrapolation and Reverse Ports.
- Phase Coherent “R over R” measurements
- Use Multiple Power Sensors for Guided Power Cal
- Perform Source Power Cal with PMAR Device

**Tip - Do you access the same PNA dialog often?**
Your Favorites are always two keystrokes away. [Learn how.](#)

---

**What's New in PNA Code Version A.09.31**

- Support for N1913A and N1914A Power Meters

---

**What's New in PNA Code Version A.09.30**

- N5247A - 67 GHz PNA-X

**Application Enhancements**

- Gain Compression on Converters (GCX)
- Support for Dual-Stage Converters in all Apps

**Standard Measurements** - available on all Models / Options

- Enhanced S-parameter Power Cal
- Marker Display enhancements
- Perform Source Power Cal at multiple power levels
- IF Gain Setting
- Receiver Overload/Compression Warning and Power OFF Preferences
- Confirm changes on Meas Class dialog
- DSP Version 5

---

**What's New in PNA Code Version A.09.22**
• Use any PNA-X Ports with Noise Figure Opt 028.

What's New in PNA Code Version A.09.20

Application Enhancements

• Integrated Pulse Measurements (Opt 008)
• Noise Figure using Standard PNA Receiver (Opt 028)
• Noise Figure on N5244A/45A (Opt H29)
• Edge and Level Trigger in Pulse
• Exclude SC12 Sweep for SMC (Opt 082/083)
• Include Phase with SMC (PNA-X with Opt 083)
• Fixturing in Apps
• Max Output Power for GCA (Opt 086)

Standard Measurements - available on all Models / Options

• Mechanical Device conflicts cause Channel Block (NOT Channel Hold)
• PSAT Marker and Power Normal Operating Point Marker
• Group Delay Aperture Setting
• Active Background Display Color
• Solid or Dotted Grid Lines
• Point Sweep on PNA “C” Models
• Fixture Power Compensation
• Sweep Delay
• Uncertainty equations using RSS Computations
• Preset Power Preference Setting
• Use Last Receiver Leveling Correction for SPC
• Data Save Enhancements
  • Recall .SNP files to view as trace
• "Save Data As" Dialog
• Save Balanced Data as SNP files
• Characterize Adaptor Macro Rev. A.02.10
• Reverse S2P
• Load the PNA Power Loss Table from an existing S2P file

What's New in PNA Code Version A.09.10

• Noise Figure on Converters (NFX)
• AgileUpdate for Customer Releases
• New Help, About Capabilities

What's New in PNA Code Version A.09.00

• External Device Configuration
• Power Meter as Receiver (PMAR)
• Power Offsets and Limits
• Display and Print Colors
• Scale Coupling
• Mechanical Device Settings
• Device side USB
• Increased Number of Channels to 200
• ECal User Chars Saved to PNA Disk Memory

Application Enhancements

• GCA Compression Analysis
• GCA Compression from Saturation
• Receiver Leveling on IMD, FCA, and GCA
• Embedded LO on SMC and IMDx
• Limited Port Mapping on IMD
- Point Averaging on FCA and IMD
- SMC Measurements with mmWave Modules
- mmWave Measurements with no Test Set

---

**What's New in PNA Code Version A.08.60**

- New 40 GHz and 50 GHz PNA-X Models

---

**What's New in PNA Code Version A.08.55**

- IMDx (Swept and Spectrum) for Converters
- ADC Measurements in a Gain Compression channel.
- New 13.5 GHz PNA-X Model

---

**What's New in PNA Code Version A.08.50**

- Fast Antenna Features for the PNA-X
- Receiver Leveling
- Phase Sweep in iTMSA
- Up to 25 User Macros
- Gain Compression Marker
- Port Extensions enhancements
- Electrical Delay enhancements
- Extra Security enhancement
- Save *.CSV and *.MDF File Types
- Noise Figure App enhancements:
  - Scalar Noise Figure measurement
  - Incident Noise Power parameters
- MM Module enhancements:
- **Power Level Control**
- Supports **iTMSA**
- **Max point count to 32,001**
- **Faster Power Sweeps**

### What's New in PNA Code Version A.08.35
- **New N5264A model**

### What's New in PNA Code Version A.08.33
- **IMD Application** (Opt 087)
- **Fast Sweep Mode**
- **New Equation Editor functions**
- **20001 Segments in Power Loss Table**
- **Data Format Units**
- **Marker=>CW Freq Function**
- **Up to 12 User Characterizations**
- **Characterize Adaptor Macro 2.0**

### What's New in PNA Code Version A.08.20
- **iTMSA**
- **User Preferences dialog**
- **Uncoupled Power Sweep**
- **Equation Editor Import Functions**
- **Wider Traces Preference**
- **24 Traces per Window**
- **LXI Compliance**
- **GCA Enhancements**
FCA - selectable ports
Support for N5261A and N5262A MM test sets
cXL Code Translation Software - Quick Link

What's New in PNA Code Version A.08.00

- Noise Figure Application (Opt 029)
- Gain Compression Application (Opt 086)
- 'Sweep' Trigger Mode
- Custom Cal Window settings (remote only)
- New Equation Editor Functions
- Minimum Number of Points = 1

Last modified:

9/28/06 Cross-browser
END-USER LICENSE AGREEMENT FOR MICROSOFT SOFTWARE
MICROSOFT WINDOWS XP PROFESSIONAL EDITION SERVICE PACK 3

IMPORTANT-READ CAREFULLY: This End-User License Agreement ("EULA") is a legal agreement between you (either an individual or a single entity) and Microsoft Corporation or one of its affiliates ("Microsoft") for the Microsoft software that accompanies this EULA, which includes computer software and may include associated media, printed materials, "online" or electronic documentation, and Internet-based services ("Software"). An amendment or addendum to this EULA may accompany the Software. YOU AGREE TO BE BOUND BY THE TERMS OF THIS EULA BY INSTALLING, COPYING, OR OTHERWISE USING THE SOFTWARE. IF YOU DO NOT AGREE, DO NOT INSTALL, COPY, OR USE THE SOFTWARE; YOU MAY RETURN IT TO YOUR PLACE OF PURCHASE FOR A FULL REFUND, IF APPLICABLE.

1. GRANT OF LICENSE. Microsoft grants you the following rights provided that you comply with all terms and conditions of this EULA:

1.1 Installation and use. You may install, use, access, display and run one copy of the Software on a single computer, such as a workstation, terminal or other device ("Workstation Computer"). The Software may not be used by more than two (2) processors at any one time on any single Workstation Computer.

1.2 Mandatory Activation. The license rights granted under this EULA are limited to the first thirty (30) days after you first install the Software unless you supply information required to activate your licensed copy in the manner described during the setup sequence of the Software. You can activate the Software through the use of the Internet or telephone; toll charges may apply. You may also need to reactivate the Software if you modify your computer hardware or alter the Software. There are technological measures in this Software that are designed to prevent unlicensed use of the Software. Microsoft will use those measures to confirm you have a legally licensed copy of the Software. If you are not using a licensed copy of the Software, you are not allowed to install the Software or future Software updates. Microsoft will not collect any personally identifiable information from your Workstation Computer during this process.

1.3 Device Connections. You may permit a maximum of ten (10) computers or other electronic devices (each a "Device") to connect to the Workstation Computer to utilize one or more of the following services of the Software: File Services, Print Services, Internet Information Services, Internet Connection Sharing and telephony services. The ten connection maximum includes any indirect connections made through "multiplexing" or other software or hardware which pools or aggregates connections. This ten connection maximum does not apply to other uses of the Software, such as synchronizing data between a Device and the Workstation Computer, provided only one user uses, accesses, displays or runs the Software at any one time. This Section 1.3 does not grant you rights to access a Workstation Computer Session from any Device. A "Session" means any use of the Software that enables functionality similar to that available to an end user who is interacting with the Workstation Computer through any combination of input, output and display peripherals.

1.4 Remote Desktop/Remote Assistance/NetMeeting. The Software contains Remote Desktop, Remote Assistance, and NetMeeting technologies that enable the Software or applications installed on the Workstation Computer (sometimes referred to as a host device) to be accessed remotely from other Devices. You may use the Software’s Remote Desktop feature (or other software which provides similar functionality for a similar purpose) to access a Workstation Computer Session from any Device provided you acquire a separate Software license for that Device. As an exception to this rule, the person who is the single primary user of the Workstation Computer may access a Workstation Computer Session from any Device without acquiring an additional Software license for that Device. When you are using Remote Assistance or NetMeeting (or other software which provides similar functionality for a similar purpose) you may share a Session with other users without any limit on the number of Device connections and without acquiring additional licenses for the Software. For Microsoft and non-Microsoft applications, you should consult the license agreement accompanying the applicable software or contact the applicable licensor to determine whether use of the software with Remote Desktop, Remote Assistance, or NetMeeting is permitted without an additional license.

1.5 Storage/Network Use. You may also store or install a copy of the Software on a storage device, such as a
network server, used only to install or run the Software on your other Workstation Computers over an internal network; however, you must acquire and dedicate an additional license for each separate Workstation Computer on or from which the Software is installed, used, accessed, displayed or run. Except as otherwise permitted by the NetMeeting and Remote Assistance features described above, a license for the Software may not be shared or used concurrently on different Workstation Computers.

2. AUTOMATIC INTERNET-BASED SERVICES. The Software features described below are enabled by default to connect via the Internet to Microsoft computer systems automatically, without separate notice to you. You consent to the operation of these features, unless you choose to switch them off or not use them. Microsoft does not obtain personal information through any of these features. For more information about these features, please see your Software documentation, the Microsoft online support site, or the privacy statement at http://go.microsoft.com/fwlink/?LinkId=25243.

2.1 Windows Update Features. If you connect hardware to your Workstation Computer, it may not have the drivers needed to communicate with that hardware. The Software’s update feature can obtain the correct drivers from Microsoft and install them on your device. You can switch this update feature off.

2.2 Web Content Features. Under the Software’s default configuration, if you are connected to the Internet, several features of the Software are enabled by default to retrieve content from Microsoft computer systems and display it to you. When you activate such a feature, it uses standard Internet protocols, which transmit the type of operating system, browser and language code of your Workstation Computer to the Microsoft computer system so that the content can be viewed properly from your Workstation Computer. These features only operate when you activate them, and you may choose to switch them off or not use them. Examples of these features include Windows Catalog, Search Assistant, and the Headlines and Search features of Help and Support Center.

2.3 Digital Certificates. The Software uses digital certificates based on the x.509 standard. These digital certificates confirm the identity of Internet users sending x.509 standard encrypted information. The software retrieves certificates and updates certificate revocation lists. These security features operate only when you use the Internet.

2.4 Auto Root Update. The Auto Root Update feature updates the list of trusted certificate authorities. You can switch off the Auto Root Update feature.

2.5 Windows Media Player. Some features of Windows Media Player automatically contact Microsoft computer systems if you use Windows Media Player or specific features of it: features that (A) check for new codecs if your Workstation Computer does not have the correct ones for content you attempt to play (this feature may be switched off), and (B) check for new versions of Windows Media Player (this feature will operate only when you are using Windows Media Player).

2.6 Windows Media Digital Rights Management. Content providers are using the digital rights management technology for Windows Media contained in this Software (“WM-DRM”) to protect the integrity of their content (“Secure Content”) so that their intellectual property, including copyright, in such content is not misappropriated. Portions of this Software and third party applications such as media players use WM-DRM to play Secure Content (“WM-DRM Software”). If the WM-DRM Software’s security has been compromised, owners of Secure Content (“Secure Content Owners”) may request that Microsoft revoke the WM-DRM Software’s right to copy, display and/or play Secure Content. Revocation does not alter the WM-DRM Software’s ability to play unprotected content. A list of revoked WM-DRM Software is sent to your Workstation Computer whenever you download a license for Secure Content from the Internet. Microsoft may, in conjunction with such license, also download revocation lists onto your Workstation Computer on behalf of Secure Content Owners. Secure Content Owners may also require you to upgrade some of the WMDRM components in this Software (“WM-DRM Upgrades”) before accessing their content. When you attempt to play such content, WM-DRM Software built by Microsoft will notify you that a WM-DRM Upgrade is required and then ask for your consent before the WM-DRM Upgrade is downloaded. WM-DRM Software built by third parties may do the same. If you decline the upgrade, you will not be able to access content that requires the WM-DRM Upgrade; however, you will still be able to access unprotected content and Secure Content that does not require the upgrade. WM-DRM features that access the Internet, such as acquiring new licenses and/or performing a required WM-DRM Upgrade, can be switched off. When these features are switched off, you will still be able to play Secure Content if you have a valid license for such content already stored on your
3. RESERVATION OF RIGHTS AND OWNERSHIP. Microsoft reserves all rights not expressly granted to you in this EULA. The Software is protected by copyright and other intellectual property laws and treaties. Microsoft or its suppliers own the title, copyright, and other intellectual property rights in the Software. The Software is licensed, not sold.

4. LIMITATIONS ON REVERSE ENGINEERING, DECOMPILATION, AND DISASSEMBLY. You may not reverse engineer, decompile, or disassemble the Software, except and only to the extent that such activity is expressly permitted by applicable law notwithstanding this limitation.

5. NO RENTAL/COMMERCIAL HOSTING. You may not rent, lease, lend or provide commercial hosting services with the Software.

6. LINKS TO THIRD PARTY SITES. You may link to third party sites through the use of the Software. The third party sites are not under the control of Microsoft, and Microsoft is not responsible for the contents of any third party sites, any links contained in third party sites, or any changes or updates to third party sites. Microsoft is not responsible for webcasting or any other form of transmission received from any third party sites. Microsoft is providing these links to third party sites to you only as a convenience, and the inclusion of any link does not imply an endorsement by Microsoft of the third party site.

7. ADDITIONAL SOFTWARE/SERVICES. This EULA applies to updates, supplements, add-on components, product support services, or Internet-based services components, of the Software that you may obtain from Microsoft after the date you obtain your initial copy of the Software, unless you accept updated terms or another agreement governs. Microsoft reserves the right to discontinue any Internet-based services provided to you or made available to you through the use of the Software.

8. UPGRADES. To use Software identified as an upgrade, you must first be licensed for the software identified by Microsoft as eligible for the upgrade. After upgrading, you may no longer use the software that formed the basis for your upgrade eligibility.

9. NOT FOR RESALE SOFTWARE. Software identified as “Not For Resale” or “NFR,” may not be sold or otherwise transferred for value, or used for any purpose other than demonstration, test or evaluation.

10. ACADEMIC EDITION SOFTWARE. To use Software identified as “Academic Edition” or “AE,” you must be a “Qualified Educational User.” For qualification-related questions, please contact the Microsoft Sales Information Center/One Microsoft Way/Redmond, WA 98052-6399 or the Microsoft subsidiary serving your country.

11. NOTICES REGARDING THE MPEG-4 VISUAL STANDARD. The Software includes MPEG-4 visual decoding technology. This technology is a format for data compression of video information. For this technology, MPEG LA, L.L.C. requires this notice: USE OF THIS PRODUCT IN ANY MANNER THAT COMPLIES WITH THE MPEG-4 VISUAL STANDARD IS PROHIBITED, EXCEPT FOR USE DIRECTLY RELATED TO (A) DATA OR INFORMATION (i) GENERATED BY AND OBTAINED WITHOUT CHARGE FROM A CONSUMER NOT THEREBY ENGAGED IN A BUSINESS ENTERPRISE, AND (ii) FOR PERSONAL USE ONLY; AND (B) OTHER USES SPECIFICALLY AND SEPARATELY LICENSED BY MPEG LA, L.L.C. If you have questions regarding this notice, please contact MPEG LA, L.L.C., 250 Steele Street, Suite 300, Denver, Colorado 80206; Telephone 303 331.1880; FAX 303 331.1879; http://www.mpegla.com.

12. EXPORT RESTRICTIONS. You acknowledge that the Software is subject to U.S. export jurisdiction. You agree to comply with all applicable international and national laws that apply to the Software, including the U.S. Export Administration Regulations, as well as end-user, enduse, and destination restrictions issued by U.S. and other governments. For additional information see http://www.microsoft.com/exporting.

13. END USER PROOF OF LICENSE. If you acquired the Software on a compact disc or other media, a genuine Microsoft “Proof of License” label with a genuine copy of the software identifies a licensed copy of the Software. To be valid, the label must appear on Microsoft software packaging. If you receive the label separately, it is invalid. You should keep the packaging that has the label on it to prove that you are licensed to use the Software.

14. SOFTWARE TRANSFER. Internal. You may move the Software to a different Workstation Computer. After the
transfer, you must completely remove the Software from the former Workstation Computer. Transfer to Third Party.
The initial user of the Software may make a one-time permanent transfer of this EULA and Software to another end
user, provided the initial user retains no copies of the Software. This transfer must include the Software and the
Proof of License label. The transfer may not be an indirect transfer, such as a consignment. Prior to the transfer,
the end user receiving the Software must agree to all the EULA terms.

15. TERMINATION. Without prejudice to any other rights, Microsoft may terminate this EULA if you fail to comply
with the terms and conditions of this EULA. In such event, you must destroy all copies of the Software and all of its
component parts.

16. NOTICE REGARDING SECURITY. To help protect against breaches of security and malicious software,
periodically back up your data and system information, use security features such as firewalls, and install and use
security updates.

17. LIMITED WARRANTY FOR SOFTWARE ACQUIRED IN THE US AND CANADA. Microsoft warrants that the
Software will perform substantially in accordance with the accompanying materials for a period of ninety (90) days
from the date of receipt. If an implied warranty or condition is created by your state/jurisdiction and federal or
state/provincial law prohibits disclaimer of it, you also have an implied warranty or condition, BUT ONLY AS TO
DEFECTS DISCOVERED DURING THE PERIOD OF THIS LIMITED WARRANTY (NINETY DAYS). AS TO ANY
DEFECTS DISCOVERED AFTER THE NINETY-DAY PERIOD, THERE IS NO WARRANTY OR CONDITION OF
ANY KIND. Some states/jurisdictions do not allow limitations on how long an implied warranty or condition lasts, so
the above limitation may not apply to you. Any supplements or updates to the Software, including without limitation,
you (if any) service packs or hot fixes provided to you after the expiration of the ninety day Limited Warranty period
are not covered by any warranty or condition, express, implied or statutory. LIMITATION ON REMEDIES; NO
CONSEQUENTIAL OR OTHER DAMAGES. Your exclusive remedy for any breach of this Limited Warranty is as
set forth below. Except for any refund elected by Microsoft, YOU ARE NOT ENTITLED TO ANY DAMAGES,
INCLUDING BUT NOT LIMITED TO CONSEQUENTIAL DAMAGES, if the Software does not meet Microsoft’s
Limited Warranty, and, to the maximum extent allowed by applicable law, even if any remedy fails of its essential
purpose. The terms of Section 19 ("Exclusion of Incidental, Consequential and Certain Other Damages") are also
incorporated into this Limited Warranty. Some states/jurisdictions do not allow the exclusion or limitation of
incidental or consequential damages, so the above limitation or exclusion may not apply to you. This Limited
Warranty gives you specific legal rights. You may have other rights which vary from state/jurisdiction to
state/jurisdiction. YOUR EXCLUSIVE REMEDY. Microsoft’s and its suppliers’ entire liability and your exclusive
remedy for any breach of this Limited Warranty or for any other breach of this EULA or for any other liability relating
to the Software shall be, at Microsoft’s option from time to time exercised subject to applicable law, (a) return of the
amount paid (if any) for the Software, or (b) repair or replacement of the Software, that does not meet this Limited
Warranty and that is returned to Microsoft with a copy of your receipt. You will receive the remedy elected by
Microsoft without charge, except that you are responsible for any expenses you may incur (e.g. cost of shipping the
Software to Microsoft). This Limited Warranty is void if failure of the Software has resulted from accident, abuse,
misapplication, abnormal use or a virus. Any replacement Software will be warranted for the remainder of the
original warranty period or thirty (30) days, whichever is longer, and Microsoft will use commercially reasonable
efforts to provide your remedy within a commercially reasonable time of your compliance with Microsoft’s warranty
remedy procedures. Outside the United States or Canada, neither these remedies nor any product support services
offered by Microsoft are available without proof of purchase from an authorized international source. To exercise
your remedy, contact: Microsoft, Attn. Microsoft Sales Information Center/One Microsoft Way/Redmond, WA
98052-6399, or the Microsoft subsidiary serving your country.

18. DISCLAIMER OF WARRANTIES. The Limited Warranty that appears above is the only express warranty made
to you and is provided in lieu of any other express warranties or similar obligations (if any) created by any
advertising, documentation, packaging, or other communications. Except for the Limited Warranty and to the
maximum extent permitted by applicable law, Microsoft and its suppliers provide the Software and support services
(if any) AS IS AND WITH ALL FAULTS, and hereby disclaim all other warranties and conditions, whether express,
IMPLIED OR STATUTORY, INCLUDING, BUT NOT LIMITED TO, ANY (IF ANY) IMPLIED WARRANTIES, DUTIES OR Conditions OF
MERCHANTABILITY, OF FITNESS FOR A PARTICULAR PURPOSE, OF RELIABILITY OR AVAILABILITY, OF ACCURACY OR COMPLETENESS OF
responses, of results, of workmanlike effort, of lack of viruses, and of lack of negligence, all with regard to the Software, and the provision of or failure to provide support or other services, information, software, and related content through the Software or otherwise arising out of the use of the Software. ALSO, THERE IS NO WARRANTY OR CONDITION OF TITLE, QUIET ENJOYMENT, QUIET POSSESSION, CORRESPONDENCE TO DESCRIPTION OR NONINFRINGEMENT WITH REGARD TO THE SOFTWARE.

19. EXCLUSION OF INCIDENTAL, CONSEQUENTIAL AND CERTAIN OTHER DAMAGES. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL MICROSOFT OR ITS SUPPLIERS BE LIABLE FOR ANY SPECIAL, INCIDENTAL, PUNITIVE, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, BUT NOT LIMITED TO, DAMAGES FOR LOSS OF PROFITS OR CONFIDENTIAL OR OTHER INFORMATION, FOR BUSINESS INTERRUPTION, FOR PERSONAL INJURY, FOR LOSS OF PRIVACY, FOR FAILURE TO MEET ANY DUTY INCLUDING OF GOOD FAITH OR OF REASONABLE CARE, FOR NEGLIGENCE, AND FOR ANY OTHER PECUNIARY OR OTHER LOSS WHATSOEVER) ARISING OUT OF OR IN ANY WAY RELATED TO THE USE OF OR INABILITY TO USE THE SOFTWARE, THE PROVISION OF OR FAILURE TO PROVIDE SUPPORT OR OTHER SERVICES, INFORMATION, SOFTWARE, AND RELATED CONTENT THROUGH THE SOFTWARE OR OTHERWISE ARISING OUT OF THE USE OF THE SOFTWARE, OR OTHERWISE UNDER OR IN CONNECTION WITH ANY PROVISION OF THIS EULA, EVEN IF MICROSOFT OR ANY SUPPLIER HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

20. LIMITATION OF LIABILITY AND REMEDIES. Notwithstanding any damages that you might incur for any reason whatsoever (including, without limitation, all damages referenced herein and all direct or general damages in contract or anything else), the entire liability of Microsoft and any of its suppliers under any provision of this EULA and your exclusive remedy hereunder (except for any remedy of repair or replacement elected by Microsoft with respect to any breach of the Limited Warranty) shall be limited to the greater of the actual damages you incur in reasonable reliance on the Software up to the amount actually paid by you for the Software or US$5.00. The foregoing limitations, exclusions and disclaimers (including Sections 17, 18, and 19) shall continue in full force and effect even if any remedy fails its essential purpose.

21. U.S. GOVERNMENT LICENSE RIGHTS. All Software provided to the U.S. Government pursuant to solicitations issued on or after December 1, 1995 is provided with the commercial license rights and restrictions described elsewhere herein. All Software provided to the U.S. Government pursuant to solicitations issued prior to December 1, 1995 is provided with "Restricted Rights" as provided for in FAR, 48 CFR 52.227-14 (JUNE 1987) or DFAR, 48 CFR 252.227-7013 (OCT 1988), as applicable.

22. APPLICABLE LAW. If you acquired this Software in the United States, this EULA is governed by the laws of the State of Washington. If you acquired this Software in Canada, unless expressly prohibited by local law, this EULA is governed by the laws in force in the Province of Ontario, Canada; and, in respect of any dispute which may arise hereunder, you consent to the jurisdiction of the federal and provincial courts sitting in Toronto, Ontario. If you acquired this Software in the European Union, Iceland, Norway, or Switzerland, then local law applies. If you acquired this Software in any other country, then local law may apply.

23. ENTIRE AGREEMENT; SEVERABILITY. This EULA (including any addendum or amendment to this EULA which is included with the Software) is the entire agreement between you and Microsoft relating to the Software and the support services (if any) and they supersede all prior or contemporaneous oral or written communications, proposals and representations with respect to the Software or any other subject matter covered by this EULA. To the extent the terms of any Microsoft policies or programs for support services conflict with the terms of this EULA, the terms of this EULA shall control. If any provision of this EULA is held to be void, invalid, unenforceable or illegal, the other provisions shall continue in full force and effect.

Si vous avez acquis votre produit Microsoft au CANADA, la garantie limitée suivante vous concerne :

GARANTIE LIMITÉE

Microsoft garantit que le Logiciel fonctionnera conformément aux documents inclus pendant une période de 90
jours suivant la date de réception. Si une garantie ou condition implicite est créée par votre État ou votre territoire et qu'une loi fédérale ou provinciale ou État en interdit le déni, vous jouissez également d'une garantie ou condition implicite, MAIS UNIQUEMENT POUR LES DÉFAUTS DÉCOUVERTS DURANT LA PÉRIODE DE LA PRÉSENTE GARANTIE LIMITÉE (QUATRE-VINGT-DIX JOURS). IL N'Y A AUCUNE GARANTIE OU CONDITION DE QUELQUE NATURE QUE CE SOIT QUANT AUX DÉFAUTS DÉCOUVERTS APRÈS CETTE PÉRIODE DE QUATRE-VINGTDIX JOURS. Certains États ou territoires ne permettent pas de limiter la durée d'une garantie ou condition implicite de sorte que la limitation ci-dessus peut ne pas s'appliquer à vous. Tous les suppléments ou toutes les mises à jour relatifs au Logiciel, notamment, les ensembles de services ou les réparations à chaud (le cas échéant) qui vous sont fournis après l'expiration de la période de quatre-vingt-dix jours de la garantie limitée ne sont pas couverts par quelque garantie ou condition que ce soit, expresse ou implicite. LIMITATION DES RECURS; ABSENCE DE DOMMAGES INDIRECTS OU AUTRES. Votre recours exclusif pour toute violation de la présente garantie limitée est décrit ci-après. Sauf pour tout remboursement au choix de Microsoft, si le Logiciel ne respecte pas la garantie limitée de Microsoft et, dans la mesure maximale permise par les lois applicables, même si tout recours n'atteint pas son but essentiel, VOUS N'AVEZ DROIT À AUCUNS DOMMAGES, NOTAMMENT DES DOMMAGES INDIRECTS. Les modalités de la clause «Exclusion des dommages accessoires, indirects et de certains autres dommages » sont également intégrées à la présente garantie limitée. Certains États ou territoires ne permettent pas l'exclusion ou la limitation des dommages indirects ou accessoires de sorte que la limitation ou l'exclusion ci-dessus peut ne pas s'appliquer à vous. La présente garantie limitée vous donne des droits légaux spécifiques. Vous pouvez avoir d'autres droits qui peuvent varier d'un territoire ou d'un État à un autre. VOTRE RECURS EXCLUSIF. L'obligation intégrale de Microsoft et de ses fournisseurs et votre recours exclusif seront, selon le choix de Microsoft de temps à autre sous réserve de toute loi applicable, a) le remboursement du prix payé, le cas échéant, pour le Logiciel ou b) la réparation ou le remplacement du Logiciel qui ne respecte pas la présente garantie limitée et qui est retourné à Microsoft avec une copie de votre reçu. Vous recevrez la compensation choisie par Microsoft, sans frais, sauf que vous êtes responsable des dépenses que vous pourriez engager (p. ex., les frais d'envoi du Logiciel à Microsoft). La présente garantie limitée est nulle si la défectuosité du Logiciel est causée par un accident, un usage abusif, une mauvaise application, un usage anormal ou un virus. Tout Logiciel de remplacement sera garanti pour le reste de la période de garantie initiale ou pendant trente (30) jours, selon la plus longue entre ces deux périodes. À l'extérieur des États-Unis ou du Canada, ces recours ou l'un quelconque des services de soutien technique offerts par Microsoft ne sont pas disponibles sans preuve d'achat d'une source internationale autorisée. Pour exercer votre recours, vous devez communiquer avec Microsoft et vous adresser au Microsoft Sales Information Center/One Microsoft Way/Redmond, WA 98052-6399, ou à la filiale de Microsoft de votre pays.

DÉNI DE GARANTIES. La garantie limitée mentionnée ci-dessus constitue la seule garantie expresse qui vous est donnée et remplace toutes autres garanties expressées (s’il en est) mentionnées dans un document ou sur un emballage. Sauf en ce qui a trait à la garantie limitée et dans la mesure maximale permise par les lois applicables, le Logiciel et les services de soutien technique (le cas échéant) sont fournis TELS QUELS ET AVEC TOUS LES DÉFAUTS par Microsoft et ses fournisseurs, lesquels par les présentes dénient toutes autres garanties et conditions expressées, implicites ou en vertu de la loi, notamment (le cas échéant) les garanties, devoirs ou conditions implicites de qualité marchande, d'adaptation à un usage particulier, d'exactitude ou d'exhaustivité des réponses, des résultats, des efforts déployés selon les règles de l'art, d'absence de virus et de négligence, le tout à l'égard du Logiciel et de la prestation des services de soutien technique ou de l’omission d’une telle prestation. PAR AILLEURS, IL N'Y A AUCUNE GARANTIE OU CONDITION QUANT AU TITRE DE PROPRIÉTÉ, À LA JOUSSOUTÉ OU LA POSSESSION PAISIBLE, À LA CONCORDANCE À UNE DESCRIPTION NI QUANT À UNE ABSENCE DE CONTREFAÇON CONCERNANT LE LOGICIEL. EXCLUSION DES DOMMAGES ACCESSOIRES, INDIRECTS ET DE CERTAINS AUTRES DOMMAGES. DANS LA MESURE MAXIMALE PERMISE PAR LES LOIS APPLICABLES, EN AUCUN CAS MICROSOFT OU SES FOURNISSEURS NE SERONT RESPONSABLES DES DOMMAGES SPÉCIAUX, CONSÉCUTIFS, ACCESSOIRES OU INDIRECTS DE QUELQUE NATURE QUE CE SOIT (NOTAMMENT, LES DOMMAGES À L'ÉGAR DU MANQUE À GAGNER OU DE LA DIVULGATION DE RENSEIGNEMENTS CONFIDENTIELS OU AUTRES, DE LA PERTE D'EXPLOITATION, DE BLESSURES CORPORELLES, DE LA VIOLATION DE LA VIE PRIVÉE, DE L'OMISSION DE REMPLIR TOUT DEVOR, Y COMPRIS D'AGIR DE BONNE FOI OU D'EXERCER UN SOIN RAISONNABLE, DE LA NÉGLIGENCE ET DE
TOUTE AUTRE PERTE PÉCUNIAIRE OU AUTRE PERTE DE QUELQUE NATURE QUE CE SOIT) SE RAPPORTANT DE QUELQUE MANIÈRE QUE CE SOIT À L’UTILISATION DU LOGICIEL OU À L’INCAPACITÉ DE S’EN SERVIR, À LA PRESTATION OU À L’OMISSION D’UNE TELLE PRESTATION DE SERVICES DE SOUTIEN TECHNIQUE OU AUTREMENT AUX TERMES DE TOUTE DISPOSITION DU PRÉSENT EULA OU RELATIVEMENT À UNE TELLE DISPOSITION, MÊME EN CAS DE FAUTE, DE DÉLIT CIVIL (Y COMPRIS LA NÉGLIGENCE), DE RESPONSABILITÉ STRICTE, DE VIOLATION DE CONTRAT OU DE VIOLATION DE GARANTIE DE MICROSOFT OU DE TOUT FOURNISSEUR ET MÊME SI MICROSOFT OU TOUT FOURNISSEUR A ÊTÉ AVISÉ DE LA POSSIBILITÉ DE TELS DOMMAGES. LIMITATION DE RESPONSABILITÉ ET RECOURS. Malgré les dommages que vous puissiez subir pour quelque motif que ce soit (notamment, tous les dommages susmentionnés et tous les dommages directs ou généraux), l’obligation intégrale de Microsoft et de l’un ou l’autre de ses fournisseurs aux termes de toute disposition du présent EULA et votre recours exclusif à l’égard de tout ce qui précède (sauf en ce qui concerne tout recours de réparation ou de remplacement choisi par Microsoft à l’égard de tout manquement à la garantie limitée) se limite au plus élevé entre les montants suivants : le montant que vous avez réellement payé pour le Logiciel ou 5,00 $US. Les limites, exclusions et dénis qui précèdent (y compris les clauses cidessus), s’appliquent dans la mesure maximale permise par les lois applicables, même si tout recours n’atteint pas son but essentiel. La présente Convention est régie par les lois de la province d’Ontario, Canada. Chacune des parties à la présente reconnaît irrévocablement la compétence des tribunaux de la province d’Ontario et consent à instituer tout litige qui pourrait découler de la présente auprès des tribunaux situés dans le district judiciaire de York, province d’Ontario. Au cas où vous auriez des questions concernant cette licence ou que vous désiriez vous mettre en rapport avec Microsoft pour quelque raison que ce soit, veuillez contacter la succursale Microsoft desservant votre pays, dont l’adresse est fournie dans ce produit, ou écrivez à : Microsoft Sales Information Center, One Microsoft Way, Redmond, Washington 98052-6399.

The following MICROSOFT GUARANTEE applies to you if you acquired this Software in any other country:

Statutory rights not affected - The following guarantee is not restricted to any territory and does not affect any statutory rights that you may have from your reseller or from Microsoft if you acquired the Software directly from Microsoft. If you acquired the Software or any support services in Australia, New Zealand or Malaysia, please see the “Consumer rights” section below.

The guarantee - The Software is designed and offered as a general-purpose software, not for any user’s particular purpose. You accept that no Software is error free and you are strongly advised to back-up your files regularly. Provided that you have a valid license, Microsoft guarantees that a) for a period of 90 days from the date of receipt of your license to use the Software or the shortest period permitted by applicable law it will perform substantially in accordance with the written materials that accompany the Software; and b) any support services provided by Microsoft shall be substantially as described in applicable written materials provided to you by Microsoft and Microsoft support engineers will use reasonable efforts, care and skill to solve any problem issues. In the event that the Software fails to comply with this guarantee, Microsoft will either (a) repair or replace the Software or (b) return the price you paid. This guarantee is void if failure of the Software results from accident, abuse or misapplication. Any replacement Software will be guaranteed for the remainder of the original guarantee period or 30 days, whichever period is longer. You agree that the above guarantee is your sole guarantee in relation to the Software and any support services.

Exclusion of All Other Terms - To the maximum extent permitted by applicable law and subject to the guarantee above, Microsoft disclaims all warranties, conditions and other terms, either express or implied (whether by statute, common law, collaterally or otherwise) including but not limited to implied warranties of satisfactory quality and fitness for particular purpose with respect to the Software and the written materials that accompany the Software. Any implied warranties that cannot be excluded are limited to 90 days or to the shortest period permitted by applicable law, whichever is greater.

Limitation of Liability - To the maximum extent permitted by applicable law and except as provided in the Microsoft Guarantee, Microsoft and its suppliers shall not be liable for any damages whatsoever (including without limitation, damages for loss of business profits, business interruption, loss of business information or other pecuniary loss) arising out of the use or inability to use the Software, even if Microsoft has been advised of the possibility of such damages. In any case Microsoft’s entire liability under any provision of this Agreement shall be limited to the
amount actually paid by you for the Software. These limitations do not apply to any liabilities that cannot be excluded or limited by applicable laws.

Consumer rights - Consumers in Australia, New Zealand or Malaysia may have the benefit of certain rights and remedies by reason of the Trade Practices Act and similar state and territory laws in Australia, the Consumer Guarantees Act in New Zealand and the Consumer Protection Act in Malaysia in respect of which liability cannot lawfully be modified or excluded. If you acquired the Software in New Zealand for the purposes of a business, you confirm that the Consumer Guarantees Act does not apply. If you acquired the Software in Australia and if Microsoft breaches a condition or warranty implied under any law which cannot lawfully be modified or excluded by this agreement then, to the extent permitted by law, Microsoft's liability is limited, at Microsoft's option, to: (i) in the case of the Software: a) repairing or replacing the Software; or b) the cost of such repair or replacement; and (ii) in the case of support services: a) re-supply of the services; or b) the cost of having the services supplied again.

Should you have any questions concerning this EULA, or if you desire to contact Microsoft for any reason, please use the address information enclosed in this Software to contact the Microsoft subsidiary serving your country or visit Microsoft on the World Wide Web at http://www.microsoft.com.
PNA User Accounts and Passwords

Important: When the PNA power is switched on, it AUTOMATICALLY logs into Windows using the default user name and password. You do NOT need to log on. This gives anyone full access to the analyzer. The following steps can be taken to increase security of your PNA.

- Require users to logon when the PNA computer is turned ON - Learn how to enable this feature
- Setup individual accounts on the PNA with varying level of access - Learn how to Add or Change User Accounts and Passwords

Please read about Anti-virus protection for your PNA

Existing User Accounts

The following user accounts already exist on new PNAs.

- **Default User Account** The Default User Account is created by Windows and cannot be deleted. We recommend you change the password and, if desired, the user name. DO NOT FORGET YOUR NEW PASSWORD. If you choose to require users to logon when the PNA is turned ON, you will not be able to start your PNA without it.
  - Beginning in Sept. 2010, PNAs are shipped from the factory with the default:
    - User name (not case sensitive): A-<model number>-<last 5 digits of the serial number> (for example: A-N5242A-12345)
    - Password (case sensitive): agilent<last 5 digits of the serial number> (for example: agilent12345)
  - Beginning in April 2004, PNAs were shipped from the factory with the default:
    - User name (not case sensitive): PNA-Admin
    - Password (case sensitive): agilent
  - Before April 2004, PNAs were shipped from the factory with the default:
    - User name: Administrator
    - Password: either tsunami or left blank.

- **Agilent Account** This Administrator account is created by Agilent for service purposes. Each PNA has a unique password for this account. Although allowed by Windows, please do not delete this account.

- **Guest Account** This account allows anyone to type in any name, without password, and gain limited access to the PNA files. This account is created by Windows and cannot be deleted. It can be renamed. This account is turned OFF when the PNA is shipped.
Notes

- When connecting the PNA to the Internet, do NOT setup an Administrator account without a password. Although allowed by Windows, Internet viruses look for, and exploit, this condition.
- You can create as many user accounts as you like.
- The user name is not case sensitive. The password IS case sensitive.
- The PNA local policies are set so that, if logon is required, you must retype the user name (and password) every time. Do not change the local policies on the PNA.

How to Require Users to Logon when the PNA Computer is turned ON.

How do I know which Operating System I have?

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the Windows taskbar, click Start, then Settings, then Control Panel.</td>
<td>On the Windows taskbar, click Start, then Run.</td>
</tr>
<tr>
<td>Double click Users and Passwords.</td>
<td>Type control userpasswords2 then click OK.</td>
</tr>
<tr>
<td>Check Users must enter a user name and password to use this computer.</td>
<td>Check Users must enter a user name and password to use this computer.</td>
</tr>
</tbody>
</table>

To turn this function OFF, perform the same procedure, but clear the checkbox. The account that is selected when the checkbox is cleared is the account that is automatically logged on when the PNA is turned ON.

Add or Change User Accounts and Passwords

If the analyzer is in a secure environment, you can setup PNA users by name and grant various levels of access. This is particularly important when the PNA is remotely controlled or accessed over LAN.

You can designate a person as the administrator and then configure the PNA to allow others to use it with reduced permissions. That is, other people can be signed on to use the analyzer but they will not have the ability to perform all of the administrative functions that you can as the administrator.

Note: To connect and install a new ECal module, you must be logged on with an Administrator account. A user account with limited access will NOT allow an ECal module driver to be loaded. Learn more.

How to add or change a user account and password

How do I know which Operating System I have?

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>In the analyzer System menu, point to Configure, and click Control Panel.</td>
<td>Click Start, then point to Settings, then click Control Panel.</td>
</tr>
</tbody>
</table>
In the **Control Panel** window, scroll down and select the **Users and Passwords** application.

Click **User Accounts**

On the **Users** tab, if the **Add** button appears dimmed, select the **Users must enter a user name and password to use this computer** check box near the top of the window.

Follow the prompts to:
- Change an account
- **Create a new account** (see below)
- Change the way users log on or off (with password)

**CAUTION:** Although allowed by Windows, do NOT allow an Administrator account without a password. Internet viruses look for, and exploit, this condition.

Click **Add** to enter the information for yourself or for another user.

In the **User name** box, enter a user name for the user. In the **Full name** box, enter the full name of the user.

In the **Description** box, enter a description for the user. Then, click **Next**.

In the **Password** box, have the user type a password. Have the user retype the password in the **Confirm password** box. Then, click **Next**.

Select the level of access that you wish to grant this user.

**Note:** Standard users and restricted users are **NOT** able to switch GPIB modes and install firmware.

There are several other levels of security that you may grant in the **Other** list. A description of each of these other levels is displayed beneath the **Other** box when it is selected. Then, click **Finish**.

In the **Users for this computer** box, validate the user name and security level group of the user.

**Note:**
A user account with limited access will **NOT** allow an ECal module driver to be loaded. [Learn more](#).

However, limited access users **ARE** able to switch GPIB modes and install firmware.
<table>
<thead>
<tr>
<th>If you want this user to be able to use the network analyzer without entering their password each use, clear the <strong>Users must enter a user name and password to use this computer</strong> check box. Click <strong>OK</strong>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>When the <strong>Automatically Log On</strong> window is displayed, have the new user type their password in the <strong>Password</strong> box and have them retype the password in the <strong>Confirm Password</strong> box.</td>
</tr>
<tr>
<td>Click <strong>OK</strong> to complete this user addition.</td>
</tr>
<tr>
<td>In the <strong>File</strong> menu, click <strong>Close</strong> to close the Control Panel.</td>
</tr>
</tbody>
</table>

Last Modified:

- **14-Sep-2010**  Updated default account
- **8-Apr-2008**  Added Logon notes
PNA Computer Properties

The PNA uses a personal computer and a Windows operating system. The following are common tasks that you may need to perform on the PNA computer.

- View or change Full Computer Name
- Check IP Address
- Check the amount of RAM
- Check CPU Speed
- Set Time and Date
- Internal and External Speakers

Other Administrative Task Topics

View or change Full Computer Name

Your PNA has a unique computer name that identifies it on a network. To view or change the computer name, you must first minimize the PNA application.

How do I know which Operating System I have?

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the desktop, right-click My Computer</td>
<td>On the desktop, right-click My computer Icon</td>
</tr>
<tr>
<td>Click Properties</td>
<td>Click Properties</td>
</tr>
<tr>
<td>Click the Network Identification tab at the top of the dialog box</td>
<td>Click the Computer Name tab at the top of the dialog box</td>
</tr>
<tr>
<td>Click Properties</td>
<td>Click Change next to &quot;..rename this computer..&quot; message</td>
</tr>
<tr>
<td>Type your new Computer Name</td>
<td>Type your new Computer Name</td>
</tr>
</tbody>
</table>

Note: To add your computer to a domain, or to set up the networking configuration, contact your company's I.T. department. This setup is custom for each company.

To restore the PNA application, click PNA Analyzer in the task bar at the bottom of the screen.

Check IP Address

If your PNA is connected to a LAN, you can view the IP address and other networking information.
1. **Minimize the PNA** application

2. Click **Start**, then **Run**

3. Type **cmd**, then click **OK**

4. At a DOS prompt, type **ipconfig /all**

---

**Check the amount of RAM**

Random Access Memory (RAM) is the amount of working memory in your computer. The PNA application can require up to 512 MB of RAM depending on the settings you use. If your PNA is operating slowly when you have more than four windows open or if you routinely use more than 1601 data points, you may need to upgrade to 512 MB.

The amount of RAM in your PNA may limit your ability to upgrade firmware. See [http://na.tm.agilent.com/pna/firmware/PNA_support_matrix.doc](http://na.tm.agilent.com/pna/firmware/PNA_support_matrix.doc)

To view the amount of PNA RAM, you must first minimize the PNA application.

---

**How do I know which Operating System I have?**

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the desktop, right-click <strong>My Computer</strong></td>
<td>On the desktop, right-click <strong>My computer Icon</strong></td>
</tr>
<tr>
<td>Click <strong>Properties</strong></td>
<td>Click <strong>Properties</strong></td>
</tr>
<tr>
<td>Click the <strong>General</strong> tab at the top of the dialog box</td>
<td>Click the <strong>General</strong> tab at the top of the dialog box</td>
</tr>
<tr>
<td>The amount of RAM appears at the bottom of the window.</td>
<td>The amount of RAM appears at the bottom of the window.</td>
</tr>
</tbody>
</table>

To restore the PNA application, click **PNA Analyzer** in the task bar at the bottom of the screen.

---

**Check CPU Speed**

The speed of the PNA processor (CPU) is a factor in determining how quickly the PNA processes data. Also, the CPU speed in your PNA may limit your ability to upgrade firmware. See [http://na.tm.agilent.com/pna/firmware/PNA_support_matrix.doc](http://na.tm.agilent.com/pna/firmware/PNA_support_matrix.doc)

You can see which CPU is in your PNA by comparing your PNA rear-panel with the images at [http://na.tm.agilent.com/pna/cputype.html](http://na.tm.agilent.com/pna/cputype.html).

Or, beginning with PNA Rev. A.09.10, on the PNA click **Help**, then **About Network Analyzer**. Learn more.

Or, you can do the following to check your PNA CPU speed:

1. **Minimize the PNA** application.

2. Then do the following:
### How do I know which Operating System I have?

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the desktop, right-click <strong>My Computer</strong></td>
<td>On the desktop, right-click <strong>My computer Icon</strong></td>
</tr>
<tr>
<td><strong>Click Manage</strong></td>
<td><strong>Click Properties</strong></td>
</tr>
<tr>
<td>Open <strong>System Tools</strong> folder, then click <strong>System Information</strong>.</td>
<td>Click the <strong>General</strong> tab at the top of the dialog box</td>
</tr>
<tr>
<td><strong>Click System Summary</strong>.</td>
<td>The CPU speed appears near the bottom of the window</td>
</tr>
<tr>
<td>After refreshing, the CPU speed appears at the end of the <strong>Processor</strong> entry.</td>
<td></td>
</tr>
</tbody>
</table>

To restore the PNA application, click **PNA Analyzer** in the task bar at the bottom of the screen.

See [PNA configurations](#) to learn if you can upgrade your PNA CPU.

### Set Time and Date

Both Windows 2000 and XP

To set the time and date on your PNA, you must first minimize the PNA application.

1. Move the cursor to the lower corner of the screen
2. When the taskbar appears, double-click on the displayed time. This opens the **Date/Time Properties** dialog box.
3. Change the date, time, and time zone as appropriate.

To restore the PNA application, click **PNA Analyzer** in the task bar at the bottom of the screen.

### Internal Speaker

There is an internal sound card and speaker in the PNA. However, there is no audio output jack.

There may be times when you might want to control the speaker volume or turn the speaker OFF, such as when the PNA is generating errors. See [Learn more about errors](#).

To control the PNA speaker volume:

1. Press **System**, then **Service**, then **Utilities**, then **Speaker Volume**
2. Enter a value between 0 (speaker OFF) and 100 (highest volume)
<table>
<thead>
<tr>
<th>Date</th>
<th>Change Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>17-Jun-2011</td>
<td>Modified speaker volume</td>
</tr>
<tr>
<td>30-Nov-2009</td>
<td>Added Help, About note.</td>
</tr>
<tr>
<td>26-Nov-2008</td>
<td>Added external speaker note</td>
</tr>
<tr>
<td>12-Mar-2008</td>
<td>Added link to support site</td>
</tr>
<tr>
<td>20-Sep-2007</td>
<td>Added speaker OFF</td>
</tr>
</tbody>
</table>
Run Error Check and Disk Defragmenter

When the PNA is shutdown unexpectedly or power is removed without first shutting down, large amounts of Hard Disk Drive space is rendered unusable. If shutdown in this manner enough times, the PNA could become unstable and no longer work.

This Hard Disk Drive space can be recovered by first running Windows Error-checking to find and correct errors on the disk, and then the Disk Defragmenter to recover Hard Disk Drive space. These programs should be run routinely, about every 1 to 4 weeks, depending on how often the PNA is unexpectedly shutdown.

To learn more about Disk Defragmenter, see the Windows Help file.

Follow this procedure to run these programs:

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the desktop, double-click My Computer</td>
<td>On the desktop, double-click My Computer</td>
</tr>
<tr>
<td>Select Local Disk (C:)</td>
<td>Select System OS</td>
</tr>
<tr>
<td>Click File, then Properties</td>
<td>Click File, then Properties</td>
</tr>
<tr>
<td>Click the Tools tab</td>
<td>Click the Tools tab</td>
</tr>
</tbody>
</table>

Error-checking

- Click Check Now.
- Check Automatically fix file system errors.
- Click Start.
- Click Yes to run disk check on next restart.
- Manually restart the PNA. The disk check will run before Windows restarts.

Approximately every six months, check the second box in addition to the first box. The error-checking process takes much longer, but performs a more complete check.

Defragmentation

- Click Defragment Now...
- Click Defragment to begin the defragment process.
- Click Close when defragmentation is complete.
Recovering from PNA Hard Drive Problems

The leading cause of PNA failures is problems with the PNA Hard Disk Drive (HDD). These problems are usually preventable (see Preventing PNA HDD Problems), and in many cases, recoverable. The following could save you weeks of downtime and the cost of replacing your PNA HDD.

This document is now on the Agilent PNA Support Website: http://na.tm.agilent.com/pna/. When at this webpage, click the Hard Drive Recovery link.

If your PNA does experience a Hard Disk Drive Problem, you will not be able to access this Help file, but you may be able to access the Internet from another computer.
Wake PNA at Specified Time

Depending on the CPU that is used in your PNA-X, you may be able to start your PNA-X at a specified time every day. This feature allows the PNA-X to be shut down at night (saving energy), then wake the PNA-X at a specified time to allow sufficient warm-up time in the morning before being used.

To use this feature, the PNA-X must have a 2000 MHz CPU board. See how to determine the CPU board.

How to configure the PNA-X to wake at a specified time:

1. Connect a USB keyboard to the PNA-X.
2. **Restart** the PNA-X.
3. During power-up, press F2 on the keyboard at the moment it indicates to do so on the bottom of the PNA-X screen.
4. In the BIOS, go to **Configuration** then **Power Control Configuration**. Set **Resume On Time:** to **ON**, and set the **Resume Time:** to the desired wake-up time.
5. Press **F10** to Save, Exit, and restart the PNA-X.
6. At the end of the day, Shut Down (not hibernate or standby) the PNA-X.
7. The PNA-X will wake-up at the specified time.

**Note:**
A utility can also be set to provide a Wake-on-LAN (WOL) call. This is outside the scope of PNA Help. Search the internet for **WOL utility**.

To configure the PNA-X for WOL, follow the above procedure for accessing the BIOS.

In the BIOS:

1. Go to **Configuration** then **LAN Configuration**. Ensure that **Power Saving when S5** to **Disabled** (default setting).
2. Go to **Configuration** then **Power Control Configuration**. Set **PME Wake From S5** to **Enabled**.
3. Press **F10** to Save, Exit.

Last Modified:

18-May-2010  MX New topic
Microsoft Windows® XP / 2000 Considerations

In this topic:

- Microsoft Windows on the PNA
- Using USB
- Plug & Play Stability and Security
- LAN Connections
- Single and Double Click option
- Windows XP Theme
- Printing

Microsoft Windows on the PNA

- Beginning in April 2004, the PNA is shipped from the factory with a modified version of Microsoft Windows XP operating system. Previously, the PNA was shipped with Windows 2000. The PNA application performs identically using these two operating systems.

- Beginning in Dec. 2005 with PNA Rev 6.0, firmware cannot be upgraded on PNA models that use Microsoft Windows 2000. For more information, see the PNA support website.

To determine which Operating System is installed on your PNA:

1. Minimize the PNA application

2. On the PNA desktop, click Start.

3. Along the side of the Start menu appears one of the following:
   - Windows 2000 Professional
   - Windows XP Professional

VERY IMPORTANT  Protect your hard drive!
The leading cause of PNA failures is problems with the PNA Hard Disk Drive (HDD). These problems are usually preventable, and in many cases, recoverable. Learn more about protecting your PNA.

Using USB

The PNA has at least two USB ports for connecting devices: one on the front panel and at least one on the rear.
The main advantages of USB are “hot” connects and disconnects and fast data transfer speeds. Electronic Calibration modules are now available with USB connections.

The first time you plug a device into a USB port there is some wait time. Windows reports it is identifying the hardware, then searching for the correct driver, then installing the driver (if it was found).

Connecting that same device back into that same port later is quick and easy, but if you move the device to a different USB port, you will have to wait through the hardware ID and driver search again.

**Learn about USB limitations.**

**Note:** Certain USB devices (such as ECAL modules) require you be logged on with Administrator privileges the first time you plug them into the PNA. This must be done for each serial number. Click **Next** to choose the default settings when installing new USB devices.

**Plug & Play Stability and Security**

Plug & Play capabilities is similar to Win 95 and 98. It provides both a stable and secure operating environment. You may notice also that it greatly reduces the number of required reboots.

**LAN Connections**

Windows supports DHCP and fixed IP addressing. Also, “Hot” connect and disconnect of the LAN cable, as well as a visual indicator of LAN status in system tray area, makes LAN connections more intuitive. In addition, the Hardware Wizard helps users with system hardware configuration.

**Single and Double Click option**

By default, Windows allows a single-click method of launching icons. To revert to double-clicking, click **Start**, then **Settings**, then **Control Panel**, then click **Mouse**. In the Mouse Properties dialog, select **Double-click to open an item**. Then click **OK**.

**Windows XP Theme**

The PNA application is designed for, and best viewed in, **Windows Classic** theme. To change the theme from Windows XP to Windows Classic,

1. Minimize the PNA application.
2. Right-click on the Desktop, then click **Properties**.
3. On the Theme tab, under **Theme** select **Windows Classic**.

**Printing**

Adding a printer should be done outside of the PNA application. **Learn more**.
PNA-X, N522x, and 'C' Model Front-Panel Tour

See Also

- 'C' Models/Options
- N522x Models/Options
- PNA-X Models/Options
- Display area
- Rear-Panel Tour

Familiar Hardkey layout, similar to Agilent 8720 and 8753 Network Analyzers

Back to the familiar layout, significantly different from legacy PNA models. Most measurement settings are made from the Stimulus Block and the Response Block.

Fully functional Hardkey/Softkey selections consistent with Menu (mouse) selections

Access ALL PNA settings from the front panel using hardkey/softkeys or from the Menu using a mouse. Both methods are consistent; learn the menu structure once, and it applies to both methods of UI navigation.
Power Switch
Used for choosing between power-on (|) and standby (O) state.
Learn to power ON and OFF the PNA.

Test Ports
The PNA-X and N522x models are available with 2 or 4 test ports.
See Specs for more information about the Test port connectors and Input damage levels.

Front panel Access Jumpers
These connectors provide direct access to the PNA source and receivers. This allows you to make a wide variety of measurements and improve dynamic range.
See the PNA-X front panel jumpers specifications

<table>
<thead>
<tr>
<th>Port 1</th>
<th>Port 3</th>
<th>Port 4</th>
<th>Port 2</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="PNA-X jumper panel 1" /></td>
<td><img src="image2.png" alt="PNA-X jumper panel 3" /></td>
<td><img src="image3.png" alt="PNA-X jumper panel 4" /></td>
<td><img src="image4.png" alt="PNA-X jumper panel 2" /></td>
</tr>
</tbody>
</table>

N5247A
Port 1 and Port 3 SW SRC OUT - COMB IN jumpers moved from rear-panel (J8 through J11) to front-panel to minimize path loss.

USB Hub
This USB hub contains four USB ports to power your PNA peripherals. There are also four USB ports on the rear panel.
Limitation: The total power consumption for all eight USB ports is limited to 4.0 amps. If this limit is exceeded, all
USB ports are disabled until a device is removed and power consumption falls below the limit. When first connected, Agilent ECal modules 8509x and N4431 draw significantly more current than other modules.

**Note:**
The **FIRST TIME** each USB device (ECal module, power sensor, and so forth) by serial number is connected to a specific PNA USB port, you must be logged in to the PNA with an **Administrator** account. Learn how.

When a **New Hardware Found** dialog appears, click **OK** to install the device.

After being installed, when that same USB device is connected to that same USB port, you can be logged in to the PNA with a Limited/User account.

**Ground terminal**
Connect a banana-type plug to this terminal for grounding to the PNA chassis.

**No probe power**
Probe power is NOT provided with the PNA-X and N522x models.

**Hardkeys**

**TRACE/CHAN Keys**
Manages the Traces and Channels on the PNA display.

<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, 2, 3, 4</td>
<td>Makes the corresponding trace active.</td>
</tr>
<tr>
<td>Traces</td>
<td>Invokes the Traces softkey menu which allows you to <strong>create</strong> a new trace, <strong>select</strong> a trace, <strong>delete</strong> a trace, or <strong>maximize</strong> the trace.</td>
</tr>
<tr>
<td>Channels</td>
<td>Invokes the Channels softkey menu which allows you to manage channels.</td>
</tr>
</tbody>
</table>

**RESPONSE Keys**
Performs operations on measurement traces after data is measured - not including Data Analysis operations.

<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys - Click to learn more</th>
</tr>
</thead>
<tbody>
<tr>
<td>Meas</td>
<td>Measurement selections</td>
</tr>
<tr>
<td>-</td>
<td>S-Parameters</td>
</tr>
<tr>
<td>-</td>
<td>Balanced Parameters</td>
</tr>
<tr>
<td>-</td>
<td>Measurement Class</td>
</tr>
</tbody>
</table>

More Meas
<table>
<thead>
<tr>
<th>Format</th>
<th>Scale</th>
<th>Display</th>
<th>Avg</th>
<th>Cal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Receivers</td>
<td>Scale</td>
<td>Display settings</td>
<td>Averaging</td>
<td>Start Cal</td>
</tr>
<tr>
<td>Format</td>
<td>Electrical Delay</td>
<td>Arrangements (Overlay...)</td>
<td>Smoothing</td>
<td>Cal Wizard</td>
</tr>
<tr>
<td>Scale</td>
<td>Phase Offset</td>
<td>Windows (Managing)</td>
<td>IF Bandwidth</td>
<td>Preferences</td>
</tr>
<tr>
<td>More</td>
<td>More</td>
<td>Measurement Setups</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Display Items</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
- Global Delta Match

Correction
Power Cal
- Source Cal
- Receiver Cal

Manage Cals
- Cal Set
- Cal Type
- Cal Set Viewer

Properties (must have a Cal ON) no idea what this is
Port Ext Toolbar
Interpolation
Fixtures
- ON | Off
- Port matching
- lots more

Manage Cal Kit
Manage ECal
System Z0
Velocity Factor

**MARKER/ANALYSIS Keys**
Control all aspects of Data Analysis including Markers and Math functions..
<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys - Click to learn more</th>
</tr>
</thead>
</table>
| **Marker** | Markers  
Properties  
- Delta Markers  
- Discrete  
- Type  
- Coupled |
| **Search** | Marker Search |
| **Memory** | Data/ Memory Math  
8510 Mode |
| **Analysis** | Limit Lines  
- Limit Test  
- Global Pass/Fail  
Trace Statistics  
Gating  
Transform  
- Windowing  
- Coupling  
- Distance Marker |
| Equation Editor | |

**STIMULUS Keys**
Controls settings that determine **what** data (stimulus range), and **how** data (sweep type and triggering), is measured.
<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys - Click to learn more</th>
</tr>
</thead>
<tbody>
<tr>
<td>Freq</td>
<td>Frequency Range</td>
</tr>
<tr>
<td></td>
<td>Frequency Offset Mode</td>
</tr>
<tr>
<td>Power</td>
<td>RF Power level</td>
</tr>
<tr>
<td></td>
<td>Power Slope</td>
</tr>
<tr>
<td></td>
<td>Power and Attenuator settings</td>
</tr>
<tr>
<td>Sweep</td>
<td>Sweep Time</td>
</tr>
<tr>
<td></td>
<td>Number of Points</td>
</tr>
<tr>
<td></td>
<td>Sweep Type</td>
</tr>
<tr>
<td></td>
<td>Sweep Setup</td>
</tr>
<tr>
<td></td>
<td>Segment Table settings</td>
</tr>
<tr>
<td>Trigger</td>
<td>Trigger settings</td>
</tr>
</tbody>
</table>

**UTILITY Keys**

Performs global PNA operations.

<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys - Click to learn more</th>
</tr>
</thead>
<tbody>
<tr>
<td>Save</td>
<td>File Save</td>
</tr>
<tr>
<td></td>
<td>Save (State) As</td>
</tr>
<tr>
<td></td>
<td>Save Data As</td>
</tr>
<tr>
<td></td>
<td>Auto Save</td>
</tr>
<tr>
<td></td>
<td>Manage Files</td>
</tr>
<tr>
<td></td>
<td>Delete Files</td>
</tr>
<tr>
<td></td>
<td>User Preset</td>
</tr>
<tr>
<td>Print</td>
<td>Print</td>
</tr>
<tr>
<td></td>
<td>Print to file</td>
</tr>
<tr>
<td></td>
<td>Page Setup</td>
</tr>
<tr>
<td>Macro/Local</td>
<td>Macro Setup</td>
</tr>
<tr>
<td></td>
<td>Run Macros</td>
</tr>
<tr>
<td>Recall</td>
<td>File Recall</td>
</tr>
<tr>
<td>System</td>
<td>Security</td>
</tr>
<tr>
<td></td>
<td>Configure</td>
</tr>
</tbody>
</table>
• SICL / GPIB
• Control Panel (Windows)
• System Z0
• Power Meter Settings
• Millimeter Module

Service
Help

• Error Messages
• About NA

User Key
Touchscreen

**ENTRY Keys**

<table>
<thead>
<tr>
<th>Hard Key</th>
<th>Invokes these Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>OK</td>
<td>Closes a dialog box and enters any values made in the dialog box.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Closes a dialog box.</td>
</tr>
<tr>
<td>Help</td>
<td>Launches this Help file.</td>
</tr>
<tr>
<td>Bk Sp</td>
<td>Back Space. Backs up the cursor and deletes any previous selection.</td>
</tr>
<tr>
<td>1 to 9</td>
<td>Selects values for measurement settings, then press Enter or G/n - M/u - k/m to complete the selection.</td>
</tr>
<tr>
<td>G/n</td>
<td>Completes the value selection, assigning a unit of measurement.</td>
</tr>
<tr>
<td>M/u</td>
<td>• G/n (Giga/Nano) E12 or E-12</td>
</tr>
<tr>
<td>k/m</td>
<td>• M/u (Mega/micro) E6 or E-6</td>
</tr>
<tr>
<td></td>
<td>• k/m (kilo/milli) E3 or E-3</td>
</tr>
<tr>
<td>Enter Off</td>
<td>Enters the values that you select for the measurement settings.</td>
</tr>
<tr>
<td><strong>Decimal point</strong></td>
<td>Enters a decimal point to designate fractions of a whole number.</td>
</tr>
<tr>
<td>-------------------</td>
<td>---------------------------------------------------------------</td>
</tr>
<tr>
<td>+/-</td>
<td>Plus - Minus  Toggles between a positive and negative value entry if it is the first key pressed in the entry.</td>
</tr>
</tbody>
</table>

**Knob**

Rotate to increase or decrease the value of the active entry.

**Navigation Keys**

These keys allow you to navigate through menus and dialog boxes and select choices from the active entry toolbar.

<table>
<thead>
<tr>
<th><strong>Hard Key</strong></th>
<th><strong>Invokes these Softkeys</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Left / Right</td>
<td>Moves left and right through menus. Moves tab-left and tab-right within dialog boxes.</td>
</tr>
<tr>
<td>Up / Down</td>
<td>Moves up and down through menus. Behaves as follows in a dialog box:</td>
</tr>
<tr>
<td></td>
<td>• Modifies a numeric value</td>
</tr>
<tr>
<td></td>
<td>• Moves through items in a drop-down list</td>
</tr>
<tr>
<td></td>
<td>• Moves through options buttons in a group of option buttons</td>
</tr>
<tr>
<td>Click</td>
<td>Makes a selection just like a mouse click.</td>
</tr>
</tbody>
</table>

Last Modified:

- 4-May-2011  Added N522x models
- 15-Apr-2008  Added ALL front panel jumper images
- 23-Aug-2007  Added front panel jumpers image
Rear Panel Tour

This image includes ALL rear-panel features for all PNA models, excluding the PNA-X. See the PNA-X Rear Panel. Your PNA may not have this capability or look.

Click on a connector for detailed information.

10 MHz Reference IN/OUT

**10 MHz Reference Input** When a 10 MHz external reference signal is detected at this port, it will be used as the instrument frequency reference instead of the internal frequency reference.

**10 MHz Reference Output** This BNC(f) connector outputs a frequency reference signal for use by other test equipment.

See PNA specifications

VGA Connector Learn more

USB Hub

This USB hub contains four Hi-Speed USB 2.0 ports to power your PNA peripherals.

Each USB 2.0 port is capable of providing .5 amps. If this limit is exceeded, all USB ports are disabled until a device is removed and power consumption falls below the limit.

**Note:** When first connected, Agilent ECal modules 8509x and N4431 draw significantly more current than other modules. See PNA specifications
USB Device  Learn more

LAN Connector
This 10/100BaseT Ethernet connection has a standard 8-pin configuration and auto selects between the two data rates.

Line Power
See PNA specifications

GPIB Controller and Talker/Listener Ports
The PNA-X can be a GPIB Controller and Talker/Listener. Learn more.

Test Set I/O
See Details

Bias IN and Fuses
Apply Bias to the PNA ports through these BNC connectors.
See PNA specifications

Material Handler I/O
See details.
Beginning April 2009, PNA-X and N5264A models shipped with this newer CPU board. The major changes include:

- **2 GHz CPU**
- **A** Removable Hard disk drive (Learn more in the Service Guide)
- **B** Relocated GPIB Controller port
- **C** Relocated VGA connector

### 10 MHz Reference IN/OUT

**10 MHz Reference Input** When a 10 MHz external reference signal is detected at this port, it will be used as the instrument frequency reference instead of the internal frequency reference.

**10 MHz Reference Output** This BNC(f) connector outputs a frequency reference signal for use by other test
equipment.

- See **SCPI** command that detects an external reference signal at this connector.
- See **PNA-X specifications**

---

**VGA Connector**  Learn more

---

**USB Hub**

This USB hub contains four USB ports to power your PNA peripherals. There are also four USB ports on the front panel.

**Limitation:** The total power consumption for all eight USB ports is limited to 4.0 amps. If this limit is exceeded, all USB ports are disabled until a device is removed and power consumption falls below the limit. When first connected, Agilent ECal modules 8509x and N4431 draw significantly more current than other modules. See **PNA-X specifications**.

See Important First-time USB connection note.

---

**USB Device**  Learn more

---

**LAN Connector**

This 10/100BaseT Ethernet connection has a standard 8-pin configuration and auto selects between the two data rates.

---

**Line Power**

See **PNA-X specifications**

---

**GPIB Controller and Talker/Listener Ports**

The PNA-X can be a GPIB Controller and Talker/Listener. Learn more.

---

**RF Path Access**

These connectors are NOT available on the **N522x** and **N5264A** models.

These connectors allow **RF Path Configuration**.

Ports 3 and 4 are not available on 2-port models.

N5247A - J8 thru J11 are moved to the **front-panel**.
**RF and LO OUT**

The **RF OUT** connector is NOT available on the **N5264A**.

For the **N5247A** and **N5227A**: Added RF2 OUT (J12) for 4-port 110 GHz single sweep PNA. Enables driving two frequency extenders simultaneously.

*Caution:* **LO OUT** has more power than previous PNA models.  
[See specifications](#)

**IF Path Inputs**

Option 020 adds these connectors, which allow access to the PNA Receiver / IF paths.

These are labeled A, B, C/R1, D/R2, R.

- For 2-port models, use A, B, R1, R2.
- For 4-port models, use A, B, C, D, R.

[See IF Path Configuration settings and block diagram.](#)

**Power I/O**

Has some of the **AUX I/O connector** functionality on the PNA-L and E836xB models.  
[See Details](#)

**28 V (BNC output)**

Used to power a noise source for the **Noise Figure App**.

**External and AUX Trigger I/O**
MEAS TRIG IN - When enabled, PNA is triggered by signals on this connector. [Learn more.](#)

MEAS TRIG RDY - When enabled, PNA outputs a 'READY' signal on this connector to other devices. [Learn more.](#)

AUX TRIG 1&2 IN - When enabled, PNA accepts signals on these connectors which indicates that the external devices is ready to be triggered. [Learn more.](#)

AUX TRIG 1&2 OUT - When enabled, PNA outputs signals on these connectors either before or after a measurement. [Learn more.](#)

**Test Set I/O**

See Details

**Bias IN and Fuses**

Apply Bias to the PNA ports through these BNC connectors.

The PNA will meet all of its RF specifications with bias up to 200 ma. As the DC bias is increased, corrected source match and directivity will degrade at low RF frequencies. Above 100 MHz, with up to 1 A bias, there is very little degradation in corrected source match and directivity.

[See specifications](#)

**Material Handler I/O**

See details.

**Pulse I/O**

See Details

**1.6 GHz CPU**

See [CPU Speed / Performance](#)

Last modified:
28-Jun-2011  Added link to first time note
4-May-2011   Added N522x
12-Nov-2010  Added N5247A RF2
9-Apr-2009   Added new CPU board
14-May-2008  Added RF Path images
4-Sep-2007   Added 28V image
June 6, 2007 Added RF and IF connector images
January 11, 2007 MX New topic
Powering the PNA ON and OFF

The following is described in this topic:

- How to...
  - Hibernate
  - ON
  - Shutdown
  - Turn OFF Autostart

**Notes**

During boot up of Windows or of the Network Analyzer application program, do NOT press keys on the front panel, rotate the RPG knob, or connect a USB device. Doing so MAY lead to a front panel lockup state.

If the PNA front-panel keypad or USB ports are not responding, SHUTDOWN or RESTART the PNA; do NOT Hibernate. This causes the PNA drivers to awaken from hibernation in the same corrupt state.

**How to Log off, Shut down, Restart, or Hibernate the PNA.**

1. BRIEFLY press the front-panel PNA power button.
2. In the What do you want the computer to do? list, choose an action:
   - Log off (closes programs)
   - **Shut down**
   - Restart (shutdown and start)
   - **Hibernate** (available ONLY on PNA “C” models)
3. Press OK to perform the action

**Note:** ONLY if the PNA is locked and you cannot operate the mouse or keypad - Press and hold the power button for at least four seconds. **This practice should be avoided!** Repeated shutdowns in this manner WILL damage the hard drive. [Learn more about damaging the PNA hard drive.](#)

**Hibernate Mode**

**Note:** Hibernate is available ONLY on PNA “C” models. [Learn about these model numbers.](#)
In hibernate mode the current instrument state is automatically saved to the hard disk before the PNA is powered OFF.

When the PNA is powered ON, this instrument state is loaded, thus saving time over a full system boot-up.

A password is usually NOT required to resume PNA operation after Hibernate mode.

The hibernation state is the normal OFF state. A small amount of standby power is supplied to the PNA when it is in the hibernation mode. This standby power only supplies the power switch circuits and the 10 MHz reference oscillator; no other CPU-related circuits are powered during hibernation. To guarantee that your measurements meet the PNA specified performance, allow the PNA to warm-up for 90 minutes after the power button light has changed from yellow back to green.

**ON Mode**

- To turn ON the PNA press the power button.
- The power indicator will change to green when power is ON.

**Turn OFF PNA Autostart**

The PNA application (835x.exe) always starts automatically when power is turned ON. To cause the PNA to NOT Autostart, do the following:

1. Minimize the PNA application.
2. From Windows Explorer, navigate to and double-click the following file: C:\Program Files\Agilent\Network Analyzer\Service\Toggle_PNA_Autostart.

The script toggles the PNA Autostart mode ON and OFF.

**Shutdown Mode**

- In shut down mode the current instrument state is NOT automatically saved before the PNA is powered OFF.
- When the PNA is again powered ON, a full system boot-up is performed and the PNA powers-up in the preset settings.
- A password may be required to resume PNA operation after being in Shutdown mode. Learn more.
- To guarantee that your measurements meet the PNA specified performance, allow the PNA to warm-up for 90 minutes after the power indicator has turned green.
- The power indicator will change to yellow when power is OFF.

**Note:** If the PNA is locked and you cannot operate the mouse or keypad, shut down the PNA by pressing and holding the power button for at least four seconds. This practice should be avoided! Repeated shutdowns in this manner WILL damage the hard drive. Learn more.
about damaging the PNA hard drive.

Unplugging the PNA

- Remove the power cord from the PNA ONLY when the power indicator is yellow, in either Hibernate or Shutdown mode. If the power cord is removed while the power indicator is green (PNA ON), damage to the hard drive is possible.

- The indicator will remain yellow for several seconds after the power cord has been removed.

- When plugged back in and the power button is pressed to ON, the PNA starts in the mode it was in when the power cord was unplugged, either Hibernate or Shutdown.

Last Modified:

4-May-2011    Added N522x models

3-Jun-2009    Updated for N5244A and N5245A
Traces, Channels, and Windows on the PNA

It is critical to understand the meaning of the following terms as they are used on the PNA.

- **Traces** - Managing
- **Channels** - Managing
- **Windows** - Managing

**Note:** You may experience a significant decrease in computer processing speed with combinations of the following: increased number of points, number of traces, and calibration error terms (full 2-port or 3-port). If this becomes a problem, you can increase the amount of RAM with PNA Option 022. To monitor the amount of PNA memory usage, press **Ctrl Alt Delete**, select **Task Manager**, then click on the **Performance** tab.

**Other Quick Start topics**

**Traces** are a series of measured **data points**. There is no theoretical limit to the number of traces. However, the practical limit is the **maximum number of windows** * the maximum number of traces per window (24).

In addition, one memory trace can be stored and displayed for every data trace. **Learn more about Math / Memory traces.**

Trace settings affect the presentation and mathematical operations of the measured data.

The following are Trace settings:

- **Parameter**
- **Format and Scale**
- **Smoothing**
- **Correction ON / OFF**
- **Electrical Delay**
- **Phase Offset**
- **Trace Math**
- **Markers**
- **Time Domain** (Opt 010)

**Managing Traces**

- **How to Select a trace**
- **How to Delete a trace**
How to Move a trace
How to Maximize a trace
How to Create a new trace
How to Change the trace parameter.
How to display a custom trace title.
How to display a wide active trace

How to Select a Trace
A trace must be selected (active) before its trace settings can be changed.

How to know which trace is Active?

Using front-panel HARDKEY [softkey] buttons
1. For Traces 1-4, press the corresponding Hard Key
2. For other trace numbers, press TRACES
3. then [Select Traces]
4. Select a trace number in the Entry toolbar.

PNA Menu using a mouse
1. Click the Trace Status label or trace.

How to Delete a Trace

Using front-panel HARDKEY [softkey] buttons
1. For Traces 1-4, press the corresponding Hard Key
2. For other trace numbers, press TRACES
3. then [Select Traces]
4. Select a trace number in the Entry toolbar.

PNA Menu using a mouse
1. Right-click the Trace Status label, then click Delete.
How to Move a trace to a different window

Using front-panel HARDKEY [softkey] buttons

1. Select the trace to move.
2. Press TRACES
3. then [Move Trace]
4. Select a window number in the following dialog.

PNA Menu using a mouse

1. Right-click the Trace Status label, then click Move Trace.

PNA-X ONLY

This dialog is launched by clicking Trace/Chan, then Delete Trace
The Select Trace dialog is launched by clicking Trace/Chan, then Select Trace

Select, Delete, Move Traces dialog box help

Both the Select Trace and Delete Trace dialogs work the same.
Select a trace, then click OK.
Only ONE trace can be Selected or Deleted.

Note:
To EASILY select a trace, click the Trace Status label.
To EASILY delete a trace, right-click the Trace Status label, then click Delete.

Trace Max
Makes the active trace the ONLY trace on the display. All other traces are hidden.

How to do Trace Max

- Select Trace, then Trace Max.
With Trace Max ON, select a different trace from the Traces softkeys to make that trace visible.

To make all traces visible again select Trace Max OFF

**Trace Title**

A Trace Title overwrites the Measurement Parameter in the Trace Status area, the Status Bar, and hardcopy prints.

- This title has priority over Equation Editor titles.
- The practical limit is about 70 characters if there is only one trace.
- Spaces are accepted but not displayed; use underscores.
- The title is annotated as follows:

```
Tr 1 NewTrace LogM 10.00dB/ 0.00dB
```

**How to enter a Trace Title**

1. Click the Trace Status label to select a trace.
2. Click Trace/Chan, then Trace, then Trace Title.
3. Click Enable, then type the trace title. Click Keyboard to type with a mouse.
4. To remove the trace title, clear the Enable checkbox, or delete the text from the dialog entry.

**Channels** contain traces. The PNA can have up to **200 independent channels** (32 before A.09.00)

Channel settings determine how the trace data is measured. All traces that are assigned to a channel share the same channel settings. A channel must be selected (active) to modify its settings. To select a channel, click the Trace Status button of a Trace in that channel. The following are channel settings:

- Frequency range
- Power level
- Calibration
- IF Bandwidth
- **Number of Points**
- **Sweep Settings**
- **Average**
- **Trigger** (some settings are global)

**Managing Channels**

**How to Select a Channel**

A channel must be selected (active) before its settings can be changed.

To make a channel active, select a trace in that channel.

**How to Turn ON or OFF a Channel**

Click **Trace/Chan**, then **Channel**, then **Turn On / Off Channel**.

![Turn ON | OFF Channel dialog box help](image)

Both the Turn ON and Turn OFF dialogs work the same.

Select a channel, then click **OK**. Only ONE channel can be selected.

When turning ON a channel, the new channel is always the Standard Measurement Class with an S11 trace.

**Note:** To create more than one trace in a new channel, click Trace, then New Trace

**Windows** are used for viewing traces.

- The PNA can show an **UNLIMITED** number of windows on the screen (16 windows previous to PNA release 6.2) with the following limitations:
  - The COM property **MaximumNumberOfWindows** returns 1000 ('unlimited' is not a number).
  - The **SCPI status register** can only track the status of up to 576 traces.
  - Each window can contain up to **24 traces** (8 traces previous to PNA release 8.2).
- Windows are completely independent of channels.
- Learn to create and manage windows.

The following is a window containing two traces. Both traces use the same channel 1 settings as indicated by the annotation at the bottom of the window.

PNA-X shows the window number in the lower-left corner of the window. The following shows window 5.

**Managing Windows -**

**How to make various window settings**

New, Close, Tile, Cascade, Minimize, Maximize

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>RESPONSE</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then <strong>[Display]</strong></td>
<td>2. then <strong>Display</strong></td>
</tr>
<tr>
<td>3. then <strong>[Windows]</strong></td>
<td>3. then <strong>Windows</strong></td>
</tr>
</tbody>
</table>

[Programming Commands]
Close Window dialog box help

Select a window, then click OK. The remaining windows are tiled.

Only ONE window can be selected.

Traces contained in a closed window are deleted.

**Note:** To EASILY close a window, click the X in the upper right corner of a window. The X is only visible when Title Bars are enabled. The remaining windows are NOT tiled.

See Customize the PNA screen to learn how to make other window settings

---

Last modified:

- 4-Sep-2008  Removed legacy content
- 1-May-2008  Increased max number of traces per window
- 9/19/06      MQ Modified for unlimited number of windows
Basic Measurement Sequence

The following process can be used to setup all PNA measurements:

**Step 1. Set Up Measurements**
Reset the analyzer, create a measurement state, and adjust the display.

**Step 2. Optimize Measurements**
Improve measurement accuracy and throughput using techniques and functions.

**Step 3. Perform a Measurement Calibration**
Reduce the measurement errors by performing a calibration.

**Step 4. Analyze Data**
Analyze the measurement results using markers, math operations, and limit tests.

**Step 5. Print, Save or Recall Data**
Save or print the measurement data.
# Frequency Blanking

For security reasons, you can prevent frequency information from appearing on the PNA screen and printouts.

## How to set Frequency Blanking

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press SYSTEM</td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then [Security]</td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Security</strong></td>
</tr>
</tbody>
</table>

## Security Setting dialog box help

### Notes

- To learn how to erase memory before moving your PNA out of a secure area, see [http://na.tm.agilent.com/pna/security.html](http://na.tm.agilent.com/pna/security.html).

## Security Levels

**None** - All frequency information is displayed on the screen and printouts.

**Low** security level - Frequency information is blanked from the following:

- Display annotation
- Calibration properties
- All tables
• All toolbars

• All printouts

• **External sources** - See Also: *Preference to Deactivate External Devices on Preset*. **Note:** Frequency blanking is fully supported ONLY on Agilent MXG sources with option 006. On MXG models without option 006 and all PSG models, the window state is turned OFF. When the “local” button is clicked on the source, then frequency is re-displayed.

**High** security level - Low security level settings PLUS:

• **GPIB console** is inactive

**Extra** security level - High security level settings PLUS:

• All ASCII **data saving** capability (.snp, .prn, .cti) is saved without frequency information. The X-axis information is replaced with data point numbers. Before A.08.50, saving these file types was NOT allowed.

• **Mixer setup files** (*.mxr) can NOT be saved.

**For ALL security levels:**

Frequency information is **NOT** blanked from the following:

• **Service Adjustment Programs**

• Your COM or SCPI programs.

**Instrument State and Cal Sets**

The security level is always saved and recalled with an instrument state. However, the instrument state may contain a Cal Set or link to a Cal Set. **Learn more.** This may influence the security level when the instrument state is recalled. Here is how.

• When a new Cal Set is created at the end of a calibration, the current system security level is stored with it.

• The only way to change an existing Cal Set’s security level is by writing a new calibration into the Cal Set.

• When later applied to a channel, if the Cal Set has a **higher** security level than the current system security level, the system security level will become upgraded to that of the Cal Set.

• When saving an instrument state to either a *.csa or *.cst file, the security levels of the system and Cal Set are saved separately. When recalled, the higher security level of the two is applied.

• To view the security level of a Cal Set, see **Cal Set Properties**.
**Re-displaying frequency information**

- When in **Low** security level, do any of the following:
  - Revisit this dialog box and select **None**
  - Perform an **instrument preset**
  - Recall an Instrument State/Cal Set with security level of **None**.
- When in **High** or **Extra** security level, do any of the following:
  - Perform an **instrument preset**
  - Recall an Instrument State/Cal Set with security level of **None**.

---

Last Modified:

- 8-Feb-2011 Clarified external sources
- 4-Nov-2010 Added external sources
- 21-Jun-2010 Clarified re-display freq information
- 17-Feb-2009 Added Extra file saving with data point numbers
- 4-Sep-2008 Removed legacy content
- 13-Aug-2008 Added ECal note
- 29-Apr-2008 Fixed hardkey presses
- 17-Jul-2007 Added Extra setting
Internal Second Source

The following PNA models include an internal second source.

<table>
<thead>
<tr>
<th>Model</th>
<th>Total # of Ports</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5230A Opt 146</td>
<td>4</td>
</tr>
<tr>
<td>N5230A Opt 246</td>
<td>4</td>
</tr>
<tr>
<td>PNA-X Opt 224</td>
<td>2</td>
</tr>
<tr>
<td>PNA-X and N522x Opt 400's</td>
<td>4</td>
</tr>
</tbody>
</table>

How to use the second source

- Set frequency using the Frequency Offset Opt 080 dialog.
- Set power using the Advanced Power dialog.
- Source power calibration of the second source is performed as usual.
- Using FCA, click the LO button to set frequency and power.
- The specifications of the second source are the same as source 1.

Benefits / Uses of the second source

- Up to five times faster than stepping an external source.
- Measure Mixers with internal swept or fixed LO.
- Measure TOI or Intermodulation distortion.

Internal Second Source Restrictions

Source 1 and Source 2 are available at specific ports as follows:

**N5230A (PNA-L) models:**

- Source 1 power is available at Port 1 OR Port 2; NOT at both ports simultaneously.
- Source 2 power is available at Port 3 OR Port 4; NOT at both ports simultaneously.

**PNA-X and N522x 4-port models**

- Source 1 power is available at Port 1 OR Port 2; NOT at both ports simultaneously.
- Source 2 power is available at Port 3 AND Port 4; BOTH ports simultaneously.
- Other routing configurations are possible using the **RF Path Configurator**.

**PNA-X Opt 224 (PNA-X 2-port model):**

[Image of PNA-X Opt 224]

- Source 1 power is available at **Port 1** OR **Port 2**; NOT at both ports simultaneously.
- Source 2 (**SRC 2**) power is available at **Out 1** AND **Out 2**; BOTH ports simultaneously.
- Other routing configurations are possible using the **RF Path Configurator**.

**Remotely Accessing the Internal Second Source**
See **Remotely Specifying a Source Port**.

---

Last modified:

4-May-2011  Modified for N522x
27-Apr-2011  All possibilities listed at Path Configurator
13-Nov-2010  Added port 1 combiner link.
26-May-2009  Added dual source path
6-Apr-2009  Replace N5242A with PNA-X
13-May-2008  Added remote section
1123-Jul-2007  MX Added PNA-X models
10/02/06  MQQ New topic
Networking and Connecting the PNA

The PNA as a PC

- PNA User Accounts and Passwords
- Drive Mapping
- Connecting the PNA to a PC
- Easy versus Secure Configuration
- Changing Network Client
- Using VNC to Control the PNA User Interface

GPIB / COM Programming

- Configure for COM/DCOM Programming
- Configure for GPIB, SCPI, and SICL

Controlling External Devices

- Configure an External Device
- E5091 TestSet Control
- External Testset Control
- Interface Control Feature
- TestSetIO Connector
- Handler IO Connector
- AuxIO Connector
PNA Preferences

PNA preferences are settings that survive a Preset or PNA Shutdown. PNA Preferences are listed on this page with links to locations that provide more information.

How to set PNA Preferences

Using front-panel HARDKEY [softkey] buttons

1. Press SYSTEM
2. then [Configure]
3. then [More]
4. then [Preferences]

PNA Menu using a mouse

1. Click Utility
2. then System
3. then Configuration
4. then Preferences

Preferences dialog box help

PNA Preferences survive a Preset and a system reboot.

A checked box makes the following statements true unless stated otherwise.

Touchscreen always ON

Active Trace is always wider.

Active Trace is briefly wider.
### Cal: Auto-save to User Cal Set

This setting only affects calibrations performed using SCPI or COM. Cals performed from the User Interface ALWAYS offer a choice to save to a named Cal Set.

With Rev 6.0 we implemented a change that defaults to saving completed calibrations to Cal Registers instead of User Cal Sets.

Check the box to revert to always automatically saving completed calibrations to an auto-named User Cal Set. **Caution:** this can cause a lot of save User Cal Sets. Learn more.

The following message appears when both of the two Cal Set choices are selected:

"Cal: Auto-save preferences conflict"

**Auto-save to User Cal Set** (above)- or - **Auto-save to current Cal Set** (below)

Uncheck one of these.

### Cal: Auto-save to current Cal Set

This setting only affects calibrations performed using SCPI or COM. Cals performed from the User Interface ALWAYS offer a choice to save to a named Cal Set.

Check the box to always automatically save a competed Cal to the Cal Set that is currently selected on the specified channel, which could be the channel Cal Register. If the channel does not yet have a selected Cal Set, the Cal will be saved to a new User Cal Set with an automatically-generated name.

### Cal: For Guided Cal, set external trigger.

With Rev 6.0 we implemented a change that allows the measurement of Cal Standards during a Guided Cal to be externally triggered when trigger source is set to External.

Check the box to revert to NOT allowing the measurement of Cal standard to be externally triggered. All Guided Cal Standard measurements are triggered internally regardless of the trigger source setting. Learn more.

### Cal: For Unguided Cal, set external trigger.

Same as above, except for Unguided Cals.

### Cal: Simulated Cal Behavior

For SCPI behavior only. Learn more.

### Cal: For Frequency Offset, use Primary Frequencies

Use when making mmWave measurements without a test set. Learn more.

### Memory: Data Math 8510 Mode

Check the box to simulate the Agilent 8510 data processing chain as it pertains to Trace Math and Memory. Learn more.

### Meas: Mathematical offset for receiver attenuation

By default, the PNA-X offsets the test port receivers by the amount of receiver attenuation. All other PNA models do **NOT** offset the receiver attenuation.

Check the box to offset the receiver attenuation.
Clear the box to NOT offset the receiver attenuation.

Learn more.

**Meas: Mathematical offset for source attenuation**

By default, the PNA offsets the power at the reference receiver by the amount of source attenuation. Clear the box to NOT offset the source attenuation. Learn more.

**Meas: RF power On during frequency sweep retrace**

By default, the PNA leaves RF power ON during a retrace of single-band frequency or segment sweeps. Clear the box to turn RF power OFF during a retrace of single-band frequency or segment sweeps. Learn more.

**Meas: Power Sweep Retrace**

By default, at the end of a power sweep, while waiting to trigger the next sweep, the PNA maintains source power at the start power level. Clear the box to maintain source power at the STOP power level. Learn more.

**Meas: External Trigger OUT is Global**

By default, the PNA-X trigger output properties follow the main Trigger Scope setting. For example, when a channel is in point sweep mode, the trigger output will automatically be sent per point.

External triggering for all other PNA models defaults to Global. To send external trigger signals for each point, the Trig OUT Per Point setting must also be checked.

Check the box to set to Global. Clear the box to set to Channel. Learn more.

**Meas: Port 1 Noise Tuner Switch state**

By default, the Noise Figure port 1 DPDT switch is set to EXTERNAL. This setting always provides incident power through the front panel loop.

Clear the box to set to Internal, which is one method to make accurate S-parameter measurements when an ECal module is connected as a Noise Tuner. Learn more.

The Noise Figure App will throw the switch as needed. However, External Test Sets (in Multiport mode) rely on this switch being set to External.

**Meas: Draw failed trace segments in red**

By default, failed data points (dots) are drawn in red.

Check the box to draw failed segments in red. Learn more.

**Report source unleveled events as errors**

For SCPI behavior only. Learn more.

**Ext Device: de-activate on PRESET and recall.**

By default, external devices are de-activated when the PNA is Preset or when a Instrument State is recalled. Clear the box to have external devices remain active when the PNA is Preset or when a Instrument State is recalled.
recalled.
Learn more about External Devices.

On PRESET set two-point group delay aperture
By default, group delay aperture is set to 11 points.
Check the box to have group delay aperture set to 2 points. Learn more.

On Preset turn power ON
By default (checked), Instrument Preset always turns source power ON.
Checkbox cleared - When the current source power setting is OFF, leaves power OFF after Preset. When the current power setting is ON, turns power ON after Preset. Learn more.

Turn Source Power Off when receiver is overloaded
By default (cleared) power remains ON when a receiver is overloaded.
Check the System Preference box to turn OFF power to ALL ports when a receiver is overloaded.
A notification dialog appears. Click OK, then lower the power level, then turn power ON. (Click Stimulus, then Power)

Report when receiver is overloaded
By default (checked), a warning message is displayed on the PNA screen indicating that a receiver is overloaded or in compression. The displayed data is probably not accurate. One error per sweep appears and is reported in the Error Log.
Clear the box to NOT show overload warnings on the screen or report these errors in the error log.

The More buttons launch dialogs that contain predefined preferences:

Define Data Saves -
While not explicitly called Preferences, all of these settings survive a shutdown. Learn more.

User Preset
Specify the Instrument State file that the PNA will use when Preset. Learn more.

User Key
Sets softkey preferences. Learn more.

Page Setup
Standard printer settings (Paper, Orientation, and Size) do NOT survive a PNA shutdown.
All other settings DO survive a PNA shutdown. Learn more.

Millimeter settings
Sets MM Wave configurations. Learn more.
**Power Limit**
Sets Power Limits and Offsets. [Learn more.]

**Disp Colors**
Sets PNA display items to custom colors. [Learn more.]

**Print Colors**
Sets PNA print items to custom colors. [Learn more.]

Although they are called preferences, the following settings do NOT survive a PNA shutdown.

<table>
<thead>
<tr>
<th>Calibration</th>
<th>UI Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Show or not, the first 'Method' Page of the Cal</td>
<td></td>
</tr>
<tr>
<td>Wizard.</td>
<td>[Cal Preferences]</td>
</tr>
<tr>
<td>Set and order default Cal Types</td>
<td></td>
</tr>
<tr>
<td></td>
<td>[Cal Preferences]</td>
</tr>
<tr>
<td>Perform orientation of the ECal module during</td>
<td></td>
</tr>
<tr>
<td>calibration?</td>
<td>[ECal Wizard]</td>
</tr>
<tr>
<td>Specify ECal port mapping when orientation is</td>
<td></td>
</tr>
<tr>
<td>OFF</td>
<td>[ECal Wizard]</td>
</tr>
<tr>
<td>Show or hide custom Cal Windows during Cal</td>
<td>[Cal Window] (remote commands only)</td>
</tr>
</tbody>
</table>

**Last Modified:**

- 10-Sep-2010 Added Cal windows
- 26-Aug-2010 Added overload preferences
- 24-Mar-2010 Removed Error messages button
- 31-Jul-2009 Added External Device, primary FOM, draw failed segments, Power Limit, disp colors, print colors (9.0)
- 6-Feb-2009 Added new Cal set selection
- 22-Apr-2008 Added UI
- 10-Apr-2008 Added RemoteCalStorage and replaced AuxTrigger properties
- 5-Feb-2007 MX New topic
Using VNC to Control the PNA User Interface

VNC (Virtual Network Computing) allows you to control the User Interface of a PNA from any PC. The PNA display appears on the connected PC display. Mouse and keyboard control can occur from both the PNA and PC, although not simultaneously.

Both the PNA and PC must be connected to the internet. The responsiveness of the PNA while using VNC is dependent of the speed of your internet connection.

Every PNA is shipped with VNC installed. However, you must download and install the VNC software onto the PC. The following procedures can help you configure VNC to view and control the PNA application from your PC.

On the PNA, run VNC Server

To do this:

1. Click View, then Minimize Application.

2. Click Start, then Programs, then TightVNC, then Launch VNC Server.

   - When the server is running, the icon is visible in the lower right corner of the display.

   - The first time you run VNC Server, you may be required to set a password to control access from remote PCs.

   - To automatically start VNC when the PNA computer boots, drag a Launch VNC Server shortcut to your User "startup" folder. The following is the Administrator folder: C:/Documents and Settings/Administrator/Start Menu/Programs/Startup.

On the PC, run VNC Viewer

To do this:

1. Download from http://www.tightvnc.com/ and install TightVNC on the PC.

2. From the PC Desktop, click Start, then Programs, then TightVNC, then TightVNC Viewer

3. When prompted for the Hostname, type the full computer name or IP address of the PNA.

4. When prompted for the password, type the password you set when configuring VNC on the PNA.

Last Modified:

15-Apr-2008  Uplighted for DatedVNC
LXI-1.1 and VXI-11.3 Compliance

PNA-X, N522x, and PNA-C models that were shipped with PNA version A.08.20 or higher are LXI-1.1 and VXI-11.3 compliant.

LXI-1.1 Compliance

A PNA is LXI-1.1 compliant if the logo appears on the PNA splash screen when the PNA application starts. Learn more about LXI at http://www.lxistandard.org/

VXI-11.3 Compliance

To be compliant with VXI-11.3, the PNA must have been either:

- Shipped from the factory with PNA version A.08.20 or higher, or
- Had the Hard Disk Drive (HDD) upgraded since about June 2008 when A.08.20 was released and using PNA Rev. A.08.20 or higher.

Learn more about VXI at http://www.vxi.org/

LAN Status

When a LAN connection is used with the PNA, the LAN Status dialog allows you to see the IP address and other LAN connection properties.

How to view LAN Status

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then <strong>[Configure]</strong></td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then <strong>[More]</strong></td>
<td>3. then <strong>Configure</strong></td>
</tr>
<tr>
<td>4. then <strong>[LAN Status]</strong></td>
<td>4. then <strong>More</strong></td>
</tr>
<tr>
<td></td>
<td>5. then <strong>LAN Status</strong></td>
</tr>
</tbody>
</table>

Programmable Commands
**LAN Status** dialog box help

**Indicator**  Shows the current status of the LAN connection.
- **NORMAL** - Indicates that the PNA LAN is ready for communication.
- **IDENTIFY** - Indicates that a remote computer has invoked an LXI identification operation on the PNA using the web-based interface or **LXIDeviceIDState** COM property.
- **FAULT** - Indicates that the PNA LAN interface is not connected to the Internet.

**IP Address**  Shows the current IP address of the PNA

**Computer Name**  Shows the full computer name of the PNA. Learn how to change this. If you see the IP address listed here, that means there is no DNS server specified in the network setup.

**MAC Address**  Shows the unique address of the PNA computer. Also known as HostID.

**LAN Reset**  Provides a LAN Configuration Initialize (LCI) mechanism. Press to return the following settings to factory default conditions:

- **IP Address Configuration (DHCP):**  Enabled
- **ICMP Ping Responder:**  Enabled
- **Web Password for configuration:**  Resets the password for the factory default account (Agilent) to 'agilent'. If the pna-admin account had been renamed by the PNA owner, this function creates a new Administrator account named pna-admin and sets its password to 'agilent'. Learn more about User names and Passwords.

**Web Server Software**

If your PNA is LXI Class C compliant (see above), you can connect to the PNA using a web browser over an internet connection.

To do this, when the above dialog indicates a **NORMAL** condition:

1. From a web browser, type `http://<your_PNA_computer_name>`. For example, to connect to the fictitious PNA in the dialog above, type: `http://pna1-22`
2. Type the log on User Name and Password

3. You will see the following welcome screen with connection links.
Using Help

This topic discusses the following:

- PNA Documentation
- Printing Help
- Copying Help to your PC
- Launching Help
- Navigating Help
- Help Languages
- Glossary
- Dialog Boxes
- Documentation Warranty
- Suggestions Please

See Also
Help, About Network Analyzer

Other Quick Start Topics

PNA Documentation

This Help file, which is embedded in the PNA, is the Users Guide and Programming Manual for the PNA. The help file is automatically updated on the PNA when firmware is updated. Only the PNA Installation and Quick Start Guide is shipped with new PNA instruments.

Hardcopy manuals are no longer available for purchase with the PNA.

All PNA documentation, including the latest online Web Help version of this Help file, and a printable .PDF version of the Help file, are available at http://na.tm.agilent.com/pna/help/index.html.

Printing Help

Beginning with the PNA 5.2 release (March 2005), we once again offer a .pdf version of PNA Help. Download the .pdf file from http://na.tm.agilent.com/pna/help/index.html. You can still print individual PNA Help topics by clicking the Print icon at the top of the PNA Help window.
Copying Help to your PC
With the Help system on your PC, you can read about the PNA while away from it. You can also Copy and Paste programming code from this Help system directly into your programming environment.

The Help file is located on your PNA hard-drive at \C:\Winnt\Help\PNAHelp.chm. If both the PNA and PC are connected to LAN, you can map a drive and copy the file directly.

The Help file can also be downloaded from http://na.tm.agilent.com/pna/help/index.html.

Launching Help
The Help system can be launched in three ways:

1. From the front panel Help button.
2. From the Help drop-down menu
3. From Dialog Box Help

Navigating Help
The Help Window contains 3 panes (regions):

1. Toolbar Pane
2. Topic Pane
3. Navigation Pane

Toolbar Pane
The Toolbar is at the top of all Help windows. It allows you to resize the window, browse and print the selected topic.
1. Hide or show the navigation pane
2. Locate the topic in the table of contents
3. Back to topic visited previously
4. Forward again if Back was clicked
5. Go to the Home page.
6. Print the topic pane.

**Navigation Pane**

Click the following tabs in the Navigation Pane to access information in the Help system:

- **Table of Contents Tab**
- **Index Tab**
- **Search Tab**
- **Favorites Tab**

**Table of Contents Tab**
1. Click tab to select Table of Contents.
2. Click a book to access related topics.
3. Click to display a topic.
4. Right click to access menu.
5. Click to display specifications
6. Click to display glossary

**Index Tab**

The index tab allows you to type a keyword and go to only the most applicable topics.

1. Click tab to select index.
2. Type keyword to find topics of interest.
3. View suggested topics. (Double-click to display topic.)
4. Click to display topic.

**Search Tab**

**TIP:** To Search any topic for a keyword, press Ctrl and F.

The following rules apply for using full-text search:

- Searches are not case-sensitive.
- You can search for any combination of letters (a-z) and numbers (0-9).
- Punctuation marks (period, colon, semicolon, comma, and hyphen) are ignored during a search.
- You can group the words of your search using double quotes or parentheses. Examples: "response calibration" or (response calibration). This requirement makes it impossible to search for quotation marks.
- Use Wildcard expressions:
  - To search for one undefined character use a question mark (?). For example, searching for cal? will find calc and calf.
  - To search for more than one undefined character use an asterisk (*). Searching for Cal* will find calibration and calculate.
- Use Boolean operators to define a relationship between two or more search words.
<table>
<thead>
<tr>
<th>Search for</th>
<th>Example</th>
<th>Results will show topics containing:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Two words in the same topic</td>
<td>response AND calibration</td>
<td>Both the words &quot;response&quot; and &quot;calibration&quot;.</td>
</tr>
<tr>
<td>Either of two words in a topic</td>
<td>response OR calibration</td>
<td>Either the word &quot;response&quot; or the word &quot;calibration&quot; or both.</td>
</tr>
<tr>
<td>The first word without the second word in a topic</td>
<td>response NOT calibration</td>
<td>The word &quot;response&quot; but not the word &quot;calibration&quot;.</td>
</tr>
<tr>
<td>Both words in the same topic, close together.</td>
<td>response NEAR calibration</td>
<td>The word &quot;response&quot; within eight words of the word &quot;calibration&quot;.</td>
</tr>
</tbody>
</table>

**Favorites Tab**

The favorites tab allows you to store (bookmark) the topics you refer to most often so that they can be recalled easily.

1. Click tab to view stored topics in Favorites.
2. Remove selected topic.
3. Display selected topic.
4. Add (store) current topic.

**Topic Pane**

The Topic pane allows you to view the contents of the selected topic.
Help Languages
Beginning with PNA Rev A.08.00, PNA Help is offered in English ONLY.

Glossary
The Glossary holds definitions of words, in alphabetical order.

Note: Click on a word in green text throughout Help to see the glossary definition.

Dialog Boxes
Documentation Warranty

THE MATERIAL CONTAINED IN THIS DOCUMENT IS PROVIDED "AS IS," AND IS SUBJECT TO BEING CHANGED, WITHOUT NOTICE, IN FUTURE EDITIONS. FURTHER, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, AGILENT DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED WITH REGARD TO THIS MANUAL AND ANY INFORMATION CONTAINED HEREIN, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. AGILENT SHALL NOT BE LIABLE FOR ERRORS OR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, USE, OR PERFORMANCE OF THIS DOCUMENT OR ANY INFORMATION CONTAINED HEREIN. SHOULD AGILENT AND THE USER HAVE A SEPARATE WRITTEN AGREEMENT WITH WARRANTY TERMS COVERING THE MATERIAL IN THIS DOCUMENT THAT CONFLICT WITH THESE TERMS, THE WARRANTY TERMS IN THE SEPARATE AGREEMENT WILL CONTROL.

Suggestions Please!

Please let us know about your experience using PNA Help. Send your comments to: pna_help@am.exch.agilent.com. Comment about any aspect of the help system. Here are a few areas that you might consider:

- Does anything appear to be broken?
- Did you find what you were looking for?
- Was the information you found helpful?
- Any suggestions as to how we can improve the help system?

Your comments go directly to the help system authors. For help with technical questions, please refer to Technical Support.
Click Help, then About Network Analyzer to learn the following about your PNA:

- Model number (see list of PNA models)
- Frequency range
- Serial number
- Installed options (Learn how to install software options)
- Application Code (firmware) Version
- Hard Disk Drive Version
- System CPU Version - Learn more
- DSP (Digital Signal Processor.) Version. Contact Agilent to upgrade the DSP. See DSP Version changes below.
DSP Version changes

For PNA-X, the DSP board number along with the FPGA version, provides access to the following features or causes the following settings to change:

<table>
<thead>
<tr>
<th>DSP Board</th>
<th>FPGA Version</th>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>27</td>
<td>Enabled <strong>FastCW triggering</strong></td>
</tr>
<tr>
<td>4</td>
<td>34</td>
<td>Enabled <strong>configurable (polarity and type) for PulseSyncIn</strong> external trigger signals.</td>
</tr>
</tbody>
</table>
| 5         | xx*          | **IF Frequencies** – The default values are changed.  
**IFAntiAliasFilter** - The Narrow and Wide settings are changed.  
**Note:** Programs that control the following DSP filter settings, or state files that are saved with these settings, will yield different results when run or recalled on PNAs with DSP 4 versions versus DSP 5 versions.  
- **Stage1 frequency** (Max=38 MHz for DSP 5).  
- **Stage2 Coefficient** setting is IGNORED with DSP 5 Versions.  
- **ADC Capture Mode** Max data points increased to PNA Max. |

*With DSP board 5, FPGA version is upgraded with firmware. This requires more time for FW installation.*

Last Modified:

- 25-Aug-2010    Updated for DSP 5.0 (A.09.30)
- 30-Nov-2009    MX New topic
Preset the PNA

When you Preset the PNA, it is set to known, or preset conditions. You can use the factory default preset conditions, or define your own User Preset conditions.

- Preset (Default) Conditions
- User Preset Conditions

See other 'Setup Measurements' topics

Preset Default Conditions

How to Preset the PNA

Tip: Press the Preset button to start the PNA application if it is not already running.

Using front-panel HARDKEY [softkey] buttons

1. Press Preset

PNA Menu using a mouse

1. Click Utility
2. then Preset

Click to view the factory preset conditions.

- Frequency Settings
- Power Settings
- Sweep Settings
- Segment Sweep Settings
- Trigger Settings
- Display Settings
- Response Settings
- Calibration Settings
- Marker Settings
- Limit Test Settings
- **Time Domain Settings (Option 010)**
- **Global Display Settings**

### Frequency Settings:

<table>
<thead>
<tr>
<th>Measurement Parameter</th>
<th>S11</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start Frequency</td>
<td>Minimum frequency of the PNA</td>
</tr>
<tr>
<td>Stop Frequency</td>
<td>Maximum frequency of the PNA</td>
</tr>
<tr>
<td>CW Frequency</td>
<td>1 GHz</td>
</tr>
</tbody>
</table>

See the [PNA configurations](#) for the minimum and maximum frequency of your PNA.

### Power Settings:

<table>
<thead>
<tr>
<th>Test Port Power</th>
<th>0 dBm for E8356/7/8A</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0 dBm for E8801/2/3A</td>
</tr>
<tr>
<td></td>
<td>0 dBm for N3381/2/3A</td>
</tr>
<tr>
<td></td>
<td>-5 dBm for N5230A - 20 GHz</td>
</tr>
<tr>
<td></td>
<td>-10 dBm for N5230 - 40 GHz</td>
</tr>
<tr>
<td></td>
<td>-15 dBm for N5230 - 50 GHz</td>
</tr>
<tr>
<td></td>
<td>-12 dBm for E8362/3/4 A or B, standard</td>
</tr>
<tr>
<td></td>
<td>-15 dBm for E8361A</td>
</tr>
<tr>
<td></td>
<td>-17 dBm for E8362/3/4 A or B with option UNL or 014</td>
</tr>
<tr>
<td>Power</td>
<td>On - See also <a href="#">Power ON and OFF during Save / Recall, User Preset, and Preset</a></td>
</tr>
<tr>
<td>Port Power Coupled</td>
<td>On</td>
</tr>
<tr>
<td>Auto Attenuation</td>
<td>On</td>
</tr>
<tr>
<td>Attenuator Value</td>
<td>0 dB</td>
</tr>
<tr>
<td>Power Slope</td>
<td>Off</td>
</tr>
<tr>
<td>Slope Value</td>
<td>0 dB/GHz</td>
</tr>
</tbody>
</table>
### Sweep Settings:

<table>
<thead>
<tr>
<th><strong>Type</strong></th>
<th>Linear Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mode</strong></td>
<td>Continuous</td>
</tr>
<tr>
<td><strong>Generation</strong></td>
<td>Analog</td>
</tr>
<tr>
<td><strong>Auto Sweep Time</strong></td>
<td>On</td>
</tr>
<tr>
<td><strong>Number of Points</strong></td>
<td>201</td>
</tr>
</tbody>
</table>

### Segment Sweep Settings:

<table>
<thead>
<tr>
<th><strong>Active Segments</strong></th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Start Frequency</strong></td>
<td>PNA start frequency</td>
</tr>
<tr>
<td><strong>Stop Frequency</strong></td>
<td>1 MHz for E8356/7/8A</td>
</tr>
<tr>
<td></td>
<td>1 MHz for E8801/2/3A</td>
</tr>
<tr>
<td></td>
<td>1 MHz for N3381/2/3A</td>
</tr>
<tr>
<td></td>
<td>1 GHz for E836xA/B</td>
</tr>
<tr>
<td><strong>Number of Points</strong></td>
<td>21</td>
</tr>
<tr>
<td><strong>Power</strong></td>
<td>PNA preset test port power</td>
</tr>
<tr>
<td><strong>IF Bandwidth</strong></td>
<td>50 KHz for N5230A</td>
</tr>
<tr>
<td></td>
<td>35 kHz for all other models</td>
</tr>
<tr>
<td><strong>Reduce IF BW at Low Frequencies</strong></td>
<td>ON</td>
</tr>
<tr>
<td><strong>Dwell Time</strong></td>
<td>0</td>
</tr>
</tbody>
</table>

### Trigger Settings

<table>
<thead>
<tr>
<th><strong>Source</strong></th>
<th>Internal</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mode</strong></td>
<td>Sweep</td>
</tr>
</tbody>
</table>

### Display Settings:

<table>
<thead>
<tr>
<th><strong>Format</strong></th>
<th>Log Mag</th>
</tr>
</thead>
</table>

*These settings apply for formats when selected:*
<table>
<thead>
<tr>
<th>Format</th>
<th>Scale</th>
<th>Reference Position</th>
<th>Reference Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Log Mag</td>
<td>10 dB/5</td>
<td>5</td>
<td>0 dB</td>
</tr>
<tr>
<td>Phase</td>
<td>45 degrees/5</td>
<td>5</td>
<td>0 degrees</td>
</tr>
<tr>
<td>Group Delay</td>
<td>10 nsec/5</td>
<td>5</td>
<td>0 s</td>
</tr>
<tr>
<td>Linear Mag</td>
<td>100 munits/0</td>
<td>0</td>
<td>0 units</td>
</tr>
<tr>
<td>SWR</td>
<td>1 unit/0</td>
<td></td>
<td>1 unit</td>
</tr>
<tr>
<td>Real</td>
<td>2 units/5</td>
<td>5</td>
<td>0 units</td>
</tr>
<tr>
<td>Imaginary</td>
<td>2 units/5</td>
<td>5</td>
<td>0 units</td>
</tr>
<tr>
<td>Polar</td>
<td>1 unit/n/a</td>
<td></td>
<td>1 unit</td>
</tr>
<tr>
<td>Smith Chart</td>
<td>1 unit/n/a</td>
<td></td>
<td>1 unit</td>
</tr>
</tbody>
</table>

### Response Settings:

- **Channel Number**: 1
- **IF Bandwidth**: 50 KHz for N5230A, 35 kHz for all other models
- **Averaging**: Off
- **Averaging Factor**: 1
- **Smoothing**: Off
- **Smoothing Factor**: 1% of span
- **Electrical Delay**: 0 s
- **Velocity Factor**: 1.0
- **Phase Offset**: 0 degrees
- **Math/Memory Trace View**: Data
### Calibration Settings:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Correction State</td>
<td>Off</td>
</tr>
<tr>
<td>Interpolation State</td>
<td>On</td>
</tr>
<tr>
<td>Calibration Type</td>
<td>None</td>
</tr>
<tr>
<td>Cal Kit Number</td>
<td>Current Cal Kit Number</td>
</tr>
<tr>
<td>System Z0</td>
<td>50 ohms</td>
</tr>
<tr>
<td>Port Extensions State</td>
<td>Off</td>
</tr>
<tr>
<td>Port Ext. Values</td>
<td>0</td>
</tr>
<tr>
<td>Input A, B</td>
<td></td>
</tr>
<tr>
<td>Port 1, 2</td>
<td></td>
</tr>
</tbody>
</table>

### Marker Settings:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Frequency</td>
<td>Current Center Frequency</td>
</tr>
<tr>
<td>Reference</td>
<td>None</td>
</tr>
<tr>
<td>Interpolation</td>
<td>On</td>
</tr>
<tr>
<td>Format</td>
<td>Trace Default</td>
</tr>
<tr>
<td>Type</td>
<td>Normal</td>
</tr>
<tr>
<td>Function</td>
<td>Max Value</td>
</tr>
<tr>
<td>Domain</td>
<td>Full Span</td>
</tr>
<tr>
<td>Table</td>
<td>Empty</td>
</tr>
<tr>
<td>Coupling</td>
<td>Always uncoupled</td>
</tr>
</tbody>
</table>

### Limit Test Settings:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Limit Testing</td>
<td>Off</td>
</tr>
<tr>
<td>Line Display</td>
<td>ON</td>
</tr>
<tr>
<td>Sound on Fail</td>
<td>Off</td>
</tr>
</tbody>
</table>
Limit List Settings:

<table>
<thead>
<tr>
<th>Type (OFF, MAX, MIN)</th>
<th>OFF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Begin Stimulus</td>
<td>0</td>
</tr>
<tr>
<td>End Stimulus</td>
<td>0</td>
</tr>
<tr>
<td>Begin Response</td>
<td>0</td>
</tr>
<tr>
<td>End Response</td>
<td>0</td>
</tr>
</tbody>
</table>

Time Domain Settings:

<table>
<thead>
<tr>
<th>Transform State</th>
<th>Off</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transform Mode</td>
<td>Band Pass</td>
</tr>
<tr>
<td>Transform Start</td>
<td>-10 ns</td>
</tr>
<tr>
<td>Transform Stop</td>
<td>10 ns</td>
</tr>
<tr>
<td>Window</td>
<td>6.0 (Kaiser-Bessel factor)</td>
</tr>
<tr>
<td>Gating State</td>
<td>Off</td>
</tr>
<tr>
<td>Gating Start</td>
<td>-10 ns</td>
</tr>
<tr>
<td>Gating Stop</td>
<td>10 ns</td>
</tr>
<tr>
<td>Gate Type</td>
<td>Band Pass</td>
</tr>
<tr>
<td>Gate Shape</td>
<td>Normal</td>
</tr>
</tbody>
</table>

Global Display Settings:

<table>
<thead>
<tr>
<th>Trace Status</th>
<th>On</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency/Stimulus</td>
<td>Off</td>
</tr>
<tr>
<td>Marker Readout</td>
<td>On (when a marker is activated)</td>
</tr>
<tr>
<td>Toolbars Shown</td>
<td>Active Entry</td>
</tr>
<tr>
<td>Status Bar State</td>
<td>ON</td>
</tr>
</tbody>
</table>

User Preset Conditions

The analyzer can be preset to either factory default conditions or User Preset conditions.
How to set User Preset

Using front-panel HARDKEY [softkey] buttons

1. Press **Save**
2. then **[User Preset]**

PNA Menu using a mouse

1. Click **Utility**
2. then **User Preset**

User Preset dialog box help

With a User Preset saved and enabled, when the PNA is Preset, the User Preset settings are recalled instead of the factory default settings. Calibration data is NOT recalled with a User Preset. [Learn more about instrument state settings.](#)

**User Preset Enable**

- **Check** - The PNA is preset to **User Preset** conditions when the Preset button is pressed.
- **Clear** - The PNA is preset to **Default** conditions when the Preset button is pressed.

**Save current state as User Preset**

Click to store the current instrument state as the User Preset conditions. File is stored as C:/ Program Files/ Agilent/ Network Analyzer/ Documents/ UserPreset.sta.

**Load existing file as User Preset**

Click to retrieve an instrument state to be used as the User Preset conditions.

---

Last modified:
9-Mar-2010   Added link to Power ON/OFF
3-Sep-2008   Removed legacy content
  9/27/06   MX Added UI
  9/12/06   Added link to programming commands
Measurement Parameters

This topic contains the following information:

- **S-Parameters** (pre-selected ratios)
- **Ratioed** (choose your own ratio)
- **Unratioed Power** (absolute power)
- **How to Select a Measurement Parameter**

**Learn about Balanced Measurements**

**See other ‘Setup Measurements’ topics**

**S-Parameters**

S-parameters (scattering parameters) are used to describe the way a device modifies a signal. For a 2-port device, there are **four S-Parameters**. The syntax for each parameter is described by the following:

\[
S_{\text{out}-\text{in}}
\]

\[
\text{out} = \text{PNA port number where the device signal output is measured (receiver)}
\]

\[
\text{in} = \text{PNA port number where the signal is applied (incident) to the device (source)}
\]

Move the mouse over each S-parameter to see the signal flow:

**S_{11}**

**Reflection/ Incident**

\[
\text{DUT}
\]

**S_{12}**

**S_{21}**

**S_{22}**

For two-port devices:

- When the source goes into port 1, the measurement is said to be in the **forward** direction.
When the source goes into port 2, the measurement is said to be in the **reverse** direction.

The analyzer automatically switches the source and receiver to make a forward or reverse measurement. Therefore, the analyzer can measure all four S-parameters for a two-port device with a single connection. See the block diagram (including receivers) of your PNA.

### Common Measurements with S-Parameters

#### Reflection Measurements (S11 and S22)
- Return loss
- Standing wave ratio (SWR)
- Reflection coefficient
- **Impedance**
- S11, S22

#### Transmission Measurements (S21 and S12)
- Insertion loss
- Transmission coefficient
- Gain/Loss
- Group delay
- Deviation from linear **phase**
- **Electrical delay**
- S21, S12

### Receiver Measurements

A 2-port PNA typically has four receivers: A, B, R1, and R2. Your PNA may not have 2 reference and 2 test port receivers. See the block diagram of your PNA.

- R1 and R2 are reference receivers. They measure the PNA source signal as it leaves the PNA and is incident on the DUT.
  - R1 measures the signal out of Port 1
  - R2 measures the signal out of Port 2.
- A and B are test port receivers. They measure the signal out (or reflecting off) of the DUT.
  - A measures the signal into PNA Port 1
  - B measures the signal into PNA Port 2

You can specify measurements using one or two of the available receivers.

**Note:** Beginning with PNA Rev. 7.22, you can use the internal ADC (Analog-Digital Converters) as measurement receivers. Learn more.

### Ratioed Measurements

Ratioed measurements allow you to choose your own ratio of any two receivers that are available in your PNA. S-parameters are actually predefined ratio measurements. For example S11 is A/R1.

The following are common uses of ratioed measurements:
Comparing the phase between two paths of a device. An example could be something simple like a power splitter or more complicated like a dual-channel receiver.

Measurements that require a higher dynamic range than the analyzer provides with S-parameters.

Your PNA MAY have front-panel jumper cables that go directly to measurement receivers. Learn about the front-panel jumpers on your PNA.

**Unratioed (Absolute Power) Measurements**

The unratioed power parameter allows you to look at the absolute power going into any of the measurement receivers that are available on your PNA.

The reference receivers are internally configured to measure the source power for a specific PNA port. Performing an absolute power measurement of a reference receiver using a different source port will measure very little power unless the front panel jumpers are removed and signal is applied directly to the receiver. An example of this would be an R1 measurement using port 2 as the source.

- Measuring phase using a single receiver yields meaningless data. Phase measurements must be a comparison of two signals.
- Averaging for Unratioed parameters is computed differently from ratioed parameters.

### How to create a NEW trace

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>For PNA-X and 'C' models</strong></td>
<td></td>
</tr>
<tr>
<td>The only measurements that can be created are those in the same measurement class as is currently assigned to the active channel. To create a measurement other than these, first assign the appropriate measurement class to a new or existing channel. Learn how.</td>
<td></td>
</tr>
<tr>
<td>After that is done...</td>
<td></td>
</tr>
<tr>
<td>1. Press TRACE 1, 2, 3, OR 4</td>
<td>1. Click Trace/Chan</td>
</tr>
<tr>
<td></td>
<td>2. then New Trace</td>
</tr>
</tbody>
</table>

### How to CHANGE the active trace

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Meas</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then select a new parameter</td>
<td>2. then <strong>Measure</strong></td>
</tr>
<tr>
<td></td>
<td>3. then select a new parameter</td>
</tr>
</tbody>
</table>
New / Change Measurement dialog box help

**Note:** The only measurements that are available are those in the [measurement class](#) currently assigned to the active channel. Other measurements are NOT compatible.

To create a measurement other than these, first assign the appropriate measurement class to a new or existing channel. [Learn how](#).

Click a tab to create or change measurements.

- When creating NEW measurements, you can choose more than one.
- When changing an EXISTING measurement, you can choose ONLY one.

**Tabs**

**S-Parameter** Select a predefined ratioed measurements. [Learn more about S-parameters](#).

<table>
<thead>
<tr>
<th>S-Parameter</th>
<th>Balanced</th>
<th>Receivers</th>
</tr>
</thead>
<tbody>
<tr>
<td>S11</td>
<td>S12</td>
<td>S13</td>
</tr>
<tr>
<td>S21</td>
<td>S22</td>
<td>S23</td>
</tr>
<tr>
<td>S31</td>
<td>S32</td>
<td>S33</td>
</tr>
<tr>
<td>S41</td>
<td>S42</td>
<td>S43</td>
</tr>
</tbody>
</table>

**Balanced** Select a balanced measurement type. (Multiport PNAs ONLY)

**Change** Click to invoke the [Balanced DUT Topology / Logical Port mappings](#) dialog box. [Learn more about Balanced Measurements](#).
Receivers  Select receivers to make Ratioed and Unratioed (absolute power) measurements. Learn more about receiver measurements.

Ratioed  Check Activate to create or change a measurement. Select a receiver for the Numerator, select another receiver for the Denominator, then select a source port for the measurement.

The Source port is ALWAYS interpreted as a logical port number.

For convenience, the table is populated with common choices.

- Learn about External Test Sets and Ratioed Measurements
- Learn more about Ratioed Measurements.

Unratioed  Same as Ratioed, but select 1 as the Denominator.

- Learn More about Unratioed Measurements.
- See the block diagram of receivers in YOUR PNA.
- The internal ADCs (Analog-Digital Converters) can be used as measurement receivers. Learn more.
Logical Receiver Notation

With PNA Rev 6.2, Ratioed and Unratioed measurements can also use logical receiver notation to refer to receivers. This notation makes it easy to refer to receivers with an External Test Set connected to the PNA. You do not need to know which physical receiver is used for each test port.

- aN - Reference receiver for logical port N
- bN - Test port receiver for logical port N

For example:

- **Ratioed**: "b12/a1" refers to the logical test port 12 receiver / the logical port 1 reference receiver.
- **Unratioed**: "b10" refers to the logical test port 10 receiver.

The old style notation (A, B, R1 and so forth) can still be used to refer to the PNA physical receivers. However, ratioed measurements MUST use the same notation to refer to both receivers; either the physical receiver notation (A, R1) or the logical receiver notation (aN, bN). For example, the following mixed notation is NOT allowed: A/b3 and a5/R2.

Programming

When entering receiver letters using programming commands, neither logical or physical receiver notation are case sensitive.

Channel / Window Selections

These selections are NOT AVAILABLE when changing an EXISTING measurement. Learn how to change a measurement.

- **Channel Number** Select the channel for the new traces.
- **Create in New Window**
  - Check to create new traces in a new window.
  - Clear to create new traces in the active window. When the PNA traces per window limitation has been reached, no more traces are added.

- **Auto-Create Windows** Check to create new traces in as many windows as necessary. See PNA number of windows limitation.

About Measurement Parameters (top of page)
See this dialog for **Integrated True Mode Stimulus Application** (iTMSA).

Create or edit DUT Topology and Logical Port Mapping.

A Logical Port is a term used to describe a physical PNA test port that has been remapped to a new port number.

- Any Two physical PNA ports are mapped to **One Balanced** Logical port
- Any One PNA physical port is mapped to **One Single-Ended** Logical port

**Note:** These selections apply to ALL measurements in the channel. If the device topology is changed, any existing measurements in the channel that are incompatible with the new topology will be automatically changed to one that is compatible.

**Topology:** Describes your DUT as you would like it tested. The following device topologies can be measured by a multiport PNA.

- **Balanced / Balanced**
  (2 logical ports - <4 actual ports>)

- **Single-ended / Balanced**
  (2 logical ports - <3 actual ports>)

- **Single-ended - Single-ended / Balanced**
  (3 logical ports - <4 actual ports>)

These topologies can be used in the reverse (⇐⇒) direction to measure:

- **Balanced / Single-ended** topology
- **Balanced / Single-ended - Single-ended** topology

For example, to measure a **Balanced / Single-ended** topology, measure the S12 (reverse direction) of a **Single-ended / Balanced** topology.
- Learn about Logical Port mapping when using an External Test Set.
- Learn more about Balanced Measurements
- Balanced parameters can be saved to SNP files. Learn more.

Last modified:

29-Apr-2011  Added images
2-Mar-2010  Link to SNP files
10/11/06  Added new UI
9/19/06  MQ Added logical receiver notation and Multiport meas toolbar.
9/12/06  Added link to programming commands
Measurement Classes

Measurement Classes are categories of measurements that can coexist on a channel.

- What are Measurement Classes
- How to assign a Measurement Class to a Channel
- Measurement Class Dialog Box Help

See other 'Setup Measurements' topics

What are Measurement Classes

The following table shows the Measurement Classes currently available for the PNA. Within each of these classes there are a number of measurements.

Measurement Classes are categories of measurements that can coexist on a channel. A measurement from one class can NOT reside in a channel with a measurement from another class. For example, a Noise Figure measurement can NOT reside in a channel that is currently hosting Scalar Mixer Measurements.

The Measurement Class dialog is accessed in the following ways:

How to assign a Measurement Class to a Channel

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press MEAS</td>
<td>1. Click Trace/Chan</td>
</tr>
<tr>
<td>2. then [Measurement Class]</td>
<td>2. then Measurement Class</td>
</tr>
</tbody>
</table>

Programming Commands
Measurements in a measurement class can NOT coexist in a channel with a measurement of a different measurement class.

Select a measurement class for the active channel or new measurement channel.

- The **Standard** measurement class contains S-Parameters. Balanced parameters, and Receiver measurements.
- All other measurement classes are commonly called "Applications".

**Title Bar** Indicates the active channel to which the measurement class will be assigned.

**Confirm changes**

- Check (default setting) to launch the Confirm Measurement Class Change dialog.
- Clear to perform the 'OK' actions without confirmation. This setting survives a Preset and PNA Shutdown.

**New Channel** Click to create the measurement class in a new channel. A default measurement for that class is created in the channel. To change the measurement, click **Trace**, then select a new measurement.

Choose to do the following:

- **OK** - Delete the existing measurements in the active channel. Create the new measurement class, and
default measurement, in that channel.

- **Cancel** - Do not create the new measurement class. Leave the old measurements (and class) in that channel and return to the Measurement Class dialog box.

Last Modified:

- 6-Aug-2010  Added Confirm changes
- 1-Jul-2010  Added Standard vs Application
- 22-Mar-2010  Updated dialog image
- 3-Sep-2008  Removed legacy content
- 18-Jun-2007  MX New topic
Frequency Range

Frequency range is the span of frequencies you specify for making a device measurement.

- How to Set Frequency Range
- Zoom
- CW Frequencies
- Frequency Resolution
- Frequency Band Crossings

See other 'Setup Measurements' topics

How to set Frequency Range

There are two ways to set the frequency range:

A. Specify the Start and Stop frequencies of the range.
B. Specify the Center frequency and desired Span of the range.

See the frequency ranges of all PNA models

Using front-panel HARDKEY [softkey] buttons

1. Press FREQUENCY

PNA Menu using a mouse

1. Click Stimulus
2. then Frequency

Programming Commands

![Frequency Start/Stop dialog box](image)
**Frequency Start/Stop dialog box help**

**Start**  Specifies the beginning frequency of the swept measurement range.

**Stop**  Specifies the end frequency of the swept measurement range.

---

**Frequency Center/Span dialog box help**

**Center**  Specifies the value at the center of the frequency sweep. This value can be anywhere in the analyzer range.

**Span**  Specifies the span of frequency values measured to either side of the center frequency.

---

**Zoom - PNA-X ONLY**

Zoom allows you to easily change the start and stop frequencies or start and stop power levels in a power sweep. Zoom operates on the Active Trace and all traces in the same channel as the active trace, regardless of the window in which they appear.

---

**How to Zoom in a measurement window**

1. Left-click the mouse or use a finger, then drag across a portion of a trace.

2. Release the mouse or lift the finger and the following menu appears:

3. Select from the following:

   - **Zoom**  - changes the channel stimulus settings to the left and right border values of the Zoom selection
   - **Zoom xy**  - changes the channel stimulus settings as above. In addition, the Y-axis scale of the active trace changes to the approximate scale of the Zoom selection.
   - **Zoom Full Out**  - changes the channel stimulus settings to the full span of the current calibration. If no calibration is ON, then the stimulus settings are changed to the full span of the PNA model.
Notes

- The stimulus settings are changed for **ALL** traces in the active channel, regardless of the window in which they appear.
- If markers are in the selected area, they remain in place.
- If markers are in the unselected area, they are moved to the right or left edge of the new span. When Zoom Full Out is selected, the markers are moved back to their original location.

Zoom is NOT available for the following:

- Smith Chart or Polar display formats
- **CW Time** and Segment sweep type
- Frequency Offset Measurements
- **FCA Opt 083 Measurements**

**CW Frequencies**

Measurements with a **CW Time sweep** or **Power sweep** are made at a single frequency rather than over a range of frequencies.

**How to set CW Frequency**

1. Set **Sweep Type** to **CW Time** or **Power**.

You can also set CW frequency from within the Sweep Type dialog box.

---

**Using front-panel HARDKEY [softkey] buttons**

**For PNA-X and 'C' models**

1. Press **FREQ**
2. then **[CW]**

**PNA Menu using a mouse**

1. Click **Stimulus**
2. then **Frequency**
3. then **CW Frequency**

---

154
Frequency Resolution

The resolution for setting frequency is 1 Hz.

Frequency Band Crossings

The frequency range of the PNA covers several internal frequency bands. The higher the frequency range of the PNA, the larger the number of bands. The source power to your DUT turns off as the stimulus frequency is swept through these band crossings. To learn more, see Power ON and OFF during Sweep and Retrace.

The listed frequencies in the following tables are the stop frequency of the specified band, and the start frequency of the following band.

You can download a PNA Band Structure Readout utility that lists the band crossings for your PNA.

Frequency band crossings are different for the following models:

- 3 GHz, 6 GHz, and 9 GHz Models
- E8362A/B, E8363A/B, E8364A/B
- E8361A
- N5230A (2-port models)
- N5230A (4-port models)
- PNA-X Models

For 3 GHz, 6 GHz, and 9 GHz (discontinued) PNA models:
<table>
<thead>
<tr>
<th>Band</th>
<th>Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10 MHz</td>
</tr>
<tr>
<td>2</td>
<td>748 MHz</td>
</tr>
<tr>
<td>3</td>
<td>1500 MHz</td>
</tr>
<tr>
<td>4</td>
<td>3000 MHz</td>
</tr>
<tr>
<td>5</td>
<td>4500 MHz</td>
</tr>
<tr>
<td>6</td>
<td>6500 MHz</td>
</tr>
</tbody>
</table>

For E8362 / 63 / 64 A/B/C
(A models do not have band 0)

<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>.045</td>
<td>1</td>
<td>0.748</td>
<td>9</td>
<td>7.60</td>
</tr>
<tr>
<td>1</td>
<td>1.500</td>
<td>10</td>
<td>10.00</td>
<td>18</td>
<td>30.00</td>
</tr>
<tr>
<td>2</td>
<td>3.00</td>
<td>11</td>
<td>12.00</td>
<td>19</td>
<td>32.00</td>
</tr>
<tr>
<td>3</td>
<td>3.80</td>
<td>12</td>
<td>12.8</td>
<td>20</td>
<td>36.00</td>
</tr>
<tr>
<td>4</td>
<td>4.50</td>
<td>13</td>
<td>15.20</td>
<td>21</td>
<td>38.40</td>
</tr>
<tr>
<td>5</td>
<td>4.80</td>
<td>14</td>
<td>16.00</td>
<td>22</td>
<td>40.00</td>
</tr>
<tr>
<td>6</td>
<td>6.00</td>
<td>15</td>
<td>20.00</td>
<td>23</td>
<td>45.60</td>
</tr>
<tr>
<td>7</td>
<td>6.40</td>
<td>16</td>
<td>22.80</td>
<td>24</td>
<td>48.00</td>
</tr>
</tbody>
</table>

For E8361A/C
<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.045</td>
<td>9</td>
<td>10.00</td>
<td>18</td>
<td>32.00</td>
</tr>
<tr>
<td>1</td>
<td>0.748</td>
<td>10</td>
<td>12.00</td>
<td>19</td>
<td>36.00</td>
</tr>
<tr>
<td>2</td>
<td>1.500</td>
<td>11</td>
<td>12.80</td>
<td>20</td>
<td>40.00</td>
</tr>
<tr>
<td>3</td>
<td>3.00</td>
<td>12</td>
<td>15.40</td>
<td>21</td>
<td>44.70</td>
</tr>
<tr>
<td>4</td>
<td>3.80</td>
<td>13</td>
<td>16.00</td>
<td>22</td>
<td>46.20</td>
</tr>
<tr>
<td>5</td>
<td>4.00</td>
<td>14</td>
<td>20.00</td>
<td>23</td>
<td>51.20</td>
</tr>
<tr>
<td>6</td>
<td>4.50</td>
<td>15</td>
<td>24.00</td>
<td>24</td>
<td>60.00</td>
</tr>
<tr>
<td>7</td>
<td>6.00</td>
<td>16</td>
<td>25.60</td>
<td>25</td>
<td>64.00</td>
</tr>
<tr>
<td>8</td>
<td>7.70</td>
<td>17</td>
<td>30.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

For N5230A/C 2-port models

<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.045</td>
<td>11</td>
<td>10.500</td>
<td>21</td>
<td>28.600</td>
</tr>
<tr>
<td>2</td>
<td>0.748</td>
<td>12</td>
<td>12.500</td>
<td>22</td>
<td>31.250</td>
</tr>
<tr>
<td>3</td>
<td>1.5</td>
<td>13</td>
<td>15.750</td>
<td>23</td>
<td>31.500</td>
</tr>
<tr>
<td>4</td>
<td>3.125</td>
<td>14</td>
<td>16.667</td>
<td>24</td>
<td>33.333</td>
</tr>
<tr>
<td>5</td>
<td>4.167</td>
<td>15</td>
<td>18.750</td>
<td>25</td>
<td>37.000</td>
</tr>
<tr>
<td>6</td>
<td>5.250</td>
<td>16</td>
<td>21.000</td>
<td>26</td>
<td>40.500</td>
</tr>
<tr>
<td>7</td>
<td>6.250</td>
<td>17</td>
<td>22.500</td>
<td>27</td>
<td>41.667</td>
</tr>
<tr>
<td>8</td>
<td>7.875</td>
<td>18</td>
<td>25.000</td>
<td>28</td>
<td>42.000</td>
</tr>
<tr>
<td>9</td>
<td>8.333</td>
<td>19</td>
<td>26.250</td>
<td>29</td>
<td>46.800</td>
</tr>
<tr>
<td>10</td>
<td>9.375</td>
<td>20</td>
<td>26.500</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

For N5230A/C 4-port models
<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.00112</td>
<td>9</td>
<td>8.333</td>
</tr>
<tr>
<td>1</td>
<td>0.010</td>
<td>10</td>
<td>9.800</td>
</tr>
<tr>
<td>2</td>
<td>0.040</td>
<td>11</td>
<td>10.500</td>
</tr>
<tr>
<td>3</td>
<td>0.748</td>
<td>12</td>
<td>12.500</td>
</tr>
<tr>
<td>4</td>
<td>1.500</td>
<td>13</td>
<td>15.000</td>
</tr>
<tr>
<td>5</td>
<td>3.125</td>
<td>14</td>
<td>15.750</td>
</tr>
<tr>
<td>6</td>
<td>4.166</td>
<td>15</td>
<td>16.666</td>
</tr>
<tr>
<td>7</td>
<td>5.250</td>
<td>16</td>
<td>18.750</td>
</tr>
<tr>
<td>8</td>
<td>6.250</td>
<td>17</td>
<td>20.100</td>
</tr>
</tbody>
</table>

For PNA-X (Band Stop Frequencies)
N5241A and N5242A

<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Reserved</td>
<td>12</td>
<td>.396</td>
<td>24</td>
<td>8.50</td>
</tr>
<tr>
<td>1</td>
<td>Reserved</td>
<td>13</td>
<td>.500</td>
<td>25</td>
<td>10.664</td>
</tr>
<tr>
<td>2</td>
<td>.014</td>
<td>14</td>
<td>.628</td>
<td>26</td>
<td>12.00</td>
</tr>
<tr>
<td>3</td>
<td>.019</td>
<td>15</td>
<td>1.00</td>
<td>27</td>
<td>12.80</td>
</tr>
<tr>
<td>4</td>
<td>.027</td>
<td>16</td>
<td>1.50</td>
<td>28</td>
<td>13.51</td>
</tr>
<tr>
<td>5</td>
<td>.038</td>
<td>17</td>
<td>2.00</td>
<td>29</td>
<td>15.40</td>
</tr>
<tr>
<td>6</td>
<td>.053</td>
<td>18</td>
<td>3.00</td>
<td>30</td>
<td>16.00</td>
</tr>
<tr>
<td>7</td>
<td>.075</td>
<td>19</td>
<td>3.20</td>
<td>31</td>
<td>18.00</td>
</tr>
<tr>
<td>8</td>
<td>.105</td>
<td>20</td>
<td>4.00</td>
<td>32</td>
<td>20.00</td>
</tr>
<tr>
<td>9</td>
<td>.146</td>
<td>21</td>
<td>5.332</td>
<td>33</td>
<td>21.328</td>
</tr>
<tr>
<td>10</td>
<td>.205</td>
<td>22</td>
<td>6.752</td>
<td>34</td>
<td>22.50</td>
</tr>
<tr>
<td>11</td>
<td>.250</td>
<td>23</td>
<td>8.00</td>
<td>35</td>
<td>24.00</td>
</tr>
</tbody>
</table>
### N5244A and N5245A

<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Reserved</td>
<td>12</td>
<td>.396</td>
<td>24</td>
<td>8.50</td>
<td>36</td>
<td>27.008</td>
</tr>
<tr>
<td>1</td>
<td>Reserved</td>
<td>13</td>
<td>.500</td>
<td>25</td>
<td>10.664</td>
<td>37</td>
<td>32.00</td>
</tr>
<tr>
<td>2</td>
<td>.014</td>
<td>14</td>
<td>.628</td>
<td>26</td>
<td>12.00</td>
<td>38</td>
<td>36.50</td>
</tr>
<tr>
<td>3</td>
<td>.019</td>
<td>15</td>
<td>1.00</td>
<td>27</td>
<td>12.80</td>
<td>39</td>
<td>40.50</td>
</tr>
<tr>
<td>4</td>
<td>.027</td>
<td>16</td>
<td>1.50</td>
<td>28</td>
<td>13.51</td>
<td>40</td>
<td>42.656</td>
</tr>
<tr>
<td>5</td>
<td>.038</td>
<td>17</td>
<td>2.00</td>
<td>29</td>
<td>15.40</td>
<td>41</td>
<td>43.50</td>
</tr>
<tr>
<td>6</td>
<td>.053</td>
<td>18</td>
<td>3.00</td>
<td>30</td>
<td>16.00</td>
<td>42</td>
<td>46.20</td>
</tr>
<tr>
<td>7</td>
<td>.075</td>
<td>19</td>
<td>3.20</td>
<td>31</td>
<td>19.00</td>
<td>43</td>
<td>48.00</td>
</tr>
<tr>
<td>8</td>
<td>.105</td>
<td>20</td>
<td>4.00</td>
<td>32</td>
<td>20.00</td>
<td>44</td>
<td>50.20</td>
</tr>
<tr>
<td>9</td>
<td>.146</td>
<td>21</td>
<td>5.332</td>
<td>33</td>
<td>21.328</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>.205</td>
<td>22</td>
<td>6.752</td>
<td>34</td>
<td>24.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>.250</td>
<td>23</td>
<td>8.00</td>
<td>35</td>
<td>26.50</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### N5247A
<table>
<thead>
<tr>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
<th>Band</th>
<th>Freq (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Reserved</td>
<td>12</td>
<td>6.75</td>
<td>24</td>
<td>32.00</td>
</tr>
<tr>
<td>1</td>
<td>Reserved</td>
<td>13</td>
<td>8.00</td>
<td>25</td>
<td>40.00</td>
</tr>
<tr>
<td>2</td>
<td>.053</td>
<td>14</td>
<td>10.70</td>
<td>26</td>
<td>40.50</td>
</tr>
<tr>
<td>3</td>
<td>.175</td>
<td>15</td>
<td>13.50</td>
<td>27</td>
<td>42.70</td>
</tr>
<tr>
<td>4</td>
<td>.250</td>
<td>16</td>
<td>15.40</td>
<td>28</td>
<td>46.20</td>
</tr>
<tr>
<td>5</td>
<td>.500</td>
<td>17</td>
<td>16.00</td>
<td>29</td>
<td>48.00</td>
</tr>
<tr>
<td>6</td>
<td>1.000</td>
<td>18</td>
<td>19.00</td>
<td>30</td>
<td>50.00</td>
</tr>
<tr>
<td>7</td>
<td>2.000</td>
<td>19</td>
<td>20.00</td>
<td>31</td>
<td>54.00</td>
</tr>
<tr>
<td>8</td>
<td>3.000</td>
<td>20</td>
<td>21.30</td>
<td>32</td>
<td>60.00</td>
</tr>
<tr>
<td>9</td>
<td>3.200</td>
<td>21</td>
<td>24.00</td>
<td>33</td>
<td>64.00</td>
</tr>
<tr>
<td>10</td>
<td>4.000</td>
<td>22</td>
<td>26.50</td>
<td>34</td>
<td>67.00</td>
</tr>
<tr>
<td>11</td>
<td>5.330</td>
<td>23</td>
<td>27.00</td>
<td>35</td>
<td>70.00</td>
</tr>
</tbody>
</table>

Last modified:

- 9-Nov-2010  Added 5247A
- 9-Mar-2010  Made minor corrections
- 6-Apr-2009  Modified models
- 3-Sep-2008  Removed legacy content
- 10/23/06   MX Added new band crossings
- 10/16/06   Moved phase lock lost indicator
- 9/12/06    Added link to programming commands
- 9/27/06    MX Added UI
Power Level

Power level is the power of the PNA source at the test ports.

- **How to make Power Settings**
- **Power Dialog**
- **Power and Attenuator Dialog**
- **Source Unleveled**
- **Setting Independent Port Power**
- **Optimum Attenuation Value**
- **Receiver Attenuation**
- **Power ON and OFF during Save / Recall, User Preset, and Preset**
- **Power ON and OFF during Sweep and Retrace**

### See other 'Setup Measurements' topics

**Power Settings**

The test port output power is specified over frequency (See the [Power Range](#) and [Frequency Range](#) specifications for your PNA)

### How to make Power settings

Use one of the following methods to set port power. Only the menu can be used to launch the Power and Attenuators dialog box.

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
</table>
| 1. Press **POWER**  
2. then [Power] or [Power and Attenuators] | 1. Click **Stimulus**  
2. then **Power** or **Power and Attenuators** |

[Programming Commands](#)
Power dialog box help

Defines and controls the PNA source power and attenuation.

**Power On (All Channels)**  Check to enable source power for all channels. Only turns power ON if channel power setting is ON or Auto. See Advanced Power.

**Port 'n’**  Active source port for which power is being set.

- **Port Power**  Sets the power level for the specified port.
  - To accurately set the power level at any point after the test port, perform a Source Power Calibration.
  - See the specified power range of your PNA model.

**Power Sweep**

- **Start / Stop Power**  Set the start and stop power values of a power sweep.
  - These settings are only available when Sweep Type is set to Power Sweep.
  - Uncoupled power sweep power can be set from the Advanced Power dialog box.
  - You can Zoom to easily change the start and stop power levels in a power sweep. Learn how.
  - Learn more about Power Sweep.

**Power Slope**

Helps compensate for cable and test fixture power losses at increased frequency.

- **Slope**  Select to set the power slope. Clear to set power slope OFF. Learn more about power slope.

Power and Attenuators dialog box help
Defines and controls the PNA source power and attenuation for the active channel.

**Beginning with PNA Rev. 7.21,** external sources can be controlled from this dialog. Learn more.

**Power On (All Channels)** Check to enable source power for all channels. Only turns power ON if channel power setting is ON or Auto.

**Port Powers Coupled**

- **Coupled** (checked) The power levels are the same at each test port. Set power at any test port and all test ports change to the same power level.

- **Uncoupled** (cleared) The power levels are set independently for each test port. Uncouple power, for example, if you want to measure the gain and reverse-isolation of a high-gain amplifier. The power required for the input port of the amplifier is much lower than the power required for the output port. A power sweep can also be performed with uncoupled power. Learn more about Setting Independent Port Power

**Name** Lists the PNA test ports.

**State**

- **Auto** Source power is turned ON at the specified test port when required by the measurement. This is the most common (default) setting. See also Power ON and OFF during Save / Recall, User Preset, and Preset.

- **ON** Source power is ALWAYS ON, regardless of measurements that are in process. Use this setting to supply source power to a DUT port that always requires power, such as an LO port. This could turn OFF power at another test port. Learn about internal second source restrictions.

- **OFF** Source power is never ON, regardless of the measurement requirements. Use this setting to prevent damage to a sensitive DUT test port.

**Port Power** Sets the power level at the output of the source.

- To accurately set the power level at any point after the test port, perform a Source Power Calibration.

- See the specified power range of your PNA model.

- See ECal Module Compression Level
Start / Stop Power  Available ONLY when sweep type is set to Power Sweep. Set the start and stop power values of a power sweep. Learn more about Power Sweep.

- In PNA release 6.04 you can specify whether to maintain source power at either the start power or stop power level at the end of a power sweep. Learn more.
- In PNA release 8.20, a power sweep can be performed with uncoupled power. Different power ranges can be swept in the forward and reverse directions.

Auto Range  Check to allow the PNA to select the optimum attenuation value to achieve the specified test port power. Clear to manually set the attenuation for each port. Type or select the attenuation value in the adjacent Attenuator Control box.

Attenuator Control  When Port Powers are Uncoupled, manual attenuator control allows you to set a wide range of power levels by setting the attenuation. See Setting Independent Port Power. Also use manual attenuation control when a measurement requires a very good impedance match with the source, such as with oscillators or conditionally unstable amplifiers. Choose an attenuation level of 10 dB or more to ensure the best source match.

- The PNA does not allow attenuators or other mechanical switches to switch continuously. Learn more.
- When Port Powers are Coupled, changing one port Attenuation Control value changes all port values.
- Attenuators are located between the source and the test port. Power to the reference receiver is not attenuated and is therefore higher than at the test port by the amount of attenuation. This will make uncalibrated measurements that use a reference receiver appear as though there is added attenuation at the test device. See the PNA Block diagram.

Note:  With PNA release 7.2, a preference can be set to mathematically offset (or NOT) the power at the reference receiver by the amount of source attenuation. Because the reference receiver is not in the attenuation path, there is more power at the reference receiver than at the test port by the amount of source attenuation.

By default, ALL PNA models currently offset the reference receivers. See Block diagram. Learn how to set the preference.

Leveling Mode

- Internal  Standard ALC leveling. Power level within an attenuator setting is limited to the ALC Range. See Source Unleveled.
- Receiver Rx  Select a Reference receiver to use for leveling the source. Learn more.
- Open Loop  (Used during pulse conditions with the internal source modulators). NOT available on PNA-C models. No leveling is used in setting the source power. The lowest settable power, without attenuation, is limited to -30dBm. The source power level accuracy is very compromised. Use a source power calibration to make the source power somewhat more accurate.

Channel Power Slope
Helps compensate for cable and test fixture power losses at increased frequency. With power slope enabled, the port output power increases (enter positive value) or decreases (enter negative value) as the sweep frequency increases.

**Slope** Select to set the power slope. Clear to set power slope OFF.

Power slope is computed and applied from 0 GHz – not from the measurement start frequency. For example, with the following measurement settings:

- Start / Stop Freq: 10 GHz to 20 GHz
- Power level: 0 dBm
- Slope: 1 dB/GHz

The power into the DUT from 10 GHz to 20 GHz is 10 dBm sloping to 20 dBm

**Offset and Limits** Launches the Offset and Limits dialog.

**Receiver Leveling** Launches the Receiver Leveling dialog.

**Receiver Attenuator** Launches the Receiver Attenuator dialog.

**Path Configurator** Launches the Path Configurator dialog.

---

**Source Unleveled**

When the power level that is required at a test port is higher than the PNA can supply, a Source Unleveled error message appears on the screen and the letters LVL appear on the status bar.

To perform a power sweep, the range of power is usually limited to the range of the Automatic Leveling Control (ALC) loop. (The PNA-X allows a very wide power range using Open Loop).

PNA specifications guarantee the ALC power range over which the PNA can supply power without an unleveled indication. However, the actual achievable power range on your PNA is probably greater than the specified range.

**How to calculate the specified achievable power range**

From the specifications for the N5230A Opt 245 for the frequency span from 15 GHz to 20 GHz:

- Max Leveled Power = -8 dBm
- Power Sweep Range (ALC) = -17 dB

For this frequency range the specified power range is calculated as:

- Max = -8 dBm
- Min = (-8)-(-17) = -25 dBm

When using Source Attenuators:

- with 10dB of attenuation, this becomes -18 dBm to -35 dBm
with 20dB of attenuation, this becomes **-28 dBm** to **-45 dBm**, and so forth.

See the output power specs for your PNA.

To resolve an unleveled condition, change either the Test Port Power or Attenuator setting. If an Unleveled condition exists within the specified power range, contact Technical Support.

### Setting Independent Port Power

The PNA allows you to **uncouple port power** and specify different power levels at each test port. There are a few things to consider when setting independent port powers.

- Does your required high and low power levels fall within the specified Min and Max power range of the PNA? See Unleveled Indicator. If they do not, you may need to use the internal Source Attenuators.

- Does the PNA have source attenuators? If so, how many source attenuators? Some PNA models have one attenuator for each port. In most multiport PNA systems, the attenuators are shared by at least two test ports. See PNA Options to see the availability and range of source attenuation on your PNA.

**Note:** To prevent premature wear, the PNA does not allow attenuators or other mechanical switches to switch continuously.

These mechanical devices are set for the entire channel. When more than one channel is used, and a mechanical device setting is NOT the same for all channels, only the ACTIVE channel is allowed to sweep. All other channels are NOT allowed to sweep (Blocked).

Learn how to view the settings of all mechanical devices in the PNA.

### Optimum Attenuation Value

The attenuator has different positions, allowing a wide range of power levels. The number of power ranges available is determined by the source attenuation installed in your PNA. See PNA Options to see the availability and range of source attenuation on your PNA.

- Each range has a total specified span (25 dB in the following Attenuation Values graphic).

- The optimum setting is the middle of the range. This range provides the best accuracy and performance of the source leveling system. The optimum ranges are the blue regions in the following graphic.

- An attenuator setting can be selected manually or automatically. If automatic is selected, the blue optimum ranges (shown in the following graphic) are used.

(Attenuator ranges vary, this particular range is 70 dB)
Note: Error correction is fully accurate only for the power level at which a measurement calibration was performed. However, when changing power within the same attenuator range at which the measurement calibration was performed, ratioed measurements can be made with nearly full accuracy (non-ratioed measurements with less accuracy).
Receiver Attenuators dialog box help

Type or select independent attenuation values for each receiver.

Receiver Attenuation, available as option 016 on some PNA models, is used to attenuate the output signal from the device under test to avoid damaging the PNA receivers. The PNA receivers typically start to compress at around +10 dBm. This causes the power level at the receiver to be less than the power at the test port by the specified amount of attenuation.

**Note:** Beginning with PNA release 7.2, a preference can be set to mathematically offset (or NOT) the displayed trace by the amount of receiver attenuation. This causes the display to show the power at the test port.

By default:

- PNA-L and E836xB do NOT offset the display.
- The PNA-X **DOES** offset the display.

Learn how to set the preference.

When an external test set is connected, Receiver Attenuation control is only available for the physical receivers in the PNA. Switching receiver attenuation using **logical receiver notation** is NOT allowed.

**CAUTION!** You can damage the analyzer receivers if the power levels exceed the maximum values. See **Technical Specifications** for the maximum input power to a receiver.

The receiver attenuator characteristics are:

- Range:
  - 0 to 50 dB (E8361A only)
  - 0 to 35 dB (all other applicable PNA models)
- Resolution:
  - 10 dB (E8361A only)
  - 5 dB (all other applicable PNA models)

**Power ON and OFF during Save / Recall, User Preset, and Preset**

To protect your DUT from being inadvertently powered ON, the following RF Power ON/OFF settings occur:

**Instrument State Save/Recall**

If power is OFF when an instrument state is saved, then power will always be OFF after the instrument state is recalled.

If power is ON when an instrument state is saved, and the current power setting is OFF, then power will be OFF after the instrument state is recalled.
**User Preset**

If power is OFF when a User Preset is saved, then power will always be OFF after a User Preset.

If power is ON when a User Preset is saved, and the current power setting is OFF, then power will be OFF after a User Preset.

**Preset**

Instrument Preset sets power ON by default. This can be changed with a Preference setting so that, if the current power setting is OFF, then power will be OFF after Preset.

**Power ON and OFF during Sweep and Retrace**

The frequency range of the PNA covers several internal frequency bands. The higher the frequency range of the PNA, the larger the number of bands. For example, a 9 GHz PNA has 6 frequency bands, a 50 GHz PNA has 25 frequency bands. See the [frequency band crossings](#).

Power to the DUT is turned OFF during band changes to avoid causing power spikes to the DUT.

Retrace occurs when the source gets to the end of your selected frequency span and moves back to the start frequency. Power to the DUT is again turned OFF when retracing across frequency bands.

Therefore, the following occurs for various stimulus settings:

1. **Single band sweep** - The power to the DUT is always ON, even during retrace. Beginning with PNA release 6.04, a preference setting can turn power OFF during a retrace. Only available in single band frequency and segment sweeps.

2. **Multi-band sweep** - The power to the DUT is turned OFF while sweeping across a band crossing. It is turned OFF again during retrace.

3. **Power sweep** - Because power sweep is always done at a single frequency, the frequency is always within a single band and the source power is always ON. At the end of a power sweep, power is immediately set to the start power.
   
   Beginning with PNA release 6.04, this behavior can be changed with a preference setting.

4. **Single sweep:**

   - **Manual trigger mode** - At the end of a multiband sweep, power is turned OFF during retrace, and then power is turned back ON before arming for the next trigger.
   
   - **Hold mode** - Power can be ON or OFF depending on when and how Hold mode is entered. However, power can be immediately turned OFF manually or remotely.

**Caution**: Avoid expensive repairs to your PNA. Read [Electrostatic Discharge Protection](#).
15-Mar-2011  Open Loop Leveling not on C models
10-Mar-2011  Removed ON/OFF restriction
23-Feb-2010  Replaced Hold with Blocked channels
4-Aug-2009  Added Offset and Limits button
9-Feb-2009  Added Receiver Leveling
3-Sep-2008  Removed legacy content
27-Jun-2008  Added link to ECal compression
10-Jun-2008  Clarified power slope
28-May-2008  Updated for uncoupled power sweep
22-Apr-2008  Link to UI preferences
26-Mar-2007  Clarified retrace power OFF
  11/16/06  Added new retrace features
  10/23/06  Modified for new power diag
  10/17/06  Clarified leveling
  9/12/06  Added link to programming commands
Receiver Leveling

Receiver Leveling uses PNA receiver measurements to adjust the source power level across a frequency or power sweep. Before each measurement sweep, a variable number of background sweeps are performed to repeatedly measure power at the receiver for each stimulus point. Those power measurements are then used to adjust the source power level and achieve greater source power level accuracy.

This is similar to a Source Power Calibration, which uses a power meter or PNA receiver to accurately measure source power, but this occurs ONLY during the Source Power Cal. The source power correction values are applied for all subsequent measurement sweeps. Because Receiver Leveling is performed for every measurement sweep, it provides more accurate source power levels, but also takes longer to perform each measurement sweep.

- Receiver Leveling can be used with most sweep types, including Segment sweep and Power sweep. See Wide Power Sweep with Receiver Leveling.
- Receiver Leveling is NOT allowed when Phase Control (Opt 088) is enabled.
- This feature is available for standard S-parameter measurements and with FCA, GCA, and IMD applications.
- Turn ON Receiver Leveling before or after doing a Calibration. When turned ON before calibrating, it is turned OFF during the calibration, then back ON after.

Receiver Leveling can be used for:

- Correcting for short term drift when using an external component, such as a booster amplifier. The booster amplifier must be connected to the front-panel jumpers, in front of the reference receiver. See the Block diagram for your PNA, located at the end of every Specifications document.
- Extending the accuracy of power leveling at very low powers where the internal detector may be too noisy.
- Providing controlled power during Pulsed measurements in an open loop mode.
- Controlling the power at the outputs of MM-Wave heads.

How to make Receiver Leveling settings

Start the Power and Attenuators dialog box as follows:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press POWER</td>
<td>1. Click Stimulus</td>
</tr>
<tr>
<td>2. then [Power and Attenuators]</td>
<td>2. then Power</td>
</tr>
<tr>
<td></td>
<td>3. then Power and Attenuators</td>
</tr>
</tbody>
</table>
Click **Leveling Mode**, then select a receiver to use for leveling. More choices are available on the Receiver Leveling dialog.

Click the **Receiver Leveling** button to launch the Receiver Leveling dialog.

---

**Receiver Leveling** dialog box help

**Overview**

To understand Receiver Leveling, the term ‘Target’ power is defined here to mean the sum of the ‘Port Power’ as shown on the **Power and Attenuators dialog** plus the ‘Power Offset’ as shown on the Receiver Leveling dialog. When Power Sweep is selected, the ‘Port Power’ is the power level at each step (stimulus point). Learn more about Power Offset.

Target Power = Port Power + Power Offset

The goal of Receiver Leveling is to adjust the source power until the measured receiver power is equal to the Target Power.

Leveling sweeps are performed in the background (not visible) before every measurement sweep to measure and apply source correction data.

**Receiver Leveling process:**

- For each leveling sweep, source power is applied at each data point and measured by the specified receiver.

- The deviation is calculated between the measured power and the target power.
The deviation is applied to the current source power, and the updated source power levels are applied on the following leveling sweep.

This process continues until the receiver power at each data point has achieved the target power within the specified tolerance value, or until the specified number of leveling sweeps (iterations) has been reached. See Leveling Properties below.

Select a (Source) Port
Each source port to be leveled is configured individually. In the box at the left of the dialog, select a source port to be configured for receiver leveling.

Receiver Selection

**Receiver**  Select a receiver to be used to level the specified source. Beginning with A.09.33, ratioed receivers can be selected. Use this in conjunction with the Phase Control feature to provide an active load.

To level power at the source output or DUT input choose the reference receiver for the source port. For example, to level the source power at port 1, then choose "R1/1.0". To level power at the DUT output, choose the receiver that is used to measure the DUT output. If the DUT output is connected to port 2, then select "B/1.0".

**Power Offset**  The ‘Port Power’ in the Power and Attenuators dialog is normally used as the target leveling power. However, when there exists an additional attenuator or booster amplifier in the source path, an offset should be set to improve the leveling speed.

Example: On the PNA-X, a 10dB amplifier is connected in the source path (red line) to the rear-panel J10 and J11.

Port Power is set to 0 dBm (out of Source 1).

Power Offset value is set to +10 dB to compensate for the amplifier.

The target power becomes 0 + 10 = 10 dBm.

Source 1 is set to 0 dBm, but the R1 reference receiver expects to see +10 dBm and applies correction based on that value.

Without the Power Offset setting, it would take several leveling sweeps until power was set to the correct value.

**ALC On**  The PNA leveling hardware can be in either:

- **ALC ON** - (checked) Uses Internal leveling hardware (Recommended)
- **ALC OFF** - (cleared) Uses Open Loop hardware (NOT available on PNA-C models).
**Use Last Result for Source Power Cal** - When checked, the latest correction data is copied to the Source Power Cal correction array. When Leveling Mode is switched back to Internal (on the Power and Attenuators dialog), [Source Power Cal](#) is automatically turned ON using this correction data.

When cleared, Source Power Cal is NOT turned ON when Leveling Mode is switched back to Internal.

**Leveling Properties**

The following settings specify the quality and speed of leveling that is to be achieved.

If the maximum port power out of the PNA is reached at any time during the leveling sweeps, the following message appears: **Power set to user power limit**.

**Apply Settings to All Ports** Check to apply the following three settings to all PNA ports.

**Tolerance** The source is considered leveled when each stimulus data point has achieved the target power level +/- (plus or minus) this tolerance value.

**Max Iterations** If every stimulus data point does NOT achieve the target leveling power after this number of leveling sweeps, the measurement sweep occurs using the correction values obtained from the last leveling sweep. The message: **Not settled, noisy trace** appears when the Max Iterations is reached. If you see this message, you can increase the Max Iterations, reduce the IFBW, or increase the Tolerance setting.

**Leveling IFBW** By default, the IFBW for the leveling sweeps is set to 100 kHz. [Learn more about IFBW](#).

- Increase this value to make faster, but noisier leveling sweeps.
- Decrease this value to make slower, more repeatable leveling sweeps.
- Uncheck the box to use the same IFBW as the measurement sweeps.

**Safe Mode**

To protect your DUT, these settings control the extent to which the source power will be changed to achieve the target power as measured at the reference receiver. These settings could be necessary when using external components with a large variation in frequency response (flatness).

**Max Power Step** Limits the change in source power at each data point from one sweep to the next. For example, assume Safe Mode is enabled, and Max Power Step is set to 1 dB. On the first leveling sweep, the first data point measures 3 dB lower than the target power, then source power for data point 1 will be increased by 1 dB for the next sweep, and likely for the following two sweeps.

**Max Port Power** Limits the maximum power out of the source. The message: **Power set to Max Power** appears when this limit is reached.

**Min Port Power** Limits the minimum power out of the source. The message: **Power set to Min Power** appears when this limit is reached.
12-May-2011  Added before or after cal
15-Mar-2011  Added ratioed receivers (9.33) - Open loop not on C models
16-Sep-2010  Removed note about commands for LSPC.
26-Aug-2010  Added link to wide power sweep topic
11-Mar-2010  Added Use Last checkbox (A.09.20)
10-Feb-2009  MX New topic (A.08.50)
# Sweep Settings

A sweep is a series of consecutive data point measurements taken over a specified sequence of stimulus values. You can make the following sweep settings:

- **Sweep Type**
  - Linear / Log
  - Power Sweep
  - CW Time
  - Segment Sweep
  - Phase
- **Sweep Time**
- **Sweep Setup**
  - Stepped vs Analog
  - Fast Sweep
  - Dwell and Delay
  - Standard vs Point Sweep

See [Triggering and other 'Setup Measurements' topics](#)

## How to set Sweep Type

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Sweep</strong></td>
<td>1. Click <strong>Stimulus</strong></td>
</tr>
<tr>
<td>2. then <strong>[Sweep Type]</strong></td>
<td>2. then <strong>Sweep</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Sweep Type</strong></td>
</tr>
</tbody>
</table>
**Sweep Type dialog box help**

**Note:** Sweep Settings are not applied until either **OK** or **Apply** is pressed.

**Channel** The active channel when Sweep Type was selected. Sweep settings will be applied to this channel.

**Sweep Type**

**Linear Frequency** Sets a linear frequency sweep that is displayed on a standard grid with ten equal horizontal divisions.

- **Start** Sets the beginning value of the frequency sweep.
- **Stop** Sets the end value of the frequency sweep.
- **Points** Sets the number of data points that the PNA measures during a sweep. Range: 2 to 20001. (Default is 201).

**Log Frequency** The source is stepped in logarithmic increments and the data is displayed on a logarithmic x-axis. This is usually slower than a continuous sweep with the same number of points.

- **Start** Sets the beginning value of the frequency sweep.
- **Stop** Sets the end value of the frequency sweep.
- **Points** Sets the number of data points that the PNA measures during a sweep. Range: 2 to 20001. (Default is 201).

**Power Sweep** Activates a power sweep at a single frequency that you specify. Learn about power sweep

- **Start** Sets the beginning value of the power sweep.
- **Stop** Sets the end value of the power sweep.
- **CW Frequency** Sets the single frequency where the PNA remains during the measurement sweep.

**CW Time** Sets the PNA to a single frequency, and the data is displayed versus time. Learn more.

- **CW Frequency** Sets the frequency where the PNA remains during the measurement.
- **Sweep Time** Sets the duration of the measurement, which is displayed on the X-axis.
Points  Sets the number of data points that the PNA measures during a sweep. Range: 2 to 20001. (Default is 201).

Segment Sweep  Sets the PNA to sweep through user-defined sweep segments. Learn how to make these settings.

- Independent Power Levels  Check to set the source power level for each segment. Test port uncoupling is also allowed.
- Independent IF Bandwidth  Check to set the IF bandwidth for each segment.
- Independent Sweep Time  Check to set the duration of the measurement for each segment.
- X-Axis Point Spacing  Check to scale the X-Axis to include only the segments. Learn more.
- Allow Arbitrary Segments  Check to allow arbitrary frequencies (overlapped or reverse sweeps). Learn more.
- Show Table  Shows the table that allows you to create and edit segments.
- Hide Table  Hides the segment table from the screen.

Phase  Sweep the phase of one or more sources relative to another source. Choose values between -360° and +360°. Learn more.

- Start  Sets the beginning value of the phase sweep.
- Stop  Sets the end value of the phase sweep.
- CW Frequency  Sets the single frequency where the PNA remains during the sweep.

OK  Applies setting changes and closes the dialog box.
Apply  Applies setting changes and leaves the dialog box open to make more setting changes.
Cancel  Closes the dialog. Setting changes that have been made since the last Apply button click are NOT applied.

Power Sweep

A power sweep either increases or decreases source power in discrete steps. Power sweep is used to characterize power-sensitive circuits, with measurements such as gain compression.

In the Sweep Type dialog, specify Start power, Stop power, and CW Frequency. Power can be swept over any attainable range within the PNA ALC range.

Note: If the PNA has source attenuators, and the attenuation must be changed in order to achieve the requested start and stop power, click Stimulus, then Power then Power and Attenuators.

The PNA does NOT allow a single power sweep over a range that requires attenuator switching. However, two power sweeps can be performed in different channels. The attenuators will not be allowed to switch continuously,
but triggering can be performed using single or group triggering. Learn more.

The remaining power settings apply in power sweep mode:

- Test Port Power setting is not available.
- Port Power can be coupled or uncoupled.
- Attenuator Control is always Manual.
- Power Slope (dB/GHz) is ignored (output frequency is CW).
- Click Stimulus then Sweep, then Number of Points to change the step size of the power sweep.

**Notes:**

- Using a preference setting, you can specify whether to maintain source power at either the start power or stop power level at the end of a power sweep.
- Beginning with PNA Rev. A.08.50, Power Sweep has been optimized for speed. For highest measurement accuracy during a power sweep, it may be necessary to increase the Dwell Time to allow the source more time to settle.
- You may be able to perform a 60 dB power sweep with Receiver Leveling. Learn how.

**Segment Sweep**

Segment Sweep activates a sweep which consists of frequency sub-sweeps, called segments. For each segment you can define independent power levels, IF bandwidth, and sweep time.

Once a measurement calibration is performed on the entire sweep or across all segments, you can make calibrated measurements for one or more segments.

In segment sweep type, the analyzer does the following:

- Sorts all the defined segments in order of increasing frequency
- Measures each point
- Displays a single trace that is a composite of all data taken

Restrictions for segment sweep:

- The frequency range of a segment is not allowed to overlap the frequency range of any other segment.
- The number of segments is limited only by the combined number of data points for all segments in a sweep.
- The combined number of data points for all segments in a sweep cannot exceed 20001.
- All segments are FORCED to have power levels within the same attenuator range to avoid premature wear of the mechanical step attenuator. See Power Level.
How to make segment sweep settings

Using front-panel HARDKEY [softkey] buttons

1. Press **Sweep**
2. then **[Sweep Type]**

PNA Menu using a mouse

1. Click **Stimulus**
2. then **Sweep**
3. then **Sweep Type**

**Insert Segment** - adds a sweep segment before the selected segment. You can also click the “down” arrow on your keyboard to quickly add many segments.

**Delete Segment** - removes the selected segment.

**Delete All Segments** - removes all segments.

**Note**: At least ONE segment must be ON or **Sweep Type** is automatically set to **Linear**.

**To Modify an Existing Segment**

To make the following menu settings available, you must first show the segment table.

Click **View**, point to **Tables**, then click **Segment Table**.

The above graphic shows the Segment table with all independent settings selected, including source power uncoupled (two power settings).

**STATE** Click the box on the segment to be modified. Then use the up / down arrow to turn the segment ON or OFF.

**START** Sets start frequency for the segment. Click the box and type a value and the first letter of a suffix (KHz, Mhz, GHz). Or double-click the box to select a value.

**STOP** Sets stop frequency for the segment. Click the box and type a value and the first letter of a suffix (KHz, Mhz, GHz). Or double-click the box to select a value.

**Note**: The segment table truncates the frequency resolution. To verify the frequency resolution that you input, create a marker at the start or stop frequency settings.

**POINTS** Sets number of data points for this segment. Type a value or double-click the box to select a value.

**To set IFBW, Power, and Sweep Time independently for each segment:**

1. On the **Sweep** menu, click **Sweep Type**, then **Segment Sweep**.
2. Check the appropriate **Sweep Properties** boxes
3. Then click the box and type a value or double-click the box and select a value.

**Note:** If the following are NOT set, the entire sweep uses the channel IFBW, Power, and Time settings.

**IFBW** Sets the IF Bandwidth for the segment.

**POWER** Sets the Power level for the segment. You can also UNCOUPLE the test port power. See Power Coupling.

**TIME** Sets the Sweep time for the segment.

### X-Axis Point Spacing - Segment Sweep ONLY

This feature affects how a segment trace is drawn on the screen.

**How to select X-Axis Point Spacing**

On the **Sweep Type** dialog box, click **Segment Sweep**

Then check **X-Axis Point Spacing**

- **Without X-axis point spacing**, a multi-segment sweep trace can sometimes result in squeezing many measurement points into a narrow portion of the x-axis.

- **With X-axis point spacing**, the x-axis position of each point is chosen so that all measurement points are evenly spaced along the x-axis.

For example, given the following two segments:

<table>
<thead>
<tr>
<th>STATE</th>
<th>START</th>
<th>STOP</th>
<th>POINTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ON</td>
<td>45.000000 MHz</td>
<td>50.000000 MHz</td>
<td>21</td>
</tr>
<tr>
<td>2 ON</td>
<td>170.000000 MHz</td>
<td>180.000000 MHz</td>
<td>21</td>
</tr>
</tbody>
</table>

**Without X-Axis Point Spacing**
With X-Axis Point Spacing

**Arbitrary Segment Sweep**

This feature allows arbitrary frequencies to be entered into the segment sweep table. With this capability, segments can have:

- overlapping frequencies.
- the stop frequency less than the start frequency (reverse sweep).

**How to enable Arbitrary Segment Sweep**

1. On the [Sweep Type](#) dialog box, click [Segment Sweep](#)
2. Check [Allow Arbitrary Segment Sweep](#)

**Notes:**

- Unusual results may occur when using arbitrary sweep segments with markers, display settings, limit lines, formatting, and some calibration features.
- When Allow Arbitrary Segment is checked, [X-axis point spacing](#) is automatically turned ON.

**Sweep Time**

The PNA automatically maintains the fastest sweep time possible with the selected measurement settings. However, you can increase the sweep time to perform a slower sweep.
How to set Sweep Time

Using front-panel HARDKEY [softkey] buttons

1. Press **Sweep**
2. then **[Time]**

PNA Menu using a mouse

1. Click **Stimulus**
2. then **Sweep**
3. then **Sweep Time**

---

**Programming Commands**

---

**Time dialog box help**

**Sweep Time**  Specifies the time the PNA takes to acquire data for a sweep. The maximum sweep time of the PNA is 86400 seconds or 1 day. Learn about other settings that affect sweep speed.

**Note:** If sweep time accuracy is critical, use ONLY the up and down arrows next to the sweep time entry box to select a value that has been calculated by the PNA. Do NOT type a sweep time value as it will probably be rounded up to the closest calculated value. This rounded value will not be updated in the dialog box.

**X-Axis Time**  Set Start and Stop time to be displayed on the X-axis. These settings are ONLY available for Pulse Profile measurements.

- The actual sweep time includes this acquisition time plus some "overhead" time.
- The PNA automatically maintains the fastest sweep time possible with the selected measurement settings. However, you can increase the sweep time using this setting.
- Enter 0 seconds to return the analyzer to the fastest possible sweep time.
- The Sweep Time setting is applied to the active channel.
- The sweep time is per sweep. A full 2-port cal requires two sweeps, both using the specified sweep time. Learn more.
- A **Sweep Indicator** appears on the data trace when the Sweep Time is 0.3 seconds or greater, or if trigger is set to **Point**. The indicator is located on the last data point that was measured by the receiver. If the indicator is stopped (point sweep mode) the source has already stepped to the next data point.
Sweep Setup

How to make Sweep Setup settings

Using front-panel HARDKEY [softkey] buttons

1. Press Stimulus
2. then [Sweep]
3. then [Sweep Setup]

PNA Menu using a mouse

1. Click Stimulus
2. then Sweep
3. then Sweep Setup

Sweep Setup dialog box help

**Channel** Specifies the channel that the settings apply to.

**Stepped Sweep** When checked (Stepped Sweep) the PNA source is tuned, then waits the specified Dwell time, then takes response data, then tunes the source to the next frequency point. This is slower than Analog Sweep, but is more accurate when testing electrically-long devices.

When cleared (Analog Sweep) the PNA takes response data AS the source is sweeping. The sweep time is faster than Stepped, but could cause measurement errors when testing electrically-long devices.

When the dialog checkbox is cleared, the PNA could be in either Analog or Step mode. The mode can change from sweep to sweep. There is **NO way** to determine whether the PNA is in Analog or Stepped Sweep. If you want to be sure what the current sweep mode is, then switch it to Stepped.

Stepped sweep is automatically selected for a number of reasons. Here are some of the reasons:

- **IF Bandwidth** is at, or below, 1 kHz.
- **Source Power Correction** is ON unless doing CW measurement.
- When more than one source is turned ON (multisource PNA models).
When step mode is a faster way to take the data.

- For all FOM and FCA measurements.
- For all ADC measurements.
- For all MMwave measurements.

**Dwell Time**  Specifies the time the source stays at each measurement point before the analyzer takes the data. Only applies to stepped sweep. The maximum dwell time is 100 seconds. See also Electrically Long Devices.

**Sweep Delay**  Specifies the time to wait just before acquisition begins for each sweep. This delay is in addition to Dwell Time (per point) and External Trigger delay if enabled.

**Fast Sweep**  NOT available on N5264A. When checked, in Analog Sweep mode the PNA source settling times are shortened in both frequency and power-control (ALC) circuits. In Stepped Sweep mode, the settling time at ALL data points are shortened. This nearly doubles the sweep speed at preset conditions, but at the expense of frequency accuracy and a few dB of amplitude variation. For ratioed measurements, such as S-Parameters, these errors substantially ratio out.

- By default, Fast Sweep is always OFF to provide maximum accuracy and stability.
- Fast Sweep is available on the N5230A/C 40 & 50 GHz models, and all PNA-X and N522x models.
- Fast Sweep is NOT allowed with Power Limit enabled.
- **Note:** PNA performance specifications do NOT apply in Fast Sweep.

**Alternate Sweeps**  This selection is longer available from the user interface. The inherent crosstalk on the PNA is sufficient without this feature. Remotely, this setting is still available and sweeps can be performed alternately. However, receivers can NOT be turned off individually on the PNA-L, PNA-X, or N522x models.

**Sweep Sequence**

- **Standard Sweep**  When checked, the PNA sweeps all data points for each source port in turn. For a 2-port PNA, this means that all data points are swept in the forward direction, then all data points are swept in the reverse direction. Even when NO reverse parameters are displayed (S22 or S12), reverse measurements are necessary when a full 2-port calibration is correcting the channel. This is the default behavior. Learn more.

- **Point Sweep**  Available ONLY on standard S-parameter channels. When checked, the PNA measures all parameters at each frequency point before stepping to the next frequency. The display trace is updated as each data point is measured. Point sweep is the same as stepped sweep mode on the 8510 and 8530.
27-Oct-2010  Added Phase Sweep and Time diag (9.30)
26-Aug-2010  Added link to Wide Power Sweep
1-Mar-2010   Sweep Delay and Point sweep now on C models
28-Sep-2009  Added segment frequency resolution note.
6-Apr-2009   Replaced N5242A with PNA-X
5-Mar-2009   Added faster power sweep
17-Feb-2009  Added point sweep availability
26-Jan-2009  Removed ‘band-crossings’ at fast sweep.
10-Oct-2008  Added point sweep (8.35)
3-Sep-2008   Removed legacy content
19-Aug-2008  Added Fast Sweep
6-Jun-2008   Added uncoupled power sweep
15-Apr-2008  Updated again for Step mode
14-Nov-2007  Added Alternate Sweep note
21-Jun-2007  Increased max data points
3-May-2007   Updated Step mode conditions
**Trigger**

A trigger is a signal that causes the PNA to make a measurement sweep. The PNA offers great flexibility in configuring the trigger function.

View the interactive [Trigger Model](#) animation to see how triggering works in the PNA.

- **How to Set Trigger**
- **Source**
- **Scope**
- **Channel Settings**
- **Restart**
- **External and Auxiliary Triggering** (separate topic)

**See other ‘Setup Measurements’ topics**

### How to set Triggering

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press TRIGGER</td>
<td>1. Click <strong>Stimulus</strong></td>
</tr>
<tr>
<td>2. then [Trigger...]</td>
<td>2. then <strong>Trigger</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Trigger</strong></td>
</tr>
</tbody>
</table>

**Note:** The **Continuous**, **Single**, and **Hold** settings apply ONLY to the active channel. These settings are available from the Trigger menu, Active Entry keys, and softkeys.
**Trigger Setup** dialog box help

View the interactive [Trigger Model](#) animation to see how triggering works in the PNA.

**Trigger Source**

These settings determine where the trigger signals originate for all existing channels. A valid trigger signal can be generated only when the PNA is not sweeping.

- **Internal**  Continuous trigger signals are sent by the PNA as soon as the previous measurement is complete.
- **Manual** One trigger signal is sent when invoked by the Trigger button, the active toolbar, or a programming command.
- **External** Trigger signals sent out or received from various connectors on the rear panel. Learn more about External and AUX Triggering.

**Manual Trigger!** - Manually sends one trigger signal to the PNA. Available ONLY when Manual trigger is selected.

**Trigger Scope**

These settings determine what is triggered.

- **Global** All channels not in Hold receive the trigger signal [Default setting]
- **Channel** Only the next channel that is not in Hold receives the trigger signal. This is not obvious or useful unless Trigger Source is set to Manual. This setting enables Point Sweep mode.

**Channel Trigger State**

These settings determine how many trigger signals the channel will accept.

- **Continuous** The channel accepts an infinite number of trigger signals.
Groups  The channel accepts only the number of trigger signals that is specified in the Number of Groups text box, then goes into Hold. Before selecting groups you must first increment the Number of Groups text box to greater than one.

Number of Groups  Specify the number of triggers the channel accepts before going into Hold. If in Point Sweep, an entire sweep is considered one group.

First increment to desired number, then select 'Groups'.

Single  The channel accepts ONE trigger signal, then goes into Hold.

Another way to trigger a single measurement is to set Trigger Source to Manual, then send a Manual trigger. However, ALL channels are single triggered.

Hold  The channel accepts NO trigger signals.

Trigger Mode

These settings determine what EACH signal will trigger.

Sweep and Point modes are available ONLY when both Trigger Source = MANUAL or EXTERNAL AND Trigger Scope = CHANNEL.

- Channel  Each trigger signal causes ALL traces in that channel to be swept in the order specified below.

- Point  Each Manual or External trigger signal causes one data point to be measured. Subsequent triggers go to the same trace until it is complete, then other traces in the same channel are swept in the order specified below. When in Groups or Single trigger, the count is decremented by one after ALL data points on ALL traces in the channel are measured. See Also, the (point) Sweep Indicator and SCPI Triggering example for use with External.

- Trace  Available ONLY when Point Sweep is selected. Each trigger signal causes two identical measurements to be triggered separately - one trigger signal is required for each measurement. Other trigger mode settings cause two identical parameters to be measured simultaneously. Trace triggering is NOT permitted when a channel is using a 2 port (or more) S-Parameter calibration.

- Sweep  Each Manual or External trigger signal causes ALL traces that share a source port to be swept in the order specified below. When in Groups or Single trigger, the count is decremented by one after ALL traces in ALL directions are swept.

When multiport correction is ON, which requires sweeps in more than one direction, traces on the screen will not update until all of the relevant directions have been swept. For example, with all four 2-port S-Parameters displayed:

- When Full 2-port correction is ON, trigger 1 causes NO traces to update; trigger 2 causes ALL S-Parameters to update. Learn more about sweeps with correction ON.

- When correction is OFF, trigger 1 causes S11 and S21 to update; trigger 2 causes S22 and S12 to update.
Trace Sweep Order

For ALL Trigger Modes, trigger signals continue in the same channel until all traces in that channel are complete. Triggering then continues to the next channel that is not in HOLD.

Traces within each channel are always swept in the following order:

- Traces are swept sequentially in source-port order. For example, in a channel with all four 2-port S-parameters, first the source port 1 traces (S11 and S21) are swept simultaneously. Then the source port 2 traces (S22 and S12) are swept simultaneously.

- In addition, when Alternate sweep is selected, traces are swept sequentially in source-port / receiver-port order. In the above example, first the S11 trace is swept, then S21, then S12, then S22.

Restart (Available only from the Trigger menu) Channels in Hold are set to single trigger (the channel accepts a single trigger signal). All other settings are unaffected, including decrementing trigger Groups.

See Also

- [External and AUX Triggering](#)
- Interactive [Trigger Model](#) animation

Last modified:

- 10-Oct-2008 Added Trace trigger (8.35)
- 3-Sep-2008 Removed legacy content
- 26-Oct-2007 Added Trigger Mode
- 15-Dec-2006 Added MX capability
- 9/12/06 Added link to programming commands
External and Auxiliary Triggering

External and Auxiliary triggering are both used to synchronize the triggering of the PNA with other equipment or events.

- **Overview**
- **Capability Summary for each PNA Model**
- **Dialogs**
  - **Auxiliary Triggering** (PNA-X and N522x only)
  - **External Trig (IN) Dialog** (All models)
  - **I/O2 Trig Out Dialog** (PNA-L and E836x)

**See Also**

- **Controlling a Handler**
- **Synchronizing an External Source**
- **PNA Triggering**
- **Pulse Triggering**

**Note:** The PNA-X and N522x models have identical External Triggering capability.

---

**Overview**

The manner in which External Triggering is performed has evolved throughout the PNA history, with each new model becoming more comprehensive and flexible. Unfortunately, our ability to update the older models is limited as a large part of external triggering is dependent on the PNA hardware. Where possible, we have updated the capability of the older models with software.

**Ready Signals versus Trigger Signals**

A 'Ready for Trigger' signal is different from a Trigger signal. The ready signal indicates that the instrument sending the signal is ready for measurement. The instrument receiving the ready signal would then send a trigger signal, indicating that the measurement will be made, or has been made. Usually the slower instrument sends the trigger signal. The following two scenarios illustrate when the PNA is faster, and slower than the external device:

- A material handler is very mechanical and takes a relatively long time to load and discharge parts. Therefore, the PNA sends a 'Ready' signal when it is setup to measure, and the handler sends a trigger signal to the PNA when it is ready for a measurement. Additional signals are available on the PNA Handler I/O to indicate that the PNA sweep has ended, and that the handler can setup for the next measurement. See a procedure.

- Alternatively, an external source usually sets up for the next measurement much faster than the PNA. This is
because the PNA acquires data, and moves both source and receivers for the next measurement. In this case, the external source sends a ‘Ready’ signal. The PNA then begins the measurement and sends a trigger signal AFTER the measurement has been made. This indicates that the measurement is complete and that the source should setup for the next measurement. See a procedure. Beginning with A.07.22, the PNA can control an external source from within the firmware. Learn more.

**Capability Summary for each PNA Model**

The following describes the capabilities and recommended method of triggering for each PNA model.

### PNA-X and N522x

These models have the most comprehensive, flexible, and easy to understand of all models in the PNA family. The following are two reciprocal pairs that can be used to accomplish efficient triggering.

<table>
<thead>
<tr>
<th>Signal Pair</th>
<th>Rear-Panel Connectors</th>
<th>Control Settings (click to learn more)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PNA Ready for Trigger (OUT)</td>
<td>MEAS TRIG RDY and Handler I/O p21</td>
<td>Meas Trigger TAB</td>
</tr>
<tr>
<td>Trigger IN to PNA</td>
<td>MEAS TRIG IN and Handler I/O p18</td>
<td>Meas Trigger TAB</td>
</tr>
<tr>
<td>Trigger OUT of PNA</td>
<td>AUX TRIG OUT (1&amp;2)</td>
<td>AUX Trig TAB</td>
</tr>
<tr>
<td>Ext Device Ready (IN to PNA)</td>
<td>AUX TRIG IN (1&amp;2)</td>
<td>AUX Trig TAB</td>
</tr>
</tbody>
</table>

### PNA-L models

The I/O (TRIG IN) and I/O TRIG OUT signal pair is the recommended signal pair to synchronize the PNA-L and external devices. Both signals result in triggering the other instrument; neither of these signals indicate a ‘Ready’ condition.

<table>
<thead>
<tr>
<th>Recommended Signal Pair</th>
<th>Rear-Panel Connectors</th>
<th>Control Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trigger IN to PNA</td>
<td>BNC IN</td>
<td>External TAB</td>
</tr>
<tr>
<td>Trigger OUT of PNA</td>
<td>BNC OUT</td>
<td>I/O Trig TAB</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Other Signals</th>
<th>Rear-Panel Connectors</th>
<th>Control Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>PNA Ready for Trigger (OUT)</td>
<td>AUX I/O p18</td>
<td>SCPI and COM Only</td>
</tr>
<tr>
<td></td>
<td>Handler I/O p21</td>
<td>SCPI and COM Only</td>
</tr>
</tbody>
</table>
### Recommended Signal Pair

<table>
<thead>
<tr>
<th>Signal Pair</th>
<th>Rear-Panel Connectors</th>
<th>Control Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trigger IN to PNA</td>
<td>BNC IN</td>
<td>External TAB</td>
</tr>
<tr>
<td>Trigger OUT of PNA</td>
<td>BNC OUT</td>
<td>I/O Trig Out TAB</td>
</tr>
</tbody>
</table>

### Other Signals

<table>
<thead>
<tr>
<th>Signal Pair</th>
<th>Rear-Panel Connectors</th>
<th>Control Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>PNA Ready for Trigger (OUT)</td>
<td>AUX I/O p18</td>
<td>SCPI and COM Only</td>
</tr>
<tr>
<td>Trigger IN to PNA</td>
<td>AUX I/O p19</td>
<td>SCPI and COM Only</td>
</tr>
<tr>
<td>Ext Device Ready (IN to PNA)</td>
<td>None</td>
<td>N/A</td>
</tr>
</tbody>
</table>

---

**E836xA/B/C**

The I/O (TRIG IN) and I/O TRIG OUT signal pair is the recommended signal pair to synchronize the E836x and external devices. Both signals result in triggering the other instrument; neither of these signals indicate a 'Ready' condition.

See how to access the Trigger Dialog
This reciprocal pair of signals on PNA-X and N522x models offer high flexibility, and robust synchronization with external devices.

- When enabled, the PNA waits indefinitely for a ‘Ready IN’ signal on the AUX IN connector from an external device.
- When received, the PNA is triggered from the usual trigger sources (Internal, External, or Manual).
- The trigger output signal on the AUX OUT connector can be sent BEFORE or AFTER data acquisition.
- Each channel can be configured differently.
- Two pair of AUX TRIG connectors allow two external devices to be controlled simultaneously.

See Also

- See how to use these connectors to synchronize with External Sources.
- To use the opposite reciprocal pair, see Meas Trig IN and Ready OUT pair.
- Pulse Triggering

Dialog Settings

Note: The Aux Trig 1 and Aux Trig 2 tabs are identical.
Enable  Check to use the Aux1 or Aux2 connectors to output signals to an external device.

Channel  All settings on this dialog affect the specified channel ONLY.

**OUT (Trigger)**

After receiving the Aux Trig IN ‘Ready’ signal, the trigger signal comes from any of the following **Trigger Sources**: 

- **Internal** - trigger occurs immediately.
- **Manual** - trigger occurs when the Trigger button is pressed.
- **External** - trigger occurs when Meas Trig In signal is received. This must be configured independently.

The following settings control the properties of the signals sent out the rear panel **AUX TRIG OUT (1&2) connectors**:

**Polarity**

- **Positive Pulse**  Outgoing pulse is positive.
- **Negative Pulse**  Outgoing pulse is negative.

**Position**

- **Before Acquisition**  Pulse is sent immediately before data acquisition begins.
- **After Acquisition**  Pulse is sent immediately after data acquisition is complete.

**Per Point**  Check to cause a trigger output to be sent for each data point. Clear to send a trigger output for each sweep. This setting controls the trigger output signal regardless of the channel **Point trigger** setting, which causes the PNA channel to trigger per point. For example, to trigger the PNA channel per point, and output a trigger signal per point, both this, and the channel setting must be checked ON.

**Pulse Duration**  Specifies the duration of the positive or negative output trigger pulse.

**Ready for Trigger Handshake**

When checked, the PNA waits indefinitely for the input line at the rear panel **AUX TRIG IN (1&2) connectors** to change to the specified level before acquiring data. This signal indicates that the external device is ready for PNA data acquisition. If the signal arrives before the PNA is ready to acquire data, it is latched (remembered). When NOT checked, the PNA does not wait, but outputs trigger signals when the PNA is ready.

This signal does NOT trigger the PNA. The trigger signal is generated from any of the usual sources: Internal, Manual, or External.

**IN (READY)**

- **Ready High**  PNA responds to the leading edge of a pulse on the Aux1 or Aux2 In connector.
- **Ready Low**  PNA responds to the trailing edge of a pulse on the Aux1 or Aux2 In connector.
- **Delay**  Time that the PNA waits after receiving the Handshake input before data acquisition.

See how to access the Trigger Dialog
Dialog box as it appears on a PNA-X and N522x. This tab is labeled **External** on PNA-L and E836x models

### Meas (External) Trigger dialog box help

#### Main Trigger Input

**Global / Channel Trigger Delay** After an external trigger is received, the start of the sweep is held off for this specified amount of time plus any inherent latency.

When **Trigger Scope** = Channel, the delay value is applied to the specified channel.

When **Trigger Scope** = Global, the same delay value is applied to ALL channels.

**Source** The PNA accepts Trigger IN signals through the following rear-panel connectors:

- **Meas Trig IN BNC** (PNA-X and N522x ONLY)
- **Handler I/O Pin 18** (PNA-L, PNA-X, and N522x ONLY)
- **I/O 1 (TRIG IN) BNC** (PNA-L and E836x ONLY)
- **Aux I/O - pin 19** (PNA-L and E836x ONLY)

#### Level / Edge

**High Level** The PNA is triggered when it is armed (ready for trigger) and the TTL signal at the select input is HIGH.

**Low Level** The PNA is triggered when it is armed (ready for trigger) and the TTL signal at the select input is LOW.

**Positive Edge** After the PNA arms, it will trigger on the next positive edge. If **Accept Trigger Before Armed** is set, PNA will trigger as soon as it arms if a positive edge was received since the last data was taken.
**Negative Edge**  After the PNA arms, it will trigger on the next negative edge. If **Accept Trigger Before Armed** is set, PNA will trigger as soon as it arms if a negative edge was received since the last data was taken.

**Note:** Edge triggering is NOT available on the following PNA models: E835xA, E880xA, N338xA, E8362A, E8363A, E8364A.

**Accept Trigger Before Armed**  When checked, as the PNA becomes armed (ready to be triggered), the PNA will immediately trigger if any triggers were received since the last taking of data. The PNA remembers only one trigger signal. All others are ignored.

- When this checkbox is cleared, any trigger signal received before PNA is armed is ignored.
- This feature is only available when positive or negative EDGE triggering is selected.
- Configure this setting remotely using **CONTrol:SIGNal** (SCPI) or **ExternalTriggerConnectionBehavior** (COM).

**Ready for Trigger Indicator**

On the PNA, when Meas Trig IN is enabled, then Meas Trig Ready (OUT) is also enabled. Choose connector to send the PNA Ready OUT signal:

- **Meas Trig RDY**
- **Handler I/O p21**
- **AUX I/O p18**

Choose Polarity of the 'Ready OUT' signal.

This capability is not available on E836x models.

- **Ready High** - TTL High indicates the PNA is ready for trigger
- **Ready Low** - TTL Low indicates the PNA is ready for trigger (default setting).

**See Also**

- **Pulse Trigger Tab**
- **Learn how to External Trigger during Calibration**

See how to access the Trigger Dialog
**I/O2 Trig Out dialog box help**

**This TAB appears ONLY on E836X and PNA-L models with a Trig I/O rear-panel connector.**

**Enable** When checked, the PNA sends synchronized trigger signals out the rear-panel I/O (TRIG OUT) BNC connector.

**Channel**
- **Global** - Trigger output properties apply for ALL channels. This is the default setting and is consistent with pre-7.2 release behavior. In this mode, the Per Point setting (below) is not allowed, but is coupled to the channel **Point trigger** property. In other words, when a channel is in point sweep mode, the trigger output will be sent per point.
- **Channel** Trigger output properties are channel dependent. To output trigger signals for each point, check Per Point (see below).

**Note:** This Channel / Global setting can be changed ONLY by [setting a Preference](#).

The current setting is annotated at the bottom of the dialog as:

Compatibility Mode on: Aux Trigger Scope = global

---

**AUX (I/O) TRIG OUT (To Device)**

- **Polarity** The trigger pulse output from the PNA is either in the Positive or Negative direction.
- **Position** The trigger pulse output is sent either BEFORE or AFTER data is acquired.
**Per Point** (Channel mode only) Check to cause a trigger output to be sent for each data point. Clear to send a trigger output for each sweep. This setting controls the trigger output signal regardless of the channel Point trigger setting, which causes the PNA channel to trigger per point. For example, to trigger the PNA channel per point, and output a trigger signal per point, both this, and the channel setting must be checked ON.

**Pulse Duration** Specifies the duration of the positive or negative output trigger pulse.

**Note:** Beginning with PNA Rev 6.0, Guided and Unguided Calibration CAN be performed in External Trigger mode. With this optional behavior, while Trigger Source is set to External, trigger signals must be sent for Calibration sweeps. This behavior does not apply to FCA calibrations.

You can set a Preference to revert to pre-6.0 behavior - the PNA calibrates using Internal trigger signals while Trigger Source is set to External.

The following dialog box appears on the PNA screen while the PNA is waiting for an External trigger signal.

Click **Abort** to cancel the wait for a trigger signal.
About the trigger model

Read Text description of triggering behaviors.

This model does not include the new Sweep trigger mode.
Data Format

A data format is the way the PNA presents measurement data graphically. Pick a data format appropriate to the information you want to learn about the test device.

- **How to set Format**
- **Rectangular (Cartesian) Display Formats**
- **Polar**
- **Smith Chart**

**See other ‘Setup Measurements’ topics**

---

**How to set the Display Format**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **FORMAT**

**PNA Menu using a mouse**

1. Click **Response**
2. then **Format**
Rectangular Display Formats

Seven of the nine available data formats use a rectangular display to present measurement data. This display is also known as Cartesian, X/Y, or rectilinear. The rectangular display is especially useful for clearly displaying frequency response information of your test device.

- Stimulus data (frequency, power, or time) appears on the X-axis, scaled linearly
- Measured data appears on the Y-Axis.

Log Mag (Logarithmic Magnitude) Format

- Displays Magnitude (no phase)
- Y-axis: dB
- Typical measurements:
  - Return Loss
  - Insertion Loss or Gain

Phase Format

Measures the phase of a signal relative to the calibration reference plane with a range of +/- 180 degrees.

- Displays Phase (no magnitude)
- Y-axis: Phase (degrees)
- The trace 'wraps' every 180 degrees for easier scaling.
- Typical Measurements:
Deviation from Linear Phase

Unwrapped Phase

- Same as Phase, but without 180 degree wrapping.

**Note:** Phase is unwrapped by comparing the phase from one data point to the next. If the phase difference between two points is greater than 180 degrees, or if the phase of the first data point is greater than 180 degrees from DC, than the phase measurement is probably NOT accurate.

Group Delay Format

- Displays signal transmission (propagation) time through a device
- Y-axis: Time (seconds)
- Typical Measurements:
  - Group Delay

**See Also:**
- Group Delay (Measurement)
- Comparing the PNA Delay Functions.
- Phase Measurement Accuracy

Linear Magnitude Format

- Displays positive values only
- Y-axis: Unitless (U) for ratioed measurements
  Watts (W) for unratioed measurements.
- Typical Measurements:
  - reflection and transmission coefficients (magnitude)
  - time domain transfer

SWR Format

- Displays reflection measurement data calculated from the formula \((1+p)/(1-p)\) where \(p\) is reflection coefficient.
- Valid only for reflection measurements.
- Y axis: Unitless
- Typical Measurements:
  - SWR

Real Format
• Displays only the real (resistive) portion of the measured complex data.
• Can show both positive and negative values.
• Y axis: Unitless
• Typical Measurements:
  o time domain
  o auxiliary input voltage signal for service purposes

**Imaginary Format**

• Displays only the imaginary (reactive) portion of the measured data.
• Y-axis: Unitless
• Typical Measurements:
  o impedance for designing matching network

**Polar Format**

Polar format is used to view the magnitude and phase of the reflection coefficient ($\Gamma$) from your S11 or S22 measurement.

You can use Markers to display the following:

• Linear magnitude (in units) or log magnitude (in dB)
• Phase (in degrees)

- The dashed circles represent reflection coefficient. The outermost circle represents a reflection coefficient ($\Gamma$) of 1, or total reflected signal. The center of the circle represents a reflection coefficient ($\Gamma$) of 0, or no reflected signal.
- The radial lines show the phase angle of reflected signal. The right-most position corresponds to zero phase angle, (that is, the reflected signal is at the same phase as the incident signal). Phase differences of 90°,
±180°, and -90° correspond to the top, left-most, and bottom positions on the polar display, respectively.

**Smith Chart Format**

The Smith chart is a tool that maps the complex reflection coefficient (Γ) to the test device's impedance. In a Smith chart, the rectilinear impedance plane is reshaped to form a circular grid, from which the series resistance and reactance can be read (R + jX).

You can use Markers to display the following:

- Resistance (in units of ohms)
- Reactance as an equivalent capacitance (in units of farads) or inductance (in units of henrys)

**Inverse Smith Chart (also known as Admittance)**

Same as standard Smith Chart, except:

- The plot graticule is reversed right-to-left.
- Admittance (in units of siemens) instead of resistance.

**Interpreting the Smith Chart**
Every point on the Smith Chart represents a complex impedance made up of a real resistance \( r \) and an imaginary reactance \( r + jX \).

- The horizontal axis (the solid line) is the real portion of the impedance - the resistance. The center of the horizontal axis always represents the system impedance. To the far right, the value is infinite ohms (open). To the far left, the value is zero ohms (short).
- The dashed circles that intersect the horizontal axis represent constant resistance.
- The dashed arcs that are tangent to the horizontal axis represent constant reactance.
- The upper half of the Smith chart is the area where the reactive component is positive and therefore inductive.
- The lower half is the area where the reactive component is negative and therefore capacitive.

Last modified:

- 4-Jan-2010  Added link to group delay measurement
- 5-Aug-2009  Removed scale
- 6-Oct-2008  Added unwrapped phase note
- 3-Sep-2008  Removed legacy content
- 9/12/06     Added link to programming commands
- 9/27/06     MX Added UI
**Scale**

The Scale, Reference Level and Reference Position settings (along with Format) determine how the data trace appears on the PNA screen.

- **Scale, Reference Level and Position**
- **Scale Coupling**
- **Magnitude Offset**

---

**See other 'Setup Measurements' topics**

---

**Scale, Reference Level and Position**

The Scale, Reference Level and Reference Position settings (along with format) determine how the data trace appears on the PNA screen.

---

**How to set Scale, Reference Level, and Position**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SCALE</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td></td>
<td>2. then <strong>Scale</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Scale</strong></td>
</tr>
</tbody>
</table>

---

**Programming Commands**
Scale dialog box help

**Note:** Beginning with PNA Rev. 9.0, the scale settings are set to couple with other traces in each window. The following settings assume that Scale Coupling is set to OFF. Learn more about Scale Coupling.

**Scale**

- **Per Division** Sets the value of the vertical divisions of a rectangular display format. In Polar and Smith Chart formats, scale sets the value of the outer circumference. Range: 0.001dB/div to 500 dB/div

- **Autoscale** - Automatically sets value of the vertical divisions and reference value to fit the ACTIVE data trace within the grid area of the screen. The stimulus values and reference position are not affected.
  
  The analyzer determines the smallest possible scale factor that will allow all the displayed data to fit onto 80 percent of the vertical grid.
  
  The reference value is chosen to center the trace on the screen.

- **Autoscale All** Automatically scales ALL data traces in the ACTIVE WINDOW to fit vertically within the grid area of the screen.

**Reference**

- **Level** In rectangular formats, sets the value of the reference line, denoted by $0.00\,\text{dB}$ on the PNA screen. Range: -500 dB to 500 dB.
  
  In Polar and Smith chart formats, reference level is not applicable.

- **Position** In rectangular formats, sets the position of the reference line. Zero is the bottom line of the screen and ten is the top line. Default position is five (middle).
  
  In Polar and Smith chart formats, reference position is not applicable.

**Scale Coupling**

With Scale Coupling enabled, traces that have the same format will have the same Scale, Reference Level, and Reference Position. You can choose to couple the scale of traces that are in the same window, couple the scale of all traces in all windows, or to have NO coupling.
How to set Scale Coupling

1. Right-click on the Y-axis labels of a window
2. then select Scale Coupling

OR

Using front-panel HARDKEY [softkey] buttons
1. Press SCALE
2. then More
3. then Scale Coupling

PNA Menu using a mouse
1. Click Response
2. then Scale
3. then Scale Coupling
Scale Coupling dialog box help

Allows traces that share the same format to have the same Scale, Reference Level and Reference Position.

Coupling Method

Off - No coupling. Traces are scaled individually. Default setting.

Window - All traces with the same format in each selected window share the same scale settings.

All - All traces in ALL selected windows with the same format share the same scale settings.

- When Window or All coupling is enabled, the scale settings for the active trace are assumed by other coupled traces with the same format.

- When there are traces with a different format present, all traces with that format assume the trace settings of the lowest-numbered trace of that format.

- Once enabled, scale settings for all coupled traces with the same format can be changed with any coupled trace being active.

Selected Windows

Available when either the Window or All method is selected. Selected windows will participate in scale coupling. All windows are selected by default. Clear a checkbox to 'Opt-out' of scale coupling for that window.

About Autoscale and Scale Coupling

Autoscale (not Autoscale All) affects the active trace in the active window. All traces that are coupled to this trace assume the new scale settings of the active trace. This could cause some traces to NOT show on the screen.

Autoscale All with Coupling Method...

- Off - All traces in the active window are autoscaled independently.

- Window - All traces in each selected window are autoscaled to fit within a common set of scaling factors.

- All - All traces in all selected windows are autoscaled to fit within a common set of scaling factors.

Magnitude Offset

Magnitude Offset allows you to offset the magnitude (not phase) data by a fixed and / or sloped value in dB. If the display format is Linear Magnitude or Real (unitless), the conversion from dB is performed and the correct amount of offset is implemented.
How to set Magnitude Offset

Using front-panel HARDKEY [softkey] buttons

1. Press SCALE
2. then [More]
3. then [Magnitude Offset]

PNA Menu using a mouse

1. Click Response
2. then Scale
3. then Magnitude Offset

Magnitude Offset dialog box help

The Magnitude offset setting affects only the active trace.

Offset  Offsets the entire data trace by the specified value.

Slope  Offsets the data trace by a value that changes with frequency. The offset slope begins at 0 Hz.

For your convenience, the offset value at the start frequency is calculated and displayed.

See where this operation is performed in the data processing chain.
Pre-configured Measurement Setups

- Pre-configured setups for NEW measurements
- Pre-configured arrangements for EXISTING measurements

Before reading this topic, it is important to understand Traces, Channels, and Windows in the PNA.

See other 'Setup Measurements' topics

Pre-configured Setups for NEW Measurements

Each of the following setups creates new traces. Existing traces and their settings will be lost, unless you first save them.

How to select a pre-configured measurement setup

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press RESPONSE</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [Display]</td>
<td>2. then Display</td>
</tr>
<tr>
<td>3. then [Meas Setsups]</td>
<td>3. then Meas Setsups</td>
</tr>
</tbody>
</table>

No programming commands exist for this feature

The following are the four pre-configured measurement setups:

![Setup A](image1.png)  ![Setup B](image2.png)
![Setup C](image3.png)  ![Setup D](image4.png)
Arranging Existing Measurements

The following arrangements place EXISTING measurements into pre-configured Window arrangements using a sort algorithm.

How to select an Existing measurement arrangement

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press Display</td>
<td>1. Click Response</td>
</tr>
<tr>
<td></td>
<td>2. then Display</td>
</tr>
</tbody>
</table>

Overlay Arrangement

This configuration places all existing traces in a single window, all overlaid on each other.

Stack 2 Arrangement

This configuration places all existing traces in two "stacked" windows.

Split 3 Arrangement

This configuration places all existing traces in three windows, two on top and one below.
Quad 4 Arrangement

This configuration places all existing traces in four windows, one window in each screen quadrant.

Sort Algorithm

The sort algorithm for the Arrange Windows feature is designed to:

- Divide traces among windows based on their properties
- Group traces with common properties

The algorithm sorting is based on the following trace properties, in order of priority:

1. Format: circular (polar or Smith) versus rectilinear (log mag, lin mag, group delay, etc.)
2. Channel number
3. Transmission versus reflection

Note: The PNA traces per window limitation overrides this algorithm. An error occurs if the arrange selection cannot be completed with the current number of traces on the screen.

Last modified:
3-Sep-2008   Removed legacy content

9/27/06      MX Added UI

9/12/06      Added link to programming commands
PNA-X RF Path Configurator

Allows you to configure hardware components that are available with selected PNA-X options. N522x models do NOT have the RF Path Configurator.

How to access Path Configurator

Using HARDKEY [softkey] buttons:
1. Press TRACE/CHAN
2. then [Channel]
3. then [Hardware Setup]
4. then [Path Config...]

Using PNA Menu using a mouse:
1. Click Trace/Chan
2. then Channel
3. then Hardware Setup
4. then Path Config...

The following image shows configuration with PNA-X Opt 423 (4-port, internal 2nd source, combiner, and mechanical switches). Your PNA-X may not include these options.

See Also
IF Path Configuration
PNA-X specs for block diagrams
Internal Second Source limitations
PNA Configurations and Options
Path Configuration dialog box help

See Noise Figure tab of the Path Configuration.

**Note:** With selected PNA-X options, pulse modulation is available ONLY on OUT1 of Src 1 and Src 2. See block diagram.

Different paths can be configured for each channel.

**Configuration**

Select, store, and delete factory configurations or user-defined configurations. Configurations are stored on the PNA hard drive.

Any configuration can be saved, and later recalled, from this dialog. Click **Store**, type a configuration name, then click **OK**.

**Text area** Displays text describing the physical connections required to complete the configuration. The text for factory configurations can NOT be edited. Text is saved as part of the configuration.

**Cancel** Closes the dialog and returns the configuration settings to the state they were in when the dialog was opened. Cancel does NOT undo **Store** and **Delete** actions that were performed while the dialog was open.

**Notes**

- Click or touch anywhere within a box on the screen to cycle through the available settings.
- Some switch settings alter graphics in areas other that where the switch is thrown.
- If you don't hear switches clicking, this could be why:
- Electronic switches are **orange** on the path configuration dialog. These switches do not make noise when being thrown. Mechanical switches are **blue**.

- The channel is not sweeping.

- The following selections do NOT throw switches, but simply indicate how to connect jumper cables by drawing red lines on the dialog:
  - Combiner (Normal/Reversed)
  - Port 2 Source (Src2 OUT1/Src1 OUT2)

**Note:** To prevent premature wear, the PNA does not allow attenuators or other mechanical switches to switch continuously.

These mechanical devices are set for the entire channel. When more than one channel is used, and a mechanical device setting is NOT the same for all channels, only the ACTIVE channel is allowed to sweep. All other channels are NOT allowed to sweep (Blocked).

Learn how to [view the settings of all mechanical devices](#) in the PNA.

- Red lines are jumpers on the front or rear panel.

- Notice on the block diagrams:
  - Extra filtering is available to optimize harmonics below 3.2 GHz on OUT1 of both sources. These filters are not used in the Hi Pwr setting.
  - Each source optionally has pulse modulation capability.

  **Note:** Pulse modulation is available ONLY on OUT1 of Src 1 and Src 2. [See block diagram](#). Your PNA must also have Pulse options. [Learn more](#).

- [Copy channel](#) feature also copies path configuration settings.

- Path Configuration is saved and recalled as part of an instrument state.

**Factory Configurations**

**Port 1 2-tone** Routes Source 2 through the internal combiner to create a two-tone signal out port 1. The standard jumper configuration is used. This is the configuration that is used to make [IMD measurements](#).

**Src 2 Out Port 2** (Option 423 ONLY) Routes Source 2 (OUT1) to port 2 using an external cable attached to the rear-panel J8 to J1. This allows source power at port 1 AND port 2 simultaneously. Source power is NOT available at port 3. [Learn more about Internal Second Source capability](#).

**2 port Dual Source** (Option 224 ONLY) Routes Source 2 (OUT1) to port 2 using an external cable attached to the rear-panel J8 to J1. This allows source power at port 1 AND port 2 simultaneously. This also allows pulsed measurements to be performed on both ports 1 and 2.

**Hot S-parameters** The rear panel jumpers must be manually reconfigured to allow the source 2 signal to be routed through the thru path of the internal combiner. Source 2 is used to drive the AUT into compression,
so the highest possible output power is required. Source 1, routed through the coupled path of the combiner, is used for S-parameter measurements, so a small signal is sufficient. These two signals are combined but the frequencies are usually offset.

**See Also**

- [Configuration for High-power measurements](#)
Source Phase Control (Opt 088)

Option 088 allows you to control the phase of a PNA-X source or an External source.

- **Overview**
- **Features and Limitations**
- **Phase Control Use-Cases**
- **How to make Phase Settings**
- **Active Load Pull Examples**

See other 'Setup Measurements' topics

**Overview**

The Source Phase Control feature, allowed on ALL PNA-X models, provides a specific phase difference between two sources. The phase difference can be fixed (for example, at 90 degrees), or swept between two arbitrary phase values (for example, from 0 to 360 degrees).

Any combination of PNA internal or external sources may be used. One source is selected as the controlled source and the other source is the reference source. You select the two sources by selecting the ports at which the sources are available. The choice of ports is limited for you on the Phase Control dialog. Learn about these limitations.

In addition to selecting source ports, you also select the receivers to be used to measure the phase for the sources. This can be test port receivers or the reference receivers for the specified source ports. The receivers measure the relative phase of the sources, then adjust the phase of the controlled source, then remeasure until the phase difference is within the tolerance that you specify.

Phase can also be set without using any receivers. This is called ‘Open Loop’ mode. In this mode, the phase of the controlled source is set once, and iterations are not done, resulting in phase that is less accurate and stable compared to using receivers to measure and set phase. Use Open Loop mode when you need to use the receivers to measure other parameters.

The phase of the controlled source can be swept relative to the reference source. The phase difference between the controlled and reference source is incremented and iterated on consecutive data points. Before starting the Phase Control dialog, select Sweep Type = Phase Sweep. Learn how.

When the phase of a source is controlled, the power of that source is also controlled using Receiver Leveling. Instead of the normal receiver-leveling mode where only one receiver is used, when phase control is active the ratio of two receivers is used to level the power of the controlled source. This is useful for making active load-pull measurements, as described below. In Open Loop mode, neither the phase nor the power of the source is controlled.

**About Phase Control and Error Correction**

Calibrate only those ports that are used for a phase control measurement. For example, if using ports 1 and 3 for phase control, then do NOT calibrate all four ports. If other ports are calibrated, then even ports 1 and 3 may not yield acceptable results.
After performing calibration, the phase is aligned and the power is accurate at the calibration plane.

**Features and Limitations**

- **Phase Control** is allowed ONLY in a standard S-parameter channel.
- Phase Control can be used with [Wideband pulse measurements](#) - NOT in narrow-band pulse mode.
- **Point Averaging** is NOT allowed.
- **External sources** are supported. Learn how to Configure an External Device. Phase can be controlled on Agilent MXG, PSG and ESG sources. The external source must be routed through the rear panel so that a reference receiver can measure its phase. Use the Path Configuration dialog to make switch settings and enable FOM mode.
- **Remote commands** are available that allow the phase and power of each point to be set individually, much like in source power calibration. Use these commands if you need to create a specific pattern of amplitude/phase states, such as characterizing the load-pull of an amplifier.

**Phase Control Use-Cases**

Phase control and phase sweep is useful in the following applications:

**Active load control**

Provide a controlled, electronically-settable impedance to the output port of a DUT under fixed or swept-frequency conditions. Some examples are: measuring the gain and output power of an amplifier with a known load, and measuring the output from a directional detector with a known load.

The reference source is applied to the DUT input port, and the controlled source is applied to the DUT output port as a reverse input wave. The phase and power level of the controlled source is set relative to the forward output wave of the DUT (which is determined by the reference source), so that any arbitrary load impedance (gamma) can be set.

Optionally, the phase of the controlled source can be swept with a constant frequency, so that the phase of gamma rotates while the magnitude of gamma remains constant. The ratio of reverse input wave and forward output wave as viewed on a Smith chart or polar display would appear as a circle. This capability can be combined with external load-pull software to create traditional load-pull power contours.

**Phase-controlled sources**

Set the phase and magnitude of one source relative to a reference source, to provide differential, quadrature, or arbitrary phase-offset signals at a fixed or swept frequency. Typically, another instrument, receiver, or detector would be required to measure the response of the DUT.
How to make Phase Control settings:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Sweep</strong></td>
<td>1. Click <strong>Stimulus</strong></td>
</tr>
<tr>
<td>2. then <strong>[More]</strong></td>
<td>2. then <strong>Sweep</strong></td>
</tr>
<tr>
<td>3. then <strong>[Phase Control]</strong></td>
<td>3. then <strong>More</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>Phase Control</strong></td>
</tr>
</tbody>
</table>

To perform a Phase Sweep:

| 1. Press **Sweep**                         | 1. Click **Stimulus**         |
| 2. then **[Sweep Type]**                  | 2. then **Sweep**             |
| 3. then **[Phase]**                       | 3. then **Sweep Type**        |
|                                            | 4. then **Phase Sweep**      |

To select (view) a Phase format: ([Learn more](#))

| 1. Press **FORMAT**                       | 1. Click **Response**        |
| 2. then **[Phase]**                       | 2. then **Format**           |
|                                            | 3. then **Phase, Unwrapped Phase, or Smith** |
Phase Control dialog box help

Phase Control: Channel 1

Phase Specification

Specify any Fixed Phase setting.

Start / Stop Phase
Available when Sweep Type is set to Phase Sweep. Learn how. Enter the Start and Stop phase values in degrees.

Phase Control
Click in the port cell to be controlled, then choose from the following:

- **Off** - Phase is NOT set or controlled.
- **Open Loop** - Phase and power is set, but receivers are NOT used to measure and iterate the phase or power of the source. Therefore, the setting of phase is not as accurate or stable. Open Loop mode can be used with phase sweep (for example, from 0 to 360 degrees). However, each sweep may not start at 0 degrees. NO settings on the following Phase Control Setup dialog are used in Open Loop.

  **Note:** After selecting Open Loop, set each source to ON (not Auto) using the Power and Attenuators dialog.

- **<rec/rec>, reference port (Controlled)** - Phase and power is measured and iterated to within the specified tolerance. Click Phase Control Setup to specify the source ports, the receivers, and iteration properties to use to control phase. Use the Receiver Leveling feature to set the power iteration properties.

- **Reference** is displayed to indicate the specified reference port.

Phase Control Setup
Click to launch the Phase Control Setup dialog.

Phase Control Setup dialog box help
Select a port to set its phase control configuration  This is the 'controlled' port.

Referenced to  Select a port to be used as a phase reference for the controlled port.

The two internal PNA sources are available ONLY at specific ports. These choices are limited for you on the Phase Control dialog. For example on a 4-port PNA-X, the possible port pairings are: 1/3, 1/4, 2/3, or 2/4. Port 1 can NOT be paired with Port 2, and Port 3 can NOT be paired with Port 4. Learn more about these limitations.

Control Parameter  Select the receivers to be used to measure the phase and power of the sources.

- The LEFT receiver (a4 in the above image) measures the controlled source.
- The RIGHT receiver (b4 in the above image) measures the reference source.

The swept phase or phase offset will be the difference between these two receivers. Use either standard notation (A, R1) or logical receiver notation (a1,b1). Learn more.

Select the receivers based on your application. You are responsible to make sure that your DUT configuration routes the signals of interest to the correct receivers. Otherwise, the phase will not be properly controlled. For example, if you select the configuration in the above dialog, both Port 4 (controlled source) and Port 1 (reference source) must be connected and measured by the Port 4 (a4 and b4) receivers. This would typically be at your DUT output. See the Active Load Pull Example below.

Background Sweep Properties

Background sweeps are phase and power measurements that are made, but the results are not displayed. For each data point, when subsequent measurements are within the specified tolerance, that point is considered settled. If consecutive phase or power measurements of the same data point are NOT within the specified tolerance before the Max Iterations is reached, then one of the following messages are displayed:

- Phase leveling warning: phase not settled.
- Phase leveling warning: power not settled.
- Phase leveling warning: phase and power not settled.

Apply Settings to All Ports  When checked, the specified settings are used for all background sweeps for all phase-controlled ports. When cleared, the following three settings are specified independently for each port pair.

Use Leveling IFBW  (NOT available for this release).

Tolerance  When consecutive phase measurements of the same data point are within this value of each
other, then the phase measurement is considered settled.

**Max Iterations** Sets the maximum number of background phase measurements to perform in order to achieve settling. If the phase is not sufficiently settled after these measurements, then the closest value is used.

---

**Active Load Control Example (4-port PNA-X)**

*See setup using a 2-port 2-source PNA-X.*

1. Select the Phase Control parameters in the above [Phase Control](#) and [Phase Control Setup](#) dialogs.

2. Setup a measurement with the same receivers that are selected on the Phase Control Setup dialog. In this example, on the 'Receivers' measurement tab, select 'a4/b4' as in the following image:

3. Select **Format**, then either **Phase** or **Smith** Chart.

---
4. If you continually see Phase leveling warning: **power** not settled, then on the **Power and Attenuators dialog**, clear Port Powers Coupled (independent port power), then add attenuation to the controlled port. This happens because of additional power being measured from both sources.

**Active Load Control with 2-Port 2-Source PNA-X Models**

1. Connect a rear-panel jumpers cable from J8 to J1.

2. On the Phase Control Setup dialog, select **Port 2** as the controlled source and **Port1 Src 2** as the reference source.

3. On the **Path Configuration dialog**, on the Configuration box, select **2- port dual source**.

4. Select the Control Parameter (receivers) **a2 / b2**.
Phase-Coherent Measurements

Coherent phase means that the measurement does NOT have random phase. Coherent-phase measurements require the comparison of phase between two sources that are sweeping simultaneously. It also requires that two receivers are measuring the two sources simultaneously.

This capability is available in the PNA-X using several applications:

- iTMSA (Opt 460) provides phase-coherent Balanced measurements.
- Phase Control (Opt 088) provides phase-coherent CW or Phase Sweep measurements.
- R/R measurements can provide simple phase-coherent measurements.

How to make phase-coherent R/R (R over R) measurements

To make a phase-coherent R/R measurements, both sources must be ON simultaneously, and the receiver measurements must be made on the same sweep. On a 4-port PNA-X, the two sources are NOT available at all PNA ports simultaneously. It is important to learn these restrictions. Also, both receiver measurements must be made on the same sweep.

1. Create a ratioed receiver measurement using two reference receivers. Learn how. For example, you might create an R1/R3 measurement, specifying the source port as either 1 or 3.

2. The source port that is selected above is turned on automatically. The other source port (port 3 in this case) must be turned ON manually using the Power and Attenuators dialog.

3. Select a phase format. Learn how.
Customize the PNA Screen

You can customize your PNA screen by showing or hiding the following display elements. All of these selections are made from the PNA View menu.

- **Status Bar**
- **Toolbars**
- **Tables**
- **Display Items**
- **Data and Memory Trace**
- **Title Bars**
- **Minimize Application**

See expanded display capabilities of the PNA-X

Learn about using pre-configured measurements and windows arrangements

Learn about Traces, Channels, and Windows on the PNA

See other ‘Setup Measurements’ topics

### Status Bar

<table>
<thead>
<tr>
<th>Status</th>
<th>CH 1:</th>
<th>S11</th>
<th>No Cor</th>
</tr>
</thead>
</table>

When enabled, the status bar is displayed along the bottom of the PNA screen. The primary status bar shows the following:

- **Channel Trigger State** (Cont, Groups, Single, Hold)
- Active channel
- Measurement parameter for the active trace
- **Trace Math**
- **Error correction** for the active trace
- **Averaging Factor** for the active channel
- **Smoothing** Percentage
- **Transform** (On)
- **Gating** (On)
- IF Gating Enabled for **Pulsed App**: (G)
Manual IF Filtering for Pulsed App: (F)
- Delay if invoked using Phase Offset, Electrical Delay, or Port Extensions.
- Loss if invoked using Magnitude Offset or Port Extensions.
- GPIB status: Local (LCL), Remote Talker Listener (RMT), or System Controller (CTL).
- Error Status: (LVL, LCK, etc)

Note: A second level status bar appears when using External Test Set Control or Interface control.

The status bar state (ON or OFF) will not change when the PNA is Preset.

### How to display the Status Bar

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press DISPLAY</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [More]</td>
<td>2. then Display</td>
</tr>
<tr>
<td>3. then [Status Bar]</td>
<td>3. then Status Bar</td>
</tr>
</tbody>
</table>

### Toolbars

You can display up to five different toolbars to allow you to easily set up and modify measurements.

### How to display Toolbars

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press DISPLAY</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [More]</td>
<td>2. then Display</td>
</tr>
<tr>
<td>3. then [Toolbars]</td>
<td>3. then More</td>
</tr>
<tr>
<td></td>
<td>4. then Toolbars</td>
</tr>
</tbody>
</table>

### List of toolbars:
• Entry Toolbar
• Markers
• Measurement
• Sweep Control
• Stimulus
• Time Domain
• Port Extension
• All Off

Note: There is also a Cal Set toolbar available for Monitoring Error Terms.

Entry Toolbar (For PNA-X and 'C' models)

When used with softkeys, this area allows numeric values to be entered for PNA-X settings. From the keyboard, enter G for Giga, M for Mega or milli, K for kilo, and so forth.

Markers Toolbar

The markers toolbar allows you to set up and modify markers. It shows:

• Marker number
• Stimulation value
• Marker functions:
  o Delta
  o Start/Stop
  o Center/Span

Tip: To use the Front Panel Knob to change marker position, first click the Stimulus field of the marker toolbar. Then turn the knob.

Learn more about Markers

Measurement Toolbar

The measurement toolbar allows you to create a new trace for a desired S-parameter measurement in a current window or new window.
Sweep Control Toolbar

In left to right order, the buttons on this toolbar set the active channel to:

- **Hold** mode
- **Single** sweep, then Hold mode
- **Continuous** sweep

Learn more about Channel Trigger State.

Stimulus Toolbar

The stimulus toolbar allows you to view, set up, and modify the sweep stimulus. It shows the:

- **Start** value
- **Stop** value
- **Number of points**

Time Domain

The Time Domain toolbar allows you to do the following:

- Turn Transform and Gating ON / OFF
- Change the Start / Stop times for both Transform and Gating
- **More**...launches the Time Domain Transform dialog box
- **X** Closes the toolbar

The front panel Tab key steps through all of the settings on all of the toolbars on the display. If Tab does not work, press one of the Active Toolbar (color) keys.

Port Extension

The Port Extension toolbar allows you to set Port Extensions while viewing the measurement trace. Learn more about Port Extensions.

All Off
This allows you to **hide all toolbars** with a single selection.

---

**Tables**

Tables are displayed at the bottom of the selected window. Only one table may be displayed at a time for a window.

### How to display tables

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>DISPLAY</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then [<strong>More</strong>]</td>
<td>2. then <strong>Display</strong></td>
</tr>
<tr>
<td>3. then [<strong>Tables</strong>]</td>
<td>3. then <strong>More</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>Tables</strong></td>
</tr>
</tbody>
</table>

### List of tables:

- [Marker Table](#)
- [Limit Line Table](#)
- [Segment Table](#)

### Marker Table

You can display a table of marker settings. These settings include the:

- Marker number
- Marker reference (for delta measurements)
- Frequency
- Time and Distance (for Time Domain measurements)
- Response

Learn more about [Markers](#)

### Limit Line Table

You can display, set up, and modify a table of limit test settings. These include:
- Type (MIN, MAX, or OFF)
- Beginning and ending stimulus values
- Beginning and ending response values

Learn more about Limit Lines

**Segment Sweep Table**

You can display, set up, and modify a table of segment sweep settings. These include:

- State (On/Off)
- Start and Stop frequencies
- Number of Points
- IF Bandwidth (if independent levels)
- Power Level (if independent levels)
- Sweep Time (if independent levels)

Learn more about Segment sweep

**Display Items**

**How to show and hide Display items**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>DISPLAY</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then <strong>[Display Items]</strong></td>
<td>2. then <strong>Display</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Display Items</strong></td>
</tr>
</tbody>
</table>

**Display items:**

- **Trace Status**
- **Frequency Stimulus**
- **Marker (Readout) Display** (separate topic)
- **Grid SOLID | dotted**
Trace Status

For PNA-X and 'C' models
Trace status is annotated at the top of each window.
The highlighted trace number indicates the Active Trace.
Click to select a trace.

Trace Status shows the following:

- Trace number (Tr x). This is the trace number of the channel; NOT the window trace number which is used in many programming commands.
- Measurement parameter. This can be replaced with a custom Trace Title.
- Format
- Scaling factor
- Reference level

How to show/hide Trace Status

Frequency/Stimulus

Ch1: Start 300.000 kHz Stop 3.00000 GHz
Frequency/stimulus information is displayed at the bottom of each window on the screen. It shows:

- Channel number
- Start value
- Stop value

How to show/hide Frequency/Stimulus information

Marker (readout) Display

Learn more

Grid: SOLID | Dotted

Set whether to display ALL open window grid lines in solid or dotted lines. The selected setting is shown in CAPS. Once set, new windows are created using this setting. Grid lines return to SOLID when the PNA is Reset.
Set the color of the grid using Display Colors.
How to display grid settings

Limit Line Test Results
Limit line test results, **Pass** or **Fail**, are displayed on the right side of the designated window.

Limit Lines
Limit lines are displayed for the active trace in the designated window. Their position depends on:

- Limit levels
- Format
- Scaling
- Reference level

Learn more about Limit Lines

How to show/hide Limit Lines and Results

Title
You can create and display a title for each **window** using the keyboard. You can also use the following Title Entry dialog box.

The title is displayed in the upper-left corner of the selected window.

To clear a title, delete the title from the dialog box entry area and click OK.

See also Trace Titles

How to show/hide a Title

Data Trace and Memory Trace
You can view or hide the active data or memory trace.

- Make a trace active by clicking the trace status button
- To view a memory trace you must first store a trace in memory. Click **Trace**, then **Math / Memory**, then **Data => Memory**.

Learn more about Math operations
**Title Bars**

The Title bar shows the window number and Minimize / Maximize icons.

- Checked - Title bars for all PNA windows are shown.
- Cleared - Title bars for all PNA windows are hidden. This allows more room to display measurement results.

### How to show/hide the Title Bars

**Using front-panel HARDKEY [softkey] buttons**

1. Press **DISPLAY**
2. then **[More]**
3. then **[Title Bars]**

**PNA Menu using a mouse**

1. Click **Response**
2. then **Display**
3. then **More**
4. then **Title Bars**

**Minimize Application**

The Network Analyzer application can be minimized to show the desktop and Windows taskbar.

### How to minimize the Network Analyzer Application

**Using front-panel HARDKEY [softkey] buttons**

1. Press **DISPLAY**
2. then **[Windows]**
3. then **[More]**
4. then **[Minimize]**

**PNA Menu using a mouse**

1. Click **File**
2. then **Minimize Application**

To restore the PNA application, click the PNA application on the Windows taskbar.

Last modified:
16-Mar-2010  Added Grid lines (9.2)

3-Sep-2008  Removed legacy content

27-Aug-2007  Edited readout section

  9/12/06  Added link to programming commands

  9/27/06  MX Added UI
Copy Channels

Copy channels allows you to make a duplicate channel of the same Measurement Class and with the same stimulus conditions as an existing channel.

- Why Copy Channels
- How to Copy Channels
- List of Channel Settings

Note: Beginning with A.09.40, Copy Channels CAN be used with PNA Applications, such as FCA, Gain Compression, or Noise Figure.

Other Setup Measurements Topics

Why Copy Channels

Copy channel settings if you need to create several channels that have slightly different settings.

For example, if you have an amplifier that you want to characterize over a frequency span with several different input power levels.

Follow these steps:

1. Create one measurement with your optimized channel settings.
2. Copy that channel to new channels.
3. Change the power level on the new channels.

The alternative to using Copy Channels is to create new default measurements on new channels. Then change every channel setting to your new requirement. This is very time consuming and thus shows the benefit of the Copy Channels feature.
**How to Copy Channels**

**Using front-panel HARDKEY [softkey] buttons**

1. Press TRACE/CHAN
2. then [Channel]
3. then [More]
4. then [Copy Channel]

**PNA Menu using a mouse**

1. Click Trace/Chan
2. then Channel
3. then Copy Channel

---

**Copy Channel** dialog box help

Copies an existing channel's settings to another channel. Measurement traces from the source channel are NOT copied.

**Copy channel** (also known as 'Source' channel): Select a channel to copy.

**to** (also known as 'Destination' channel): Scroll to select a channel to copy settings to. Compatible channel numbers that are currently being used are highlighted. They can be selected and overwritten.

The following are compatible destination channels:

- A channel that does not yet exist. The new channel is created with the channel's default measurement.
- A channel of the same Measurement Class as the source. The existing measurements remain on the destination channel.
- A channel of any Measurement Class that contains no measurements. Again, the destination channel is created with the channel's default measurement.

**Notes:**

- You can copy channel settings to ONLY one new or existing channel. Repeat this operation to copy to more than one channel.
The source channel is ALWAYS copied to the Active window. If you want the destination channel in a separate window, first create a compatible new measurement in a new window. Then make sure it is the Active window before you copy the channel into it.

The measurement in the destination channel becomes the active measurement.

For example:

1. **Source** channel 1: Standard S21 measurement
2. **Destination** NEW channel 2
3. **Result**: Source channel 1, S21 Measurement AND channel 2, S11 measurement. Both with same stimulus settings and in the same window. Channel 2, S11 measurement is the active measurement.

For more information see Traces, Channels, and Windows on the PNA

**List of Channel Settings**

- Frequency Span
- Power
- Cal Set usage
- Source Power Cal data
- IF Bandwidth
- Number of Points
- Sweep Settings
- Average
- **Trigger (some settings)**

Last modified:

14-Jul-2011  Works with Apps (A.09.40)
13-Feb-2008  Added note about Apps
9/12/06      Added link to programming commands
ADC Measurements

The PNA is equipped with one or more ADC (Analog to Digital Converter) inputs. These ADC inputs can be used as measurement receivers and display measurements on the PNA screen.

- Analog Inputs can be used for measuring from -10V to +10V. These inputs can be considered auxiliary receivers and used in a similar way as S-Parameter receivers.
- Analog Output Sense inputs (AOS1 and AOS2) can be used to measure the corresponding DAC outputs.
- Analog Ground input (AIG) can be used to measure the instruments analog ground (PNA-X only).

Supported Hardware

PNA-X:  **Power I/O connector** (Pins 7 and 8)
Other models:  **Aux I/O connector** (Pin 14)

How to create ADC receiver measurements

1. Press **TRACES**
2. then **[New Trace]**

1. Click **Trace/Chan**
2. then **New Trace**

![New Trace](attachment:image.png)
New Trace (ADC) dialog box help

**Note:** Sweep speed slows dramatically when measuring more than two ADC receivers.

On the **New Trace dialog**, click the **Receivers** tab.

**Activate** - check any empty line to create a trace.

**Numerator** - select from the following:

- **AI1** - Input 1
- **AI2** - Input 2 (PNA-X only)
- **AOS1, AOS2** - Output sense 1 or 2
- **AIG** - Analog ground (PNA-X only)

**Denominator** - NOT available (ONLY unratioed measurements)

**Source Port** - The ADC receiver is measured when the specified source port is sweeping.

ADC receiver traces are labeled as shown in the following images:

- The ADC1 input is being measured, with 2 as the source port.
- The Y axis is U (unitless).
- The default trace format is Real (linear).

**ADC Traces and other useful PNA functions**

Although most PNA functions work with ADC traces, the following may be especially useful.

- **Equation Editor** can be used with the trace data. Although the PNA-X ADC is measuring voltage (-10V to +10V range in 14 bits), by using a trace formula, this voltage can represent other types of measurement parameters (such as current, temperature, or a scaled voltage). See PAE example.
- **Trace averaging** and **Trace Smoothing** can be used to remove trace noise.
- **Dwell time** can be used to allow for settling.

**PNA Functions Not Supported**

- Calibration for ADC receivers is NOT supported.
- Use with FCA is NOT supported.
While the PNA is sweeping an ADC measurement, do NOT use the rear-panel Analog I/O SCPI commands.
Dynamic Range

Dynamic range is the difference between the analyzer receiver's maximum input power and the minimum measurable power (noise floor). For a measurement to be valid, input signals must be within these boundaries. Increasing dynamic range is important if you need to measure very large variations in signal amplitude, such as filter bandpass and rejection. The dynamic range is shown below for an example measurement.

To help reduce measurement uncertainty, the analyzer dynamic range should be greater than the response that the DUT exhibits. For example, measurement accuracy is increased when the DUT response is at least 10 dB above the noise floor. The following methods can help you increase the dynamic range.

- **Increase the Device Input Power**
- **Reduce the Receiver Noise Floor**
- **Improving Dynamic Range using the Front-Panel Jumpers**

**Other topics about Optimizing Measurements**

**Increase Device Input Power**

Increase the DUT input power so that the analyzer can more accurately detect and measure the DUT output power. However, use caution - too much power can damage the analyzer receiver or cause compression distortion.

**Caution! Receiver input damage level: +15 dBm.**

See how to increase input power to the device

**Tip:** You can further increase dynamic range by using an external booster amplifier to increase the input power to the DUT. See High Power Amplifier Measurements.

**Reduce the Receiver Noise Floor**

You can use the following techniques to lower the noise floor and increase the analyzer's dynamic range.

- Reduce crosstalk between the PNA receivers when measuring signals close to the noise floor. See Receiver Crosstalk.
- Use **Sweep Averaging** - learn more about Sweep Average
- Reduce the **IF Bandwidth** - learn more about [IF Bandwidth](#).

- In **Segment sweep** mode each segment can have its own IF bandwidth. For example, when measuring a filter:
  
  - In the passband, the IF bandwidth can be set wider for a fast sweep rate, as long as high-level trace noise is kept sufficiently small.
  
  - In the reject band, where noise floor contributes significantly to measurement error, the IF bandwidth can be set low enough to achieve the desired reduction in average noise level.

**Improving Dynamic Range using the Front-Panel Jumpers**

**Direct Access**

The simplest method to improve dynamic range is to remove a RCVR ‘n’ IN front-panel jumper and route the DUT output directly into that PNA receiver. This bypasses the directional coupler and limits the ability to provide Full Error Correction because the signal can not be applied in the reverse direction.

Refer to the [PNA specifications](#) to learn the dynamic range that is available with direct receiver access.

**Reverse the directional coupler**

Another method to improve dynamic range is to reverse the signal path in the test-port coupler and bypass the loss typically associated with the coupled arm.

As shown in the following graphic, the signal is applied to Port 2. The signal bypasses the coupled arm via the jumper cable connected to the Coupler Thru (or Coupler In) and the Receiver B In (or B In) ports.

When making measurements in reverse direction, the system dynamic range is degraded by 15 dB.

**Note:** Your analyzer’s block diagram may contain different components than shown below. To see the block diagram for your PNA model, see the bottom of the [specs/data sheet](#).
See Also

Front-panel Jumpers (image)
Using the Front Panel Jumpers
Specifications

Last Modified:

5-Jan-2011  Removed old E835x std reference
5-Aug-2008  Combined several topics
Number of Points

A data point is a sample of data representing a measurement at a single stimulus value. You can specify the number of data points that the PNA measures across a sweep. (A “sweep” is a series of consecutive data point measurements, taken over a sequence of stimulus values.)

The PNA sweep time changes proportionally with the number of points. However, the overall measurement cycle time does not. See Technical Specifications for more information on how the number of points, and other settings, affect the sweep time.

Note: You may experience a significant decrease in computer processing speed with increased number of points, number of traces, and calibration error terms (full 2-port or 3-port). If this becomes a problem, you can increase the amount of RAM with PNA Option 022.

How to change the number of data points

Select a number or click Custom to invoke a dialog box.

Using front-panel HARDKEY [softkey] buttons

1. Press Sweep
2. then [Number of Points]

Using a mouse with PNA Menus

1. Click Stimulus
2. then Sweep
3. then Number of Points
**Number of Points** dialog box help

Specifies the number of data points that the analyzer gathers during a measurement sweep. You can specify any number from **1** to **32,001**. The default value is 201.

**Note:** Some measurement classes may have different maximum points limitations.

Two data points are required for Time Domain.

**Tips:**

- To achieve the greatest trace resolution, use the maximum number of data points.
- For faster throughput use the smallest number of data points that will give you acceptable resolution.
- To find an optimized number of points, look for a value where there is not a significant difference in the measurement when you increase the number of points.
- To ensure an accurate measurement calibration, perform the calibration with the same number of points that will be used for the measurement.

---

**Last modified:**

- 2-Mar-2009  Increased max
- 3-Sep-2008  Removed legacy content
- 14-Dec-2007  Decreased min to 1
- 21-Jun-2007  MX Increased maximum
- 9/12/06  Added link to programming commands
Phase Measurement Accuracy

You can increase the accuracy of phase measurements by using the following PNA features.

- **Electrical Delay**
- **Phase Offset**
- **Spacing Between Frequency Points (Aliasing)**

**See Also**
- Port Extensions
- Comparing the PNA Delay Functions
- Phase Control
- Phase Coherent Measurements

Learn more about Phase measurements

### Electrical Delay

Electrical delay is a mathematical function that simulates a variable length of lossless transmission line. Use the electrical delay feature to compensate for the linear phase shift through a device. This feature allows you to look at only the deviation from linear phase of the device.

You can set the electrical delay independently for each measurement trace.

**How to set Electrical Delay**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SCALE</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then [<strong>Electrical Delay</strong>]</td>
<td>2. then <strong>Scale</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Electrical Delay</strong></td>
</tr>
</tbody>
</table>
Electrical Delay

Specifies the value of delay added or removed, in Time or Distance. This compensates for the linear phase shift through a device. You can set the electrical delay independently for each measurement trace.

Click the Step icon next to either Time or Distance to start the Step Size dialog.

Velocity Factor

Specifies the velocity factor that applies to the medium of the device that was inserted after the measurement calibration. The value for a polyethylene dielectric cable is 0.66 and 0.7 for PTFE dielectric. 1.0 corresponds to the speed of light in a vacuum.

Velocity factor can also be set from the Port Extensions dialog and Time Domain Distance Marker Settings.

Softkey Display

Allows you to enter delay in either Time or Distance using the softkeys and Active Entry toolbar.

Distance Units

Select from Meters, Inches, or Feet. The step size will not change automatically when this value is changed. Learn more about Step Size.

Media

- Coax Select if the added length is coax. Also specify the velocity factor of the coax.
- Waveguide Select if the added length is waveguide. Also specify the low frequency cutoff of the waveguide.
- Cutoff Freq Low frequency cutoff of the waveguide.

Learn about Electrical Delay (scroll up)
**Step Size** dialog box help

Changes the step size that occurs when the Time or Distance up/down arrows are pressed on the Electrical Delay dialog.

- **Auto**  Step Size is set to the default value.
- **User Defined** Enter a step size value, then click OK.

This value remains the same when the units are changed. For example if a step size of 12 is entered on this dialog, then you change the units from Inches to Feet, the step size of 12 inches becomes 12 feet, not 1 feet. Therefore, change the units first, then set the step size.

**Phase Offset**

Phase offset mathematically adjusts the phase measurement by a specified amount, up to 360°. Use this feature in the following ways:

- **Improve the display of a phase measurement.** This is similar to the way you would change the reference level in an amplitude measurement. Change the phase response to center or align the response on the screen.

- **Emulate a projected phase shift in your measurement.** For example, if you know that you need to add a cable and that the length of that cable will add a certain phase shift to your measurement, you can use phase offset to add that amount and simulate the complete device measurement.

**How to set Phase Offset**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SCALE</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then <strong>[Phase Offset]</strong></td>
<td>2. then <strong>Scale</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Phase Offset</strong></td>
</tr>
</tbody>
</table>

**Programming Commands**

![Phase Offset dialog](image)
**Phase Offset** dialog box help

**Phase Offset** Type a value or use the up and down arrows to select any value up to 360 degrees.

Learn about **Phase Offset** (scroll up)

---

**Spacing Between Frequency Points (Aliasing)**

The analyzer samples data at discrete frequency points, then connects the points, creating a trace on the screen. If the phase shift through a device is >180° between adjacent frequency points, the display can look like the phase slope is reversed. This is because the data is undersampled and aliasing is occurring.

If you are measuring group delay and the slope of the phase is reversed, then the group delay will change sign. For example, the following graphic shows a measurement of a SAW bandpass filter.

- The left measurement has 51 points and indicates the group delay is negative, which is a physical impossibility. That is, the response is below 0 seconds reference line.

- The right measurement shows an increase to 201 points which indicates the group delay is positive. That is, the response is above the 0 seconds reference line.

---

**Tip:** To check if aliasing might be occurring in a measurement, either increase the number of points or reduce the frequency span.

---

Last modified:

- 12-Feb-2009 Added Step Size and Distance
- 3-Sep-2008 Removed legacy content
- Sept.12, 2006 Added link to programming commands
Electrically-Long Device Measurements

A signal coming out of a device under test may not be exactly the same frequency as the signal going in to a device at a given instant in time. This can sometimes lead to inaccurate measurement results. You can choose between two techniques to eliminate this situation and increase measurement accuracy.

- Why Device Delay May Create Inaccurate Results
- Solutions to Increase Measurement Accuracy
  - Slow the Sweep Speed
  - Add Electrical Length to the R Channel

Why Device Delay May Create Inaccurate Results

The following graphic shows an example of this situation:

- In the network analyzer, the source and receiver are phase locked together and sweep simultaneously through a span of frequencies.
- The signal flow through the Device Under Test (DUT) is shown as different colors for different frequencies.
- You can see as a stimulus frequency travels through the DUT, the analyzer tunes to a new frequency just before the signal arrives at the receiver. This causes inaccurate measurement results.

If the analyzer is measuring a long cable, the signal frequency at the end of the cable will lag behind the network analyzer source frequency. If the frequency shift is appreciable compared to the network analyzer's IF detection bandwidth (typically a few kHz), then the measured result will be in error by the rolloff of the IF filter.

**Note:** There is no fixed electrical length of a device where this becomes an issue. This is because there are many variables that lead to measurement speed. When high measurement accuracy is critical, lower the sweep speed until measurement results no longer change.

Solutions to Increase Measurement Accuracy
Choose from the following methods to compensate for the time delay of an electrically long device.

**Slow the Sweep Speed**

The following methods will slow the sweep speed.

- **Increase the Sweep Time**
- **Increase the Number of Points**
- **Use Stepped Sweep**
- **Set Dwell Time**

**Add Electrical Length to the R Channel**

*Note:* This method applies to PNA models with front panel loops.

Instead of slowing the sweep, you can compensate for the electrical length of a cable or fixture.

a. Remove the R-channel jumper on the front panel of the analyzer.

b. Replace the jumper with a cable of about the same length as the device under test.

1. Add the cable on the R1 channel for S11 and S21 measurements.
2. Add the cable on the R2 channels for S22 and S12 measurements.

b. Set the analyzer for a fast sweep.

**Configuration for S22 and S12 Measurements**

This method balances the delays in the reference and test paths, so that the network analyzer’s ratioed transmission measurement does not have a frequency-shift error.

*Note:* This method works well if the delay is in a cable or fixture. For devices with long delays, this method is only suitable for uncalibrated measurements.
Reflection Accuracy on Low-Loss 2-Port Devices

To make accurate reflection measurements that have a 1-port calibration, you should terminate the unmeasured port.

- Why Terminate the Unmeasured Port
- How to Terminate the Unmeasured Port
- Resulting Measurement Uncertainty

Why Terminate the Unmeasured Port

A 2-port calibration corrects for all 12 twelve error terms. A 1-port calibration corrects for directivity, source match and frequency response, but not load match. Therefore, for highest accuracy, you must make the load match error as small as possible. This especially applies for low-loss, bi-directional devices such as filter passbands and cables. You do not need to be concerned with load match when you are measuring a device with high reverse isolation, such as an amplifier.

How to Terminate the Unmeasured Port

Use one of the following methods:

- Connect a high-quality termination load (from a calibration kit, for example) to the unmeasured port of your device. This technique yields measurement accuracy close to that of a Full SOLT 2-port calibration.

- Connect the unmeasured port of your device directly to the analyzer, inserting a 10 dB precision attenuator between the device output and the analyzer. This improves the effective load match of the analyzer by approximately twice the value of the attenuator, or 20 dB.

Resulting Measurement Uncertainty

The following graph illustrates the measurement uncertainty that results from terminating with and without a
precision 10 dB attenuator on the output of the test device.

Legend

- Filter Reflection
- Uncertainty with attenuator
- Uncertainty without attenuator

The calculations below show how adding a high-quality 10 dB attenuator improves the load match of the analyzer.

**Note:** The corresponding linear value is shown in parentheses.

**Network Analyzer:**

Load match (NALM) = 18 dB (.126)

Directivity (NAD) = 40 db (.010)

**Filter:**

Insertion loss (FIL) = 1dB (.891)

Return loss (FRL) = 16 dB (.158)

**Attenuator:**

Insertion loss (AIL) = 10 dB (.316)

\[
\text{SWR (ASWR)} = 1.05 (.024)
\]

32.26 dB Return Loss

**Calculations:**
## Without Attenuator

\[ \rho_{\text{NA}} = (FIL) \cdot (N_{\text{ALM}}) \cdot (FIL) \]
\[ = (.891) \cdot (.126) \cdot (.891) \]
\[ = .100 \]

\[ \rho_{\text{Attenuator}} = \text{NA} \]

\[ \text{Worst Case Error (EWC)} = \rho_{\text{NA}} \]
\[ = .1 \]

\[ \text{Uncertainty Adds} = -20 \log_{10} (FRL) + (EWC) + (N_{\text{AD}}) \]
\[ = -20 \log_{10} (.158) + (.100) + (.010) \]
\[ = \text{11.4 dB} \]

## With Attenuator

\[ = (FIL) \cdot (AIL) \cdot (N_{\text{ALM}}) \cdot (AIL) \cdot (FIL) \]
\[ = (.891) \cdot (.316) \cdot (.126) \cdot (.316) \cdot (.891) \]
\[ = .010 \]

\[ \rho_{\text{Attenuator}} = \text{NA} \]

\[ \text{Worst Case Error (EWC)} = \rho_{\text{NA}} + \rho_{\text{Att}} \]
\[ = .01 + .019 \]
\[ = .029 \]

\[ \text{Uncertainty Adds} = -20 \log_{10} (FRL) + (EWC) + (N_{\text{AD}}) \]
\[ = -20 \log_{10} (.158) + (.029) + (.010) \]
\[ = \text{14.1 dB} \]

\[ \text{Uncertainty Subtracts} = -20 \log_{10} (FRL) - (EWC) - (N_{\text{AD}}) \]
\[ = -20 \log_{10} (.158) - (.029) - (.010) \]
\[ = \text{18.5 dB} \]

Last Modified: 10-Mar-2010

Fixed Aswr- thanks Alex!
Measurement Stability

There are several situations that can cause unstable measurements. To ensure that you are making repeatable measurements, you can use various methods to create a stable measurement environment.

- **Frequency Drift**
- **Temperature Drift**
- **Inaccurate Measurement Calibrations**
- **Device Connections**

Other topics about Optimizing Measurements

**Frequency Drift**
The analyzer frequency accuracy is based on an internal 10 MHz frequency oscillator. See Technical Specifications for stability and aging specifications.

If your measurement application requires better frequency accuracy and stability, you can override the internal frequency standard and provide your own high-stability external frequency source through the 10 MHz Reference Input connector on the rear panel.

**Temperature Drift**
Thermal expansion and contraction changes the electrical characteristics of the following components:

- Devices within the analyzer
- Calibration kit standards
- Test devices
- Cables
- Adapters

To reduce the effects of temperature drift on your measurements, do the following.

- Switch on the analyzer 1/2 hour before performing a measurement calibration or making a device measurement.
- One hour before you perform a measurement calibration, open the case of the calibration kit and take the standards out of the protective foam.
- Use a temperature-controlled environment. All specifications and characteristics apply over a 25 °C ±5 °C range (unless otherwise stated).
- Ensure the temperature stability of the calibration kit devices.
● Avoid handling the calibration kit devices unnecessarily during the calibration procedure.

● Ensure the ambient temperature is ±1°C of the measurement calibration temperature.

Inaccurate Measurement Calibrations

If a measurement calibration is inaccurate, you will not measure the true response of a device under test. To ensure that your calibration is accurate, you should consider the following practices:

● Perform a measurement calibration at the points where you connect the device under test, that is, the reference plane.

● If you insert any additional accessory (cable, adapter, attenuator) to the test setup after you have performed a measurement calibration, use the port extensions function to compensate for the added electrical length and delay.

● Use calibration standards that match the definitions used in the calibration process.

● Inspect, clean, and gage connectors. See Connector Care.

See Accurate Measurement Calibrations for more detailed information.

Device Connections

Good connections are necessary for repeatable measurements. To help make good connections, do the following:

● Inspect and clean the connectors for all of the components in the measurement setup.

● Use proper connection techniques.

● Avoid moving the cables during a measurement.
Noise Reduction Techniques

Random electrical noise which shows up in the analyzer receiver chain can reduce measurement accuracy. The following PNA functions help reduce trace noise and the noise floor which can lead to better dynamic range and more accurate measurements.

**Note:** The trace noise in microwave PNAs becomes worse below 748 MHz and is especially obvious between 10 MHz and 45 MHz. See [Reduce IFBW](#).

- **Averaging**
- **IF Bandwidth**
- **Trace Smoothing**

**See Also**
- [Group Delay](#)
- [Increase Dynamic Range](#)
- PNA data processing map.

### Other topics about Optimizing Measurements

**Averaging**

Averaging is a feature that reduces the effects of random noise on a measurement. The PNA computes each data point based on the average of several measurements. You determine the number of measurements by setting the Average factor. The higher the average factor, the greater the amount of noise reduction.

**Effects of Sweep Average**

- **Sweep averaging** - consecutive measurements occur over a number of sweeps.
- **Point averaging** - consecutive measurements occur immediately before moving to the next data point. Point averaging is usually much faster than sweep averaging. NOT available on 'C' models and [Gain Compression](#) or [Noise Figure](#) Apps.
- An **Average Counter** appears on the screen when Sweep Averaging is selected, displaying the number of sweeps that has been averaged. The effect on the signal trace can be viewed as the Average Factor increases. This can assist in the selection of the optimum number of sweep averages.
• **Channel wide** - Averaging is applied to all measurements in a channel. The Average counter is displayed for each channel.

• **Unratioed** measurements - Although you can average unratioed (single receiver) measurements, you may get unexpected results:
  
  o Phase results may tend toward 0. This is because phase measurements are relative by nature. Measuring absolute phase with a single receiver appears random. Averaging random positive and negative numbers will tend toward 0.
  
  o The noise floor does not drop when averaging unratioed measurements as on ratioed measurements.

• **Average vs IF Bandwidth** - Both can be used for the same benefit of general noise reduction. For minimizing very low noise, using Average is more effective than reducing IF bandwidth. Generally, Averaging takes slightly longer than IF bandwidth reduction to lower noise, especially if many averages are required. Also, changing the IF bandwidth after calibration results in **uncertain accuracy**.

• **Calibration** - Because averaging is a mathematical process that occurs after the raw measurement is made, averaging can be turned ON before, or after, calibration without invalidating the error correction terms. If averaging is ON before calibration, the measurement of calibration standards are averaged measurements. More time is needed to perform the calibration, but there will be less noise in the resulting error correction terms. Subsequent corrected measurements will also have less noise error. In addition, noise is further reduced by turning Averaging ON after calibration. See the PNA data processing map.

**Averaging Formula**

\[ \text{NewAvg} = \frac{\text{NewData}}{n} + \left( \frac{\text{OldAvg} \times (n-1)}{n} \right) \]

where \(n = \text{average factor}\)

From the formula, you can see that data from the first \(n\) sweeps continues to be included in the results of subsequent sweeps. Its effect is increasingly smaller but never diminishes to zero. For example, with \(n = 5\), the average of the 5 sweeps is displayed. On the 6th sweep, you see \(4/5\) the average of the first 5 sweeps plus \(1/5\) the new sweep.

The effects of older data can be eliminated by restarting the average after \(n\) sweeps.

**How to Set Averaging**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Avg</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then <strong>[Averaging]</strong></td>
<td>2. then <strong>Avg</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Average</strong></td>
</tr>
</tbody>
</table>
### Average dialog box help

**Average ON**
- Checked - Averaging is applied
- Cleared - Averaging is NOT applied

**Average Factor**  Specifies the number of measurements that are averaged. Range of 1 to 65536 (2^16).

**Average Type**
- **Sweep**  Each data point is based on the average of the same data point measured over several consecutive sweeps.
- **Point**  Each data point is based on the average of the same data point measured repeatedly before going to the next data point. NOT available on Gain Compression App.

**Restart**  Applies only to Sweep averaging - NOT Point. Begins a new set of measurements that are used for the average.

[Learn more about Averaging](scroll up)

---

### IF Bandwidth

The PNA converts the received signal from its source to a lower intermediate frequency (IF). The bandwidth of the IF bandpass filter is adjustable from 40 kHz (for most PNA models) down to a minimum of 1 Hz.

Reducing the IF receiver bandwidth reduces the effect of random noise on a measurement. Each tenfold reduction in IF bandwidth lowers the noise floor by 10 dB. However, narrower IF bandwidths cause longer sweep times.

- **Channel wide**  - IF bandwidth can be set independently for each channel
- **Segment sweep**  - IF bandwidth can be set independently for each segment of segment sweep.
- **Calibration**  - Changing the IF bandwidth after calibration will cause a 'C-delta' correction level, which means that calibration accuracy is uncertain.

### Effect of Reducing IF Bandwidth

---

265
How to set IF Bandwidth

Using front-panel HARDKEY [softkey] buttons

1. Press *Avg*
2. then [IF Bandwidth]

Using a mouse with PNA Menus

1. Click *Response*
2. then *Avg*
3. then *IF Bandwidth*

**IF Bandwidth** dialog box help

**IF Bandwidth** specifies the IF (receiver) bandwidth. The value of IF bandwidth is selected by scrolling through the values available in the IF bandwidth text box. The IF BW is set independently for each channel.

The list of selectable IF Bandwidths is different depending on PNA model.

The following values are common to all models:

- 1 | 2 | 3 | 5 | 7 | 10 | 15 | 20 | 30 | 50 | 70 | 100 | 150 | 200 | 300 | 500 | 700 | 1k | 1.5k | 2k | 3k | 5k | 7k | 10k | 15k | 20k | 30k

In addition, the following values are PNA Model specific:

5230A Opt 020, 025, 120, 125, 140, 145, 146, 240, 245, 146

- 50k | 70k | 100k | 150k | 200k | 280k | 360k | 600k

N5230A Opt 220, 225, 420, 425, 520, 525:
The following limitations apply for the highlighted IFBW settings (1 MHz and above).

Note: These wider IFBWs are used to make wideband pulsed measurements. They do NOT provide faster sweep speeds for non-pulsed measurements.

- **Dwell time** is not allowed.
- **Sweep times** that are slower than the default value are not allowed.
- **Step sweep** mode only - NOT available in Analog sweep.
- **External Trigger Delay** is not allowed.
- **Number of points** for CW sweep is limited to 1001.
- A slight shift (.1dB) in Log Mag traces may be seen when switching in and out of these bandwidths.

**Reduce IF BW at Low Frequencies**

On PNA models with a maximum frequency of 20 GHz and higher, the trace noise becomes worse below 748 MHz. This is especially obvious between 10 MHz and 45 MHz and also when Time Domain is ON. See PNA models / maximum frequencies.

When this box is checked, the PNA uses a smaller IF Bandwidth than the selected value at frequencies below 748 MHz.

This setting:

- can be made for each channel.
- is ON (checked) by default.
- also applies to segment sweep.
- is NOT available on 4-port PNA-L (model N5230A Opt 240 and 245).

Use the following calculations to determine the actual IF Bandwidth value that is used below 748 MHz.

If the result is NOT a selectable IF BW value, the next higher selectable value is used.
<table>
<thead>
<tr>
<th>Model Type</th>
<th>10 MHz to 44.999999 MHz</th>
<th>45 MHz to 748 MHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Less than 20 GHz models:</td>
<td>Not applicable</td>
<td>Not applicable</td>
</tr>
<tr>
<td>ALL 2-port 20 GHz PNA models:</td>
<td>Actual IF BW = (selected IF BW) x (.05)</td>
<td>Actual IF BW = selected IF BW (No reduction)</td>
</tr>
<tr>
<td>ALL 40 GHz and higher models:</td>
<td>Actual IF BW = (selected IF BW) x (.025)</td>
<td>Actual IF BW = (selected IF BW) x (.5)</td>
</tr>
</tbody>
</table>

**PNA-X Models**

<table>
<thead>
<tr>
<th>Start Freq</th>
<th>Stop Freq</th>
<th>Actual IF BW = (selected IF BW) x n</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 MHz</td>
<td>19 MHz</td>
<td>n = .05</td>
</tr>
<tr>
<td>19 MHz+</td>
<td>53 MHz</td>
<td>n = .1</td>
</tr>
<tr>
<td>53 MHz+</td>
<td>75 MHz</td>
<td>n = .5</td>
</tr>
<tr>
<td>75 MHz+</td>
<td>26.5 GHz</td>
<td>n = 1</td>
</tr>
</tbody>
</table>

+ indicates plus 1 Hz

**Example:**

On a 67 GHz PNA, the selected IF BW is 30 KHz.

With *Reduce IF BW at Low Frequencies* checked, the actual IF Bandwidths used are:

- From **10 MHz to 44.999999 MHz**: 30,000Hz * .025 = 750 Hz (PNA uses next higher selectable value: 1000 Hz.)
- From **45 MHz to 748 MHz**: 30,000Hz * .5 = 15 KHz
- From **748 MHz** to stop sweep: **30 KHz**

**OK** Selects the value of IF bandwidth shown in the text box.

Learn about IF Bandwidth (scroll up)

**Trace Smoothing**

Trace smoothing averages a number of adjacent data points to smooth the displayed trace. The number of adjacent data points that get averaged together is also known as the smoothing aperture. You can specify aperture as either the number of data points or the percentage of the x-axis span.

Trace Smoothing reduces the peak-to-peak noise values on broadband measured data. It smoothes trace noise and does not increase measurement time significantly.

Because Trace Smoothing follows Format in the PNA data processing map, the formatted data is smoothed.
Smoothing is automatically turned off if the format is Polar or Smith Chart.

Learn more about Data Format Types.

See the PNA data processing map.

**Tips:**

- Start with a high number of display points and reduce until you are confident that the trace is not giving misleading results.
- Do not use smoothing for high-resonance devices, or devices with wide trace variations. It may introduce misleading information.
- Smoothing is set independently for each trace.

![Effects of Smoothing on a Trace](image)

**How to set Trace Smoothing**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **Avg**
2. then **[Smoothing]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Avg**
3. then **Smoothing**

![Smoothing Settings](image)
<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>15-Feb-2011</td>
<td>Removed point NOT on C models and Noise Fig.</td>
</tr>
<tr>
<td>22-Dec-2009</td>
<td>Added avg formula</td>
</tr>
<tr>
<td>10-Aug-2009</td>
<td>Point Avg Not available on GCA or NF Apps.</td>
</tr>
<tr>
<td>8-Jun-2009</td>
<td>Point averaging not available on C models.</td>
</tr>
<tr>
<td>6-Apr-2009</td>
<td>Replaced N5242A with PNA-X</td>
</tr>
<tr>
<td>12-Mar-2009</td>
<td>Edit IFBW for WB pulse</td>
</tr>
<tr>
<td>9-Oct-2008</td>
<td>Updated for Point Average (8.33)</td>
</tr>
<tr>
<td>3-Sep-2008</td>
<td>Removed legacy content</td>
</tr>
</tbody>
</table>

**Smoothing dialog box help**

**Smoothing ON**  When checked, applies smoothing to the displayed trace.

**Percent of Span**  Specify percent of the swept stimulus span to smooth. For example, for a trace that contains 100 data points, and specify a percent of span = 11%, then the number of data points that are averaged is 11.

**Points**  Specify the number of adjacent data points to average.

[Learn about Trace Smoothing](scroll up)
Crosstalk

Crosstalk is energy leakage between analyzer signal paths. This can be a problem with high-loss transmission measurements. Although the crosstalk specification of the PNA is exceptional, you can reduce the effects of crosstalk by doing the following:

- Set the Sweep to Alternate
- Perform an Isolation Calibration

### Other topics about Optimizing Measurements

**Set the Sweep to Alternate**

This selection is longer available from the user interface. [Learn more.]

**Perform an Isolation Calibration**

For transmission measurements, a response and isolation measurement calibration helps reduce crosstalk because the analyzer measures and then subtracts the leakage signal during the measurement calibration. The calibration improves isolation so that it is limited only by the noise floor.

**Note:** Isolation is never performed on a Smart (Guided) Calibration. [Learn more.]

Generally, the isolation error falls below the noise floor. So when you are performing an isolation calibration you should use a noise reduction technique such as sweep averages or reducing the IF bandwidth.

Last Modified:

3-Mar-2010  Edited alternate sweep text
Effects of Accessories

Accessories in a configuration may affect the results of a device measurement. You can choose between two analyzer features that reduce the effects of accessories.

- **Power Slope to Compensate for Cable Loss**
- **Gating to Selectively Remove Responses**

Other topics about Optimizing Measurements

Power Slope to Compensate for Cable Loss

If you have a long cable or other accessory in a measurement configuration where a power loss occurs over frequency, apply the power slope function. This function increases the analyzer source power by a rate that you define (dB/GHz).

1. In the **Channel** menu, click **Power**.
2. If the slope function is not already switched on, click the **Slope** check box.
3. In the dB/GHz box, enter the rate that you want the source power to increase over the frequency sweep. Click **OK**.

Gating to Selectively Remove Responses

Gating is a feature in the time domain (option 010) that allows the analyzer to mathematically remove responses. You can set the gate for either a reflection or transmission response, but you will see different results.

- **Gating a reflection response** isolates a desired response (such as a filter's return loss), from unwanted responses (such as adapter reflections or connector mismatches).
- **Gating a transmission response** isolates a specific path in a multipath device that has long electrical lengths.

See [Time Domain Gating](#) for more information.
Achieve Fastest Sweep

You can achieve the fastest measurement sweep by adjusting the following:

- **Sweep Settings**
- **Noise Reduction Settings**
- **Measurement Calibration Choice**
- **Unnecessary Functions**

## Other topics about Optimizing Measurements

### Sweep Settings

Consider changing each of the following settings as suggested.

- **Frequency Span** - Measure only the frequencies that are necessary for your device.
- **Segment Sweep** - Use segments to focus test data only where you need it.
- **Switch Off Stepped Sweep** - Use linear swept mode to minimize sweep time when possible.
- **Auto Sweep Time** - Use this default to sweep as quickly as possible for the current settings.
- **Number of Points** - Use the minimum number of points required for the measurement.

For more information on how number of points and other settings affect sweep cycle time, see [Technical Specifications](#).

### Noise Reduction Settings

Using a combination of these settings, you can decrease the sweep time while still achieving an acceptable measurement.

- **IF Bandwidth**. Use the widest IF bandwidth that will produce acceptable trace noise and [dynamic range](#).
- **Average**. Reduce the average factor, or switch Average off.

### Measurement Calibration Choice

Choose the appropriate type of calibration for the required level of accuracy.

When full 2-port error correction is applied, the PNA takes both forward and reverse sweeps to gather all 12 error correction terms. This occurs even with a single S11 measurement displayed. All displayed measurements are updated as the second sweep is performed. Both sweeps are performed using the specified sweep time.
When calibrating greater than 2 ports, the following formula is used to determine the number of sweeps required:

- \( N \times (N-1) \) where \( N \) = the number of ports.

When full 3-port calibration is applied, 6 sweeps are required; forward and reverse for each port pair. With full 4-port correction, 12 sweeps are required, and so forth.

To limit the measurement time, perform ONLY the level of calibration that your measurements require. For example, if making only an S11 measurement, perform a 1-port calibration on that port.

Sweep speed is about the same for uncorrected measurements and measurements done using a response calibration, or one-port calibration. For more information see Select a Calibration.

### Unnecessary Functions

The analyzer must update information for all active functions. To achieve an additional increase in sweep speed, switch off all of the analyzer functions that are not necessary for your measurement application.

- **Delete Unwanted Traces**
- **Switch Off Unwanted Markers**
- **Switch Off Smoothing**
- **Switch Off Limit Testing**
- **Switch Off Math Functions**

Analyzer sweep speed is dependent on various measurement settings. Experiment with the settings to get the fastest sweep and the measurement results that you need.
Switch Between Multiple Measurements

If you need to make multiple measurements to characterize a device, you can use various methods to increase throughput. Experiment with these methods to find what is best for your measurement application needs.

- Set Up Measurements for Increased Throughput
  - Arrange Measurements in Sets
  - Use Segment Sweep
  - Trigger Measurements Selectively
- Automate Changes Between Measurements
- Recall Measurements Quickly

Other topics about Optimizing Measurements

Set Up Measurements for Increased Throughput

To achieve optimum throughput of devices that require multiple measurements, it is helpful to know the operation of the PNA. This knowledge allows you to set up the measurement scenarios that are best for your applications.

Learn more about Traces, Channels, and Windows on the PNA

Arrange Measurements in Sets

If you arrange measurements to keep the complete set of device measurements in one instrument state, you can save them so that you can later recall a number of measurements with one recall function.

See Pre-configured Measurement Setups for more information.

Use Segment Sweep

Segment sweep is helpful if you need to change the following settings to characterize a device under test.

- Frequency Range
- Power Level
- IF Bandwidth
- Number of Points

The segment sweep allows you to define a set of frequency ranges that have independent attributes. This allows you to use one measurement sweep to measure a device that has varying characteristics.

See Segment Sweep for more information.
**Trigger Measurements Selectively**

You can use the measurement trigger to make measurements as follows:

- Continuously update only the measurements that have rapidly changing data.
- Occasionally update measurements that have infrequently changing data.

For example, if you had four channels set up as follows:

- Two channels measuring the data that is used to tune a filter
- Two channels measuring the data for the out-of-band responses of the filter

You would want to constantly monitor only the measurement data that you use for tuning the filter. If you continuously update all of the channels, this could slow the response of the analyzer so that you would not be able to tune the filter as effectively.

**Note:** You must either trigger the infrequent measurement manually or with remote interface commands.

**To trigger measurements selectively:**

This procedure shows you how to set up two different measurements with the following behavior:

- Channel 1 measurement will continuously update the data.
- Channel 2 measurement will occasionally update the data.

1. In the Windows menu, click Meas Setups, Setup D.

**Set Up a Measurement Trigger for Continuous Updates**

2. In the Sweep menu, click Trigger, Trigger....


4. Under Channel Trigger State, select Channel 1, and click Continuous.

**Set Up a Measurement Trigger for Occasional Updates**

5. Under Channel Trigger State, select Channel 2, and click Single, OK.

   - If you want the analyzer to trigger more than a single sweep, click the Enable Groups check box and enter the number of sweeps.

6. In the System menu, click Keys, Trigger.

**Update the Measurement**

7. Click on the lower window to make Channel 2 the active channel.
8. On the active entry toolbar, click the type of trigger you set up.

   - Click **Single** if you set up the analyzer for a single sweep per trigger.
   - Click **Groups** if you set up the multiple sweeps per trigger.

**Note:** A trace must be active for you to initiate a trigger for that measurement.

**Automate Changes Between Measurements**

If there are slight differences between the various measurements that you need to characterize a device, you may find that it is faster to change the measurement settings using programming.

**Recall Measurements Quickly**

The most efficient way to recall measurements is to recall them as a set of measurements (instrument state).

- It only takes a short time longer to recall an instrument state that includes multiple measurements, than it does to recall an instrument state with only one measurement.

- Each recall function has time associated with it. You can eliminate that time by setting up the measurements as a set so you can recall them as a set.

See [Save and Recall Files](#) for more information.
Data Transfer Speed

When testing devices remotely using COM or SCPI, the following techniques can be used to transfer data quickly between the PNA and remote computer, helping you achieve the best measurement throughput.

- **Use single sweep (trigger) mode** to ensure that a measurement is complete before starting a data transfer.

- **Transfer the minimum amount of data** needed. For example, a trace with a few points, using segment sweep rather than a full trace with many linearly spaced points. Also, use markers instead of trace transfers.

- **Choose the REAL data format** to provide the fastest transfer speed when using SCPI programs for automated applications.

- **Use SCPI over LAN** for applications that are automated with SCPI programs.

- **Use COM programs** to provide the fastest transfer speed when using an automated application. See Data Transfer Time specifications.

**Note:** The following data is obsolete, but still serves to illustrate the relative speed between COM and SCPI.

![Data Transfer Speed Comparison](image_url)

**Other topics about Optimizing Measurements**
Using Macros

Macros are executable programs that you write, load into the analyzer, and then run from the analyzer. You can have up to 25 macros set up to run on the analyzer.

**How to Setup Macros**

**How to Run Macros**

**Macro Example**

## How to Setup Macros

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>For PNA-X and 'C' models</td>
<td>Programming Commands</td>
</tr>
<tr>
<td>1. Press MACRO</td>
<td>1. Click Utility</td>
</tr>
<tr>
<td>2. then [Macro Setup]</td>
<td>2. then Macro</td>
</tr>
<tr>
<td></td>
<td>3. then Macro Setup</td>
</tr>
</tbody>
</table>

**In the Macro Setup dialog box:**

1. Create an executable program and save it on the PNA hard drive. See SCPI or COM example programs in VBscript.

2. Use a mouse or the front-panel 'down-arrow' to select a blank line below the last entry. (There may be NO entry.)

3. Click Edit to start the Edit Macro Setup dialog.

4. In the Macro Title box, type a descriptive title for your macro.

5. Click Browse.

6. Change Files of Type.

7. Find and select your executable file. Change Files of Type if necessary.

8. Click OK on the Edit Macro Setup dialog.

9. Click OK on the Macro Setup dialog.

10. Press MACRO to run. It may be necessary to first Preset the PNA to see your macro in the menu.
**Macro Setup dialog box help**

Allows you to create a set of up to 25 macros that can be launched from the PNA application.

An external keyboard is required to enter the Macro Title and the Run string parameters.

**To add a Macro**, use a mouse or the front-panel 'down arrow' (NOT the 'Down' key) to select a blank line. Then click **Edit**.

- **Macro Title**  Shows the titles that appear in the softkeys and menu when you press the Macro key. These titles are associated with the executable files and should be descriptive so you can easily identify them.

- **Macro Executable**  Lists the complete path to the executable file. To follow the example of launching the Agilent PNA Series Home Page, the path to the executable could be "C:/Program Files/Internet Explorer/iexplore.exe.

- **Macro Runstring Parameters**  Lists the parameters that get passed to the program that is referenced in the executable file. Again following the example of launching the PNA Series Home Page, you could assign the runstring parameters "http://www.agilent.com/find/pna".

**Edit**  Invokes the **Macro Edit dialog box**.

**Delete**  Deletes the selected macro.

**Up**  Allows you to reorder the macros, moving the selected macro up one line. This order determines how they appear in the PNA Menu and in the softkeys and when you press the Macro front-panel key.

**Down**  Moves the selection down one line in the list of macros.
**Macro Edit dialog box help**

**Macro Title**  Allows you to modify the title that appears in the softkeys and menu.

**Macro Executable**  Allows you to modify the complete path to the macro executable file.

**Browse**  Allows you to look through drives and directories, to locate the macro executable file and establish the complete path to the file.

**Macro run string parameters**  Allows you to modify the parameters that are passed to the program referenced in the executable file.

See Macro Setup dialog box

---

**How to Run Macros**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>For PNA-X and 'C' models</strong></td>
<td></td>
</tr>
<tr>
<td>1. Press <strong>MACRO</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then select the macro to run</td>
<td>2. then <strong>Macro</strong></td>
</tr>
<tr>
<td></td>
<td>3. then select the macro to run</td>
</tr>
</tbody>
</table>

---

**Macro Example**

The following is an example Visual Basic Scripting (vbs) program that you can copy, install, and run on your PNA.

**Note:** Print these instructions if viewing in the analyzer. This topic will be covered by the Macro Setup dialog box.

1. Copy the following code into a Notepad file.
2. Save the file on the analyzer hard drive in the **C:/Documents** folder. Name the file **FilterTest.vbs**
3. Close Notepad
4. **Setup the macro in the PNA**
5. **Run the macro**

```
'start copying here
'this program creates a s21 measurement, with bandwidth
'markers for testing a 175MHz Bandpass filter
'it is written in vbscript using COM commands

Set PNA = CreateObject("AgilentPNA835x.Application")
PNA.Preset
```
Set chan=PNA.activechannel
Set meas=PNA.activeMeasurement
Set limts = meas.LimitTest
Set trce = PNA.ActiveNAWindow.ActiveTrace

meas.ChangeParameter "S21",1
chan.StartFrequency = 45e6
chan.StopFrequency = 500e6
trce.ReferencePosition = 8
PNA.TriggerSignal = 3

'Do Test
for t=1 to 5
   call measure
   call compare
next
msgbox("Done Testing")

sub measure
   msgbox("Connect Device " & t & " and press OK")
   PNA.ManualTrigger True
   meas.SearchFilterBandwidth
end sub

sub compare
   BW = meas.FilterBW
   if bw>6.5e7 then msgbox("Failed BW: " & BW)
   Loss = meas.FilterLoss
   if loss>5 then msgbox("Failed Loss: " & Loss)
end sub
'End copying here

Last Modified:

17-Feb-2009   Added 25 limit
4-Sep-2008    Removed legacy content
Calibration Overview

The following is discussed in this topic:

- What Is Measurement Calibration?
- Why Is Calibration Necessary?
- Conditions Where Calibration Is Suggested
- What Is ECal?

What Is Measurement Calibration?

Calibration removes one or more of the systematic errors using an equation called an error model. Measurement of high quality standards (for example, a short, open, load, and thru) allows the analyzer to solve for the error terms in the error model. See Measurement Errors.

You can choose from different calibration types, depending on the measurement you are making and the level of accuracy you need for the measurement. See Select a Calibration Type.

The accuracy of the calibrated measurements is dependent on the quality of the standards in the calibration kit and how accurately the standards are modeled (defined) in the calibration kit definition file. The calibration-kit definition file is stored in the analyzer. In order to make accurate measurements, the calibration-kit definition must match the actual calibration kit used. To learn more, see Accurate Calibrations.

Calibration Wizard provides the different calibration methods used in the PNA. See Calibration Wizard.

There are quick checks you can do to ensure your measurement calibration is accurate. To learn more see Validity of a Measurement Calibration.

If you make your own custom-built calibration standards (for example, during in-fixture measurements), then you must characterize the calibration standards and enter the definitions into a user modified calibration-kit file. For more information on modifying calibration kit files, see Calibration Standards.

Note: Instrument Calibration is ensuring the analyzer hardware is performing as specified. This is not the same as measurement calibration.

Why Is Calibration Necessary?

It is impossible to make perfect hardware that would not need any form of error correction. Even making the hardware good enough to eliminate the need for error correction for most devices would be extremely expensive.

The accuracy of network analysis is greatly influenced by factors external to the network analyzer. Components of the measurement setup, such as interconnecting cables and adapters, introduce variations in magnitude and phase that can mask the actual response of the device under test.

The best balance is to make the hardware as good as practically possible, balancing performance and cost. Calibration is then a very useful tool to improve measurement accuracy.
**Conditions Where Calibration Is Suggested**

Generally, you should calibrate for making a measurement under the following circumstances:

- You want the best accuracy possible.
- You are adapting to a different connector type or impedance.
- You are connecting a cable between the test device and an analyzer test port.
- You are measuring across a wide frequency span or an electrically long device.
- You are connecting an attenuator or other such device on the input or output of the test device.

If your test setup meets any of the conditions above, the following system characteristics may be affected:

- Amplitude at device input
- Frequency response accuracy
- Directivity
- Crosstalk (isolation)
- Source match
- Load match

**What Is ECAL**

ECal is a complete solid-state calibration solution. It makes one port (Reflection), full two and three-port calibrations fast and easy. See [Using ECal](#).

- It is less prone to operator error.
- The various standards (located inside the calibration module) never wear out because they are switched with PIN-diode or FET switches.
- The calibration modules are characterized using a TRL-calibrated network analyzer.
- ECal is not as accurate as a good TRL calibration.

For information about ordering ECal modules, see [Analyzer Accessories](#) or contact your [Agilent Support Representative](#).
Calibration Standards

This following section explains the general principles and terms regarding calibration kit files. To learn how to modify calibration kit files, See Modify Calibration Kits.

- **About Calibration Kits**
- **Calibration Standards**
- **Standard Type**
- **Standard Definitions**
- **Class Assignments**

See other Calibration Topics

### About Calibration Kits

A calibration kit is a set of physical devices called standards. Each standard has a precisely known or predictable magnitude and phase response as a function of frequency.

In order to calibrate the analyzer using the standards in a calibration kit, the response of each standard must be mathematically defined and then organized into standard classes that correspond to the error models used by the analyzer.

To be able to use a particular calibration kit, the known characteristics from each standard in the kit must be stored into analyzer memory. This is done for you with the PNA. All Agilent Cal Kits containing standard definitions are stored in the PNA. For a list of Agilent calibration kits, see Analyzer Accessories.

### Calibration Standards

Calibration standards provide the reference for error-corrected measurements in the network analyzer. Each standard has a precisely known definition that includes electrical delay, impedance, and loss. The analyzer stores these definitions and uses them to calculate error correction terms.

During measurement calibration, the analyzer measures standards and mathematically compares the results with "ideal models" of those standards. The differences are separated into error terms that are later removed from device measurements during error correction. See Systematic Errors.

### Standard Type

A standard type is one of five basic types that define the form or structure of the model to be used with that standard. The standard types are shown below:
<table>
<thead>
<tr>
<th>Standard</th>
<th>Terminal Impedance</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHORT</td>
<td>zero ohms</td>
</tr>
<tr>
<td>OPEN</td>
<td>infinite ohms</td>
</tr>
<tr>
<td>LOAD</td>
<td>system impedance, Z₀</td>
</tr>
<tr>
<td>THRU/LINE</td>
<td>no terminal impedance</td>
</tr>
<tr>
<td>ARBITRARY</td>
<td>user-defined</td>
</tr>
</tbody>
</table>

Learn about other Calibration Standards:

- Data-Based Standard
- Sliding Load
- Offset Load

Standard Definitions

Standard definitions describe the electrical characteristics of the standards and the frequencies they will be used. Standard definitions can be viewed from the Advanced Modify Cal Kit menu selection. Standard definitions include:

- **Minimum Frequency**  Specifies the minimum frequency the standard is used for calibration.
- **Maximum Frequency**  Specifies the maximum frequency the standard is used for calibration.
- **Z₀**  Specifies the characteristic impedance of the standard (not the system characteristic impedance or the terminal impedance of the standard).
- **Delay**  Specifies a uniform length of transmission line between the standard being defined and the actual calibration plane.
- **Type**  Specifies type of standard (SHORT, OPEN, THRU/LINE, LOAD, ARBITRARY).
- **Loss**  Specifies energy loss, due to skin effect, along a one-way length of coaxial cable.

Loss model equation:

- The value of loss is entered as ohms/second at 1 GHz.
- To compute the loss of the standard, measure the delay in seconds and the loss in dB at 1 GHz. Then use the following formula:

\[
\text{Loss} = \frac{\text{loss (dB)} \times Z₀(\Omega)}{43429(\text{dB/second}) \times \text{delay (s)}}
\]
**Capacitance model equation:**

*C0, C1, C2, C3.* Specifies the fringing capacitance for the open standard.

- \[ C = (C0) + (C1 \times F) + (C2 \times F^2) + (C3 \times F^3) \]
- \( F \) is the measurement frequency.
- The terms in the equation are defined when specifying the open as follows:
  - \( C0 \) term is the constant term of the third-order polynomial and is expressed in Farads.
  - \( C1 \) term is expressed in F/Hz (Farads/Hz).
  - \( C2 \) term is expressed in F/Hz².
  - \( C3 \) term is expressed in F/Hz³.

**Inductance model equation:**

*L0, L1, L2, L3.* Specifies the residual inductance for the short standard.

- \[ L = (L0) + (L1 \times F) + (L2 \times F^2) + (L3 \times F^3) \]
- \( F \) is the measurement frequency.
- The terms in the equation are defined when specifying the short as follows:
  - \( L0 \) term is the constant term of the third-order polynomial and is expressed in Henries.
  - \( L1 \) term is expressed in H/Hz (Henries/Hz).
  - \( L2 \) term is expressed in H/Hz².
  - \( L3 \) term is expressed in H/Hz³.

**Class Assignments**

Once a standard is characterized, it must be assigned to a standard "class". A standard class is a group of standards that are organized according to the calibration of the PNA error model.

The number of classes needed for a particular calibration type is equal to the number of error terms being corrected.

A class often consists of a single standard, but may be composed of multiple standards. These may be required for accuracy or to cover a wide frequency range.

**Example:** A response calibration requires only one class, and the standards for that class may include an OPEN, or SHORT, or THRU. A 1-port calibration requires three classes. A 2-port calibration requires 10 classes, not including two for isolation.

The number of standards assigned to a given class may vary from one to seven for unguided calibrations. Guided calibrations allow as many standards as needed.

Calibration Classes are assigned in the Advanced Modify Cal Kit menu selection.

**The different classes used in the PNA:**

*S11A, S11B, S11C (S22A, S22B, S22C and so forth)*

These are the three classes for port 1-reflection calibrations (three classes also for S22 and S33). They are used in...
the one-port calibrations and the full two-port calibration. They are required in removing the directivity, source match, and reflection tracking errors. Typically, these classes might consist of an open, a short and a load standard for each port.

**Transmission and Match (forward and reverse)**

These classes are used to perform a full two-port calibration. The transmission class relates primarily to the transmission tracking, while the match class refers to load match. For both of these classes, the typical standard is a thru or delay.

**Isolation**

The isolation classes are used to perform a full two-port and the TRL two-port calibrations. The isolation classes apply to the forward and reverse crosstalk terms in the PNA error model.

**TRL THRU**

These are used to perform a TRL two-port calibration. The TRL thru class should contain a thru standard or a short line. If it contains a non-zero length thru standard, then the calibration type is called LRL or LRM.

**TRL REFLECT**

This class is used to perform a TRL two-port calibration. The TRL reflect class should contain a standard with a high reflection coefficient, typically an open or short. The actual reflection coefficient need not be known, but its phase angle should be specified approximately correctly (± 90 deg). The exact same reflection standard must be used on both ports in the TRL calibration process.

**TRL LINE or MATCH**

These are used to perform a TRL two-port calibration. The TRL line or match class should contain line standards, load standards, or both. If a line standard is used, its phase shift must differ from that of the TRL THRU standard by 20° to 160°. This limits the useable frequency range to about 8 to 1. Two or more line standards of different lengths may be specified to get broader frequency coverage. It is also common to include a load standard for covering low frequencies, where the line's length would be impractically long. When a load is used, the calibration type is called TRM or LRM.

**Note:** For more information, read *Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers* (Application Note 1287-11)
Calibration Wizard

The Calibration Wizard allows you to choose a Calibration method and then perform the calibration.

- How to Start Calibration Wizard
- SmartCal (Guided Calibration)
- Unguided Calibration
- Saving a Calibration

Other Cal Topics

How to start Calibration Wizard

Using front-panel HARDKEY [softkey] buttons

1. Press CAL
2. then [Start Cal]
3. then [Cal Wizard]

Using a mouse with PNA Menus

1. Click Response
2. then Cal Wizard

[Diagram: Calibration Wizard: Begin Calibration window]
Select the calibration method:

**SmartCal (Guided Calibration)**
This method provides a step-by-step "wizard" interface. You describe the connectors on your DUT and the cal kits you will use; it walks you through the most accurate calibration possible.

*Note:* SmartCal allows you to measure calibration standards in any order. However, you must click **Next** and **Back** without measuring standards until you get to the standard you want to measure.

- Supports ALL Cals **EXCEPT** simple open, short, and thru response Cals. See Also [TRL Calibration](#).
- Use a different Cal Kit (**including ECal**) for each port.

**Unguided Calibration**
This method provides a familiar calibration interface, but with limited capability. You choose the type of cal to perform; it allows you the flexibility to measure the standards in any order.

- Supports all Cals **EXCEPT** full 3-port, full 4-port.
- TRL is NOT supported on multiport PNAs.
- Only one Cal Kit can be used.
- Can NOT use Offset Load standard.

**Use Electronic Calibration**

- This method provides fast, software-controlled calibrations.
- Only one ECal module can be used. Use SmartCal when more than one ECal module is needed.

**Save Preferences**

- Clear to continue to see this page on subsequent calibrations.
- Check to save your calibration method choice and no longer see this page. To make this dialog re-appear, click **Response**, then **Cal**, then **Start Cal**, then **Cal Preferences**.
- Learn more about [Calibration Preferences](#).
**The Calibration Window / Channel**

During a Guided Calibration, a 'Cal Window' is created for you to view the connection of calibration standards before standards are measured. This Cal Window uses a new Cal channel that is created and duplicates the settings in the channel being calibrated. *Correction is ALWAYS OFF* for the displayed calibration channel. At the completion of the calibration, the calibration channel and window are deleted.

With PNA Rev. 7.50.27, the measurement of calibration standards can be performed while viewing any PNA window configuration you choose. The Cal Window is appended to your Custom Cal Window setting, and all windows are visible and sweeping below the Cal Wizard before the Measure (cal standard) button is pressed. The windows to be viewed and channels to be swept during the cal process are specified using [Remote commands](#). See an example.

The new Cal Window settings do not work in a FCA channel.

**SmartCal (Guided Calibration)**

A Guided Calibration automatically determines the calibration type and suggests a calibration kit that matches your DUT connectors. Guided Calibration can perform the following Cal Types:

- ALL Cals EXCEPT Open, Short, and Thru Response Cals.
- ECal on one or more ports, beginning with [PNA firmware revision 5.24](#).
- TRL - [Learn how to do TRL cals](#)

**Programming Commands**

*Note:* SmartCal DOES allow you to measure calibration standards in any order. However, you must click Next and Back without measuring standards until you get to the standard you want to measure.

The PNA displays the following dialog boxes when performing a Guided calibration on standard channels.

To learn about Calibrations for Application channels, refer to the help topic for the [Application](#).
Select Ports for Guided Calibration dialog box help

Allows you to select ports to calibrate.

**Cal Type Selection**  Select the number of ports to calibrate.

**N Port Cal Configuration**  If not calibrating all PNA ports, specify which ports to calibrate.

**Calibrate source and receiver power**  Check to perform a Guided Power Calibration. Learn more.

See Also: [Use Multiple Power Sensors to perform a Guided Power Cal](#)

**Show Advanced Settings** (Orientation & Thru Cal Section)  Available only for ECal.

**Back**  Return to Cal Wizard Begin dialog. If you did not see the 'Cal Wizard Begin' dialog but want to, click Back, then clear the Save Preferences checkbox.

For greater than 4-port cals, see [External Test Set calibration - Select Cal Type](#).

Select DUT Connectors and Cal Kits dialog box help

Allows you to select the connector type and Cal Kit for each DUT port to be calibrated.

**Connectors**  To change selection, click the connector field for each DUT port.

If your DUT connectors are not listed, you can create your own connector type and calibration kit file. The PNA includes the following example cal kits that can be used as a template. See Calibration kits for more information.

- If using a gendered (male and female) connector type, select **Type A** as the connector type.
- If using a connectorless device such as on-wafer probes, select **Type B** as the connector type.

**Cal Kits**  Select the Cal Kit to be used to calibrate each test port. The list for each DUT Port displays kits having the same connector type as the DUT.

**Identical ECal models connected?**  ECal modules can be distinguished by serial number. This can have implications on your remote [SCPI](#) programs.
85056K

The 85056K definitions in the PNA are for 2.92mm standards (2.4mm plus 2.92 adapters). To calibrate 2.4 mm connectors using the 85056K cal kit, select 85056A as the cal kit when you need the sliding load. Otherwise, select 85056D as the cal kit. Both the 85056A and the 85056D kits contain exactly the same standards as the 85056K cal kit WITHOUT the adapters.

TRL

- To perform a TRL Cal, assign a TRL Cal Kit to the lowest port number of each port pair.

- When selecting a TRL Cal Kit on a 4-port PNA, and a Global Delta Match Cal is not available, the Cal type will be set to SOLT and a “Could not find a Global Delta Match Cal.” message is displayed on the dialog box. If the selected Cal Kit will not support SOLT, the Next button will not be available. Then you must select a different Cal Kit to proceed or Cancel and perform a Global Delta Match Cal.

Modify Cal Check, then click Next, to Modify Cal (Standards AND Thru Method).

For greater than 4-port cals, see External Test Set calibration - Select DUT Connectors.

Error dialog box help

The current cal kit does not cover the current frequency range of the measurement. Do one of the following to correct the problem:

Cal Kit Class Category Choose from SOLT and TRL. Not available with ECal modules. Click Edit to modify the appropriate class assignments.

Frequency Change the frequency range of the active channel.

Edit Modify the class assignments so that a different standard is selected.

Back Select a different Cal Kit that covers the required frequency range.

Cancel Exit the Cal Wizard
Modify Cal dialog box help

Thru #n
Lists the proposed Thru connections to be made during the calibration process. You can change these Thru connections to better suit your test setup.

- The proposed Thru connections are listed automatically.
- Additional Thru connections can be selected for higher accuracy. Learn more.

Add Thru
Click to add a Thru connection. Learn more

Remove Thru
Select a Thru by clicking the "Thru #N" field or the "1st Port / 2nd Port" field. Then click "Remove Thru". This selection is NOT available if the selected Thru is required for the calibration.

1st Port / 2nd Port
Click to select the two ports to be included in the Thru connection. The order of the port numbers is not critical.

Thru Cal Method
Lists the available Thru Cal methods for the specified port pairs. Learn about the Thru Cal Method choices.

Cal Type/ Stds
Click to invoke the View / Modify Properties of Cal dialog box

Do orientation - Appears ONLY if an ECal module is selected for use.
When this box is checked (default) the PNA automatically senses the model and direction in which an ECal module port is connected to the PNA ports. If power to the ECal module is too low, it will appear as if there is no ECal module connected. If you use low power and are having this problem, clear this check box to provide the orientation manually.

Orientation occurs first at the middle of the frequency range that you are calibrating. If a signal is not detected, it tries again at the lowest frequency in the range. If you have an E8361A or E836xB PNA and do an ECal completely within 10 - 20 MHz OR 60 - 67 GHz, you may need to do orientation manually. There may not be sufficient power to orient the ECal module at those frequencies.
Choose delta match - Available when a Delta Match Cal is required.

- Check, then click Next to invoke the Select Cal Set for Delta Match dialog box.
- Clear - The Cal Wizard uses the Global Delta Match Cal if available.

View/Detect ECaI Characterizations - Appears ONLY if an ECaI module is selected for use.
Click to invoke the View ECaI Modules and Characterizations dialog box. Displays a list of ECaI modules that are connected to the PNA.

View/Modify Properties of Cal for Ports... dialog box help

Select calibration type

Another chance to change the Thru method. 
Learn about the Thru Cal Method choices.

Advanced

Select the cal method for each connector of the Thru pair.

- **TRL** is only available if a TRL cal kit was selected for the lowest port number of the port pair.
- **QSOLT** Only available when "Defined Thru" or "Flush Thru" is selected. "QSOLT 2 <= 1" refers to the receive port 2 and source port 1(where reflection standards are connected).
- **Enhanced Response** Only available when “Defined Thru” or "Flush Thru" is selected. “EnhResp 2 <= 1” refers to the receive port 2 and source port 1.

View Modify  Click to invoke the Preview and Modify Calibration Selections dialog box.

**Note:** Changes made to the Cal Kit through this dialog are temporary that last only for this calibration. To make permanent changes to the Cal Kit, perform Advanced Modify Cal Kits.
**Select Cal Set for Delta Match** dialog box help

This dialog box appears when a Delta Match Cal is required and **Choose delta match** was selected. [Learn more](#).

Displays the Cal Sets that meet the requirements of the Delta Match Cal.

Select either a User Cal Set or **Global Delta Match Cal**.

If there is no suitable choice for a Delta Match Cal:

1. Click **Cancel**, then **Cancel** again to quit the Cal Wizard.
2. Perform either a **Global Delta Match Cal** or a SOLT cal and save the result in a User Cal Set.
3. Start the Cal Wizard to re-initiate this calibration.
4. Select the Global Delta Match Cal or User Cal Set.
**Calibration Steps** dialog box help

**Note:** Beginning in PNA Rev. 6.0, calibration can be performed with External triggers. Learn more.

As each new cal step prompt appears, the traces are setup for the next standard measurement. Also, sweeps are triggered continuously until the Measure button is pressed. This way you can view the integrity of the standard connection.

Prompts for standards to be measured.

- **Measure** Click to measure the standard.
- **Done** Click after a standard is re-measured and all measurements for the calibration are complete.
- **Next** Click to continue to the next calibration step. Does NOT measure the standard.

If a standard is NOT measured, a warning appears and **Done** will not be available after the last Cal step.

**Note:** Smart (Guided) Cal allows you to measure calibration standards in any order. However, you must click **Next** and **Back** without measuring standards until you get to the standard you want to measure.

---

**Sliding Load Measurement** dialog box help

Allows you to measure the sliding load standard. Learn more about the Sliding Load standard.

To ensure an accurate calibration, carefully follow the instructions that were provided with your sliding load.

**To Measure a Sliding Load:**

1. Connect the sliding load to the measurement port.

2. Position the sliding element, then click **Measure**. Do not move the sliding element until measurement is complete.

3. Measure the sliding load for at least five and up to seven positions for best accuracy.

   **Note:** The positions of the sliding element should cover the full length of the slide, but be unequally spaced to reduce the possibility of overlapping data points. Most sliding loads have marks for each slide position.

4. Click **Done** after the final measurement.
4. Remove sliding load from the measurement port.

5. Measure the remaining standards.

**Specify delay** dialog box help

This dialog appears ONLY when Adapter Removal or Unknown Thru calibrations are performed.

The following values were estimated from the measurement. Most of the time, they are adequate. However, for CW sweep or frequency sweep with large step sizes, the accuracy of the values may be improved.

**Adapter delay** To improve this value, measure and record the delay of the adapter with a dense step size. Enter that value here. The required precision value is the accuracy that is required to characterize the delay value.

**Nominal phase offset** (Waveguide ONLY). To improve this value, measure and record the phase offset of the Waveguide adapter with dense step size. Enter that value here.

When one connector is coax and the other connector is waveguide, the phase offset has an ambiguity of 180 degrees. For consistency, the estimate provided here is always between 0 and 180 degrees. You can change this estimate to any value between -180 degrees and +180 degrees.

**For FCA calibrations**, this dialog box appears twice: once for the input frequencies and once for the output frequencies. The values can be slightly different.

The **Calibration Complete** dialog box appears after all standards are measured.

**Unguided Calibration**

The PNA displays the following dialog boxes when performing an Unguided calibration:
Select Calibration Type for Mechanical Standards dialog box help

Unguided calibration does **NOT** support cals greater than 2 ports or **ECal** calibrations.

**TRL Cal** should be performed using **Guided Calibration**.

### Calibration Type Selection

- 2-Port SOLT
- 1-Port SOL
- TRL - NOT available on PNA models with more than 2 ports.
- **Response** - Reflection and Thru (if the active measurement is transmission)

### Cal Configuration

If not calibrating all PNA ports, specify which ports to calibrate.

**Back**  Return to **Cal Wizard Begin** dialog. If checked, you can clear the **Save Preferences** checkbox to see the Begin page when the Cal Wizard begins.

**View/Select Cal Kit**  Click to invoke the **Select Cal Kit dialog box**.

**Next**  Click to continue to **Measure Mechanical Standards** dialog box.

**Note:** If the DUT connector type has an impedance other than 50 ohms (waveguide = 1 ohm), change the system impedance before performing a calibration. See **Setting System Impedance**.
Select Cal Type dialog box help

This dialog box only appears if the selected Cal Type is TRL in the previous dialog box.

**TRL Reference Plane**  Select which standard to use to establish the position of the measurement reference plane.

- **THRU Standard**  Select if the THRU standard is zero-length or very short.
- **REFLECT Standard**  Select if the THRU standard is not appropriate AND the delay of the REFLECT standard is well defined.

**TRL Impedance**

- **LINE Standard**  Specifies that the characteristic impedance of the LINE standard should be used as the system impedance. This ignores any difference between Offset Z0, Offset Loss, and System Z0.
- **SYSTEM Impedance**  Transforms the LINE standard impedance and loss to that of the system impedance for use with the calibration error terms. The TRL calibration will first compute the error terms assuming the LINE standard impedance is the system's characteristic impedance (same as previous LINE selection), then modify the error terms to include the impedance transformation. This should only be used with coax since the skin effect model used is a coaxial model.

Learn how to change System Z0.

To learn to substitute other calibration kits, see Advanced Modify Cal Kits.

Select Cal Kit dialog box help

Displays the calibration kit files available for Unguided calibration. Select the desired calibration kit file and click OK.

**Choose class type**  Unguided TRL calibration is NOT available on the 4-port PNA.

**Edit Class Assignments**  Allows modification of the selected Cal Kit class assignments.

To learn to substitute other calibration kits, see Advanced Modify Cal Kits.

**Note:**  If the Cal Kit has an impedance other than 50 ohms (waveguide=1 ohm), change the system impedance before performing a calibration. See Setting System Impedance.
Measure Mechanical Standards dialog box help

**Note:** Beginning in PNA Rev. 6.0, calibration can be performed with External triggers. [Learn more.](#)

Displays the calibration kit file and standards required for the calibration.

- Standards may be connected and measured in any order.
- Connect the standard to the measurement port and click its associated green button. A check mark indicates the standard has been measured.
- If a standard type contains multiple standards, the [Multiple Standards dialog box](#) opens to display the multiple standards included in the calibration kit file.
- If a sliding load is included in the calibration kit file, the [Sliding Load dialog box](#) opens to perform the measurement with the standard.

**Reflection Response** Select EITHER Open or Short standard, then click **Next**.

**Isolation** Requires one load for each test port of the PNA. [Learn more about Isolation.](#) Use when your measurement requires maximum dynamic range (> 90 dB). See also [Isolation Portion of 2-Port Calibration](#).

**Normalize** Available when performing a response cal for any measurement. After Normalize is pressed and the Cal is complete, the data trace is flat when the same physical connections are present on the port. This is similar to [Data/Memory](#), except that the response cal is saved with Cal data and can be applied to other like measurements. Data/Memory is still available after using Normalize. You would usually connect a THRU standard when calibrating a transmission measurement, and a SHORT standard when calibrating a reflection measurement.

**Show Prompts** Check to provide a reminder for the required connection when you click on the standard.
Multiple Standards dialog box help

Select the standards to be measured.

**Note:** You may see both male and female standards. The Unguided cal has no knowledge of the gender of your connector types. **Choose the gender of your DUT connector**, NOT the test port. Then click OK.

To modify this calibration class to show only one standard, on the Calibration menu, click **Advanced Modify Cal Kits**. Select the Cal kit and click **Edit Kit**. In **Class Assignment**, click **Edit**. Learn more about **Modify Calibration Class Assignments**.

- Connect the standard to the measurement port and click its associated button. A check mark in the **Acquired** box indicates the standard has been measured.

- To cover the entire frequency range, you may need to measure more than one standard. The order in which the standards are measured is important. The last standard that is measured will override the others in respect to the frequency range of the standard definition. **Example:** In the case of measuring both a broadband load and a sliding load, you would measure the sliding load last. This is because the frequency range of the sliding load is a subset of the broadband load.

Learn more about **Modify Calibration Class Assignments**.

Saving a Calibration

SmartCal, ECal, and Unguided Calibrations end with the following dialog box:

**Calibration Completed** dialog box help

**Finish**  Save to the channel's calibration register.

**Save As User Cal Set**  Invokes the **Save as User Cal Set dialog box** AND save to the channel's calibration register.

**Cancel**  Calibration is NOT applied or saved.

Learn about **Calibration Registers**.

Learn about **User Cal Sets**.
### Save as User Cal Set dialog box help

**Existing Cal Sets** - Lists the Cal Set names saved on the PNA.

**Select Cal Set from list or type new name below**  Specify a name for the new Cal Set. Either accept the suggested new name, type a new name, or select a name from the list to overwrite an existing name.

**Edit Name**  If there is no keyboard, click to start the PNA typing tool that can be used from the PNA front panel.

**Save**  Saves the Cal Set to the new Cal Set name.

Learn about [User Cal Sets](#)

---

Last modified:

<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>14-Jul-2011</td>
<td>Removed WG impedance restriction for Guided</td>
</tr>
<tr>
<td>16-Sep-2010</td>
<td>Added note for TRL and Noise Cal</td>
</tr>
<tr>
<td>3-Sep-2008</td>
<td>Removed legacy content</td>
</tr>
<tr>
<td>14-Apr-2008</td>
<td>Added note about Offset Load</td>
</tr>
<tr>
<td>4-Mar-2008</td>
<td>Added Cal Window feature</td>
</tr>
<tr>
<td>21-Sep-2007</td>
<td>Added note about no TRL on 4-port PNAs</td>
</tr>
<tr>
<td>January 20, 2007</td>
<td>Added note about any order for SmartCal.</td>
</tr>
<tr>
<td>18-Sept-2006</td>
<td>MQ Major modifications for multiport</td>
</tr>
</tbody>
</table>
Select a Calibration Type

The following calibration types are available in the PNA.

<table>
<thead>
<tr>
<th>Cal Type</th>
<th>Interface</th>
<th>Accuracy</th>
<th>Thru Methods allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRL Family</td>
<td>SmartCal</td>
<td>Very High</td>
<td>All except Unknown Thru</td>
</tr>
<tr>
<td>SOLT</td>
<td>Both</td>
<td>High</td>
<td>All</td>
</tr>
<tr>
<td>Enhanced Response</td>
<td>SmartCal</td>
<td>High</td>
<td>Defined Thru or Flush Thru</td>
</tr>
<tr>
<td>QSOLT (Quick SOLT)</td>
<td>SmartCal</td>
<td>Medium</td>
<td>Defined Thru or Flush Thru</td>
</tr>
<tr>
<td>1-Port Reflection</td>
<td>Both</td>
<td>High</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Open/Short Response</td>
<td>Unguided</td>
<td>Low</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Thru Response</td>
<td>Unguided</td>
<td>Low</td>
<td>Known Thru or Flush Thru</td>
</tr>
</tbody>
</table>

Learn how to select a default Cal Type.

Other Cal Types (Separate Topic)

- Source and Receiver Power Cals

See other Calibration Topics

TRL Family

Application: Used to accurately calibrate any pair of ports when calibration standards are not readily available.

Note: A Delta Match Cal is required to cal test ports that do not have a dedicated reference receiver.

- Learn more about TRL family cal
- For more information on modifying standards, see Calibration Standards.

Calibration Method: SmartCal

General Accuracy: Very High

Standards Required: THRU, REFLECT, LINE or similar combination

Systematic Errors Corrected:
- Directivity
- Source match
- Isolation (see exceptions)
- Load match
- Frequency response transmission tracking
• Frequency response reflection tracking

SOLT

Application: Used to accurately calibrate any number of ports.

General Accuracy: High

Calibration Method: SmartCal, Unguided Calibration, ECal

Standards Required: (SHORT, OPEN, LOAD, THRU) or ECal module

Systematic Errors Corrected (on all ports):
• Directivity
• Source match
• Isolation (see exceptions)
• Load match
• Frequency response transmission tracking
• Frequency response reflection tracking

Enhanced Response

Application: Used to calibrate two ports when only measurements in one direction (forward OR reverse) are required. Measurements are faster because a second sweep is NOT required.

• Reflection Standards (OPEN, SHORT, LOAD) are connected to the source port to be calibrated.
• Defined THRU or Flush THRU standard is connected between port pairs.
• Much quicker than SOLT when using a mechanical cal kit. ECal can also be used.

To select Enhanced Response:

For a standard S-parameter Cal, select SmartCal in the Cal Wizard.

Then, for all cals:

1. At the ‘Select DUT Connectors page’, check Modify Cal, then click Next.

2. Under ‘Cal Type’, select Enhanced Response.

Enhanced Response cal also be selected as the default Cal Type using Cal Preferences.

General Accuracy: High

Calibration Method: SmartCal, ECal

Standards Required: (SHORT, OPEN, LOAD, Defined THRU or Flush THRU)

Systematic Errors Corrected:
• Directivity (source port)
• Source match (source port)
• Isolation (see exceptions)
• Load match (receiver port) - used only to produce transmission tracking term.
• Frequency response transmission tracking (receiver port).
• Frequency response reflection tracking (source port).

**QSOLT (Quick SOLT)**

Application: Used to quickly calibrate any number of ports. Developed specifically for use with external multiport test sets.

**Note:** A Delta Match Cal is required to cal test ports that do not have a dedicated reference receiver.

- Reflection Standards (OPEN, SHORT, LOAD) are connected to only ONE of the ports to be calibrated. The lower port number of the ports to be calibrated is selected by default. This can be changed through the Modify Cal / Cal Type setting.
- Defined THRU or Flush THRU standards are connected from the reflection standard port to the remaining ports to be calibrated.
- Much quicker than SOLT when using a mechanical cal kit.
- Based on TRL math.

**General Accuracy:** Not as high as SOLT

Calibration Method: SmartCal, ECaI

Standards Required: (SHORT, OPEN, LOAD, Defined THRU or Flush THRU)

Systematic Errors Corrected:
- Directivity
- Source match
- Isolation (see exceptions)
- Load match
- Frequency response transmission tracking
- Frequency response reflection tracking

**1-Port (Reflection)**

Application: Used to accurately calibrate any single test port for reflection measurements only.

Calibration Method: SmartCal, Unguided Calibration, ECaI

**General Accuracy:** High

Standards Required: (SHORT, OPEN, LOAD) or ECaI module

Systematic Errors Corrected:
- Directivity
- Source match
- Frequency response reflection tracking
Open / Short Response

Application: Used to quickly calibrate any single test port for reflection measurements only.

Calibration Method: Unguided Calibration

General Accuracy: Low

Standards Required: OPEN or SHORT

Systematic Errors Corrected:
Frequency response reflection tracking

Thru Response (Isolation Optional)

Application: Used to quickly calibrate any pair of test ports for transmission measurements only. Isolation is not usually recommended. Learn more about Isolation

Calibration Method: Unguided Calibration

General Accuracy: Low

Standards Required: THRU

Isolation: One LOAD for each PNA test port.

Systematic Errors Corrected:
• Frequency response reflection tracking
• Isolation

Last modified:

5-Sep-2008    Added note for ER Load Match
16-Apr-2008    Removed AR for TRL limitation
23-Feb-2007    Added Enhanced Response
12-Sept-2006   Added QSOLT
Using Calibration Sets

- What are PNA Cal Sets
- Cal Registers and User Cal Sets
- How to Manage and Apply Cal Sets
- Examples of Cal Set Usage
- Archiving Cal Sets using .cal files

See Also
Save and Recall: Instrument States and Cal Set Data

See other Calibration Topics

What are PNA Cal Sets
At the completion of a calibration, all calibration data is stored to a Cal Set. The Cal Set can be applied later to any channel that has the same stimulus settings as the Cal Set, thereby saving the time it takes to perform another calibration. The following data is saved to a Cal Set:

- Name
- Cal Set Description
- Cal Set Attributes - stimulus settings, cal type, port association
- Standards data
- Error term data
- GUID (Globally Unique IDentifier)

Cal Registers and User Cal Sets
There are two types of Cal Sets:

- Cal Registers (channel specific)
- User Cal Sets

Calibration data is automatically saved to a Cal Register at the end of every calibration. You can also choose to save the cal data to a User Cal Set.
New with PNA Release 5.0, Calibration Registers are designed to simplify calibrations for most users. When a calibration is complete, the data is automatically saved to the channel's Cal Register, overwriting (or appended to) the previous cal data stored in that register. This concept is similar to previous Agilent Vector Network Analyzers.

- Every channel has ONE dedicated Cal Register. They are named CHn_CALREG, where n is the channel number. The name cannot be changed.

- Cal Registers are more volatile because they are overwritten (or appended) each time a calibration is performed on that channel. The Cal data is always saved, but only temporarily.

- Cal Registers can be applied to other measurements, but ONLY on the same channel as the Cal Register.

**User Cal Sets**

At the end of a calibration, you can choose to also save cal data to an existing or new User Cal Set.

- User Cal Sets can be applied to any number of channels simultaneously.

- User Cal Sets are named by you for easy identification.

- You can have an unlimited number of User Cal Sets.

- At any time, you can copy Cal Register data to create a User Cal Set. See [Cal Set Properties](#).

**Note:** You can set a Preference to revert the pre-5.0 behavior - Cal data was automatically saved to a User Cal Set instead of a Cal Register.

**Appending Data in a Cal Set**

At the end of a calibration, data is saved to the channel's Cal Register and, if you choose, to a User Cal Set. When you choose to save to an existing User Cal Set, the PNA attempts to append the new error terms to the existing User Cal Set. The existing Cal Set data is completely overwritten UNLESS the new data can coexist with the existing data according to the following two rules:

- The stimulus settings of the new data must exactly match the existing data.

- The new cal must involve different ports from the existing cal.

For example:

**Case 1** - An existing Cal Set contains a full 2-port cal between ports 1 and 2. Using the same stimulus settings, you perform a 1-port cal on port 3. At the end of the cal, you click Save As User Cal Set and select the existing full 2-port User Cal Set.

**Result:** The 1-port cal is appended to the 2-port User Cal Set. There is NO overlap between them.

**Case 2** - Same situation as Case 1, except the 1-port cal is performed on port 1.

**Result:** The Cal Set will contain a 1 port cal on port1 and a 1 port cal on port 2. The overlapping tracking terms are removed rendering the original full 2 port cal invalid.

**How to Manage and Apply Cal Sets and Cal Types**

309
The PNA attempts to apply a Cal Set, and turn error correction ON, for ALL of the measurements on the active channel. This may not always be possible. For example, suppose a channel contains both S11 (reflection) and S21 (transmission) measurements. If a Cal Set that contains only an S11 Cal Type is applied to that channel, the Cal Set does not contain the error terms to correct the S21 measurement. Error correction is turned ON for the S11 measurement and NOT turned on for the S21 measurement.

There are two ways to apply an existing Cal Set (Cal Register or User Cal Set) to a measurement:

1. Recalling an Instrument State with Cal data (.cst file) - A .cst file contains an Instrument State with all measurement attributes AND a 'pointer' to the Cal Set that was used to calibrate the measurement. Before saving a .cst file, be sure that a User Cal Set (NOT a Cal Register) is being used for the measurement. Because Cal Registers are automatically overwritten when a new calibration is performed, it is likely that the Cal Register data will change before the .cst file is recalled.

2. Create a new measurement and select a Cal Set to apply to the active channel.

**Note:** NEVER copy or modify Cal Sets from Windows Explorer or other applications. Cal Sets should only be accessed through the PNA Application.

**How to select and apply a Cal Set to the active channel**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[Manage Cals]**
3. then **[Cal Set]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Cal**
3. then **Manage Cals**
4. then **Cal Sets**

**Calibration Selection** dialog box help

This dialog box allows you to manage and apply Cal Sets. Although the number of Cal Sets you can have is limited only by the amount of PNA memory, it is considered
unusual to have more than about 10 existing Cal Sets, or one current Cal Set for every unique channel setup. Old Cal Sets (with 'stale' data) should be deleted or overwritten.

The active channel’s Cal Register always appears, even if empty. Cal Registers that belong to other channels appear in the list of Cal Sets only if the channel exists, and only if they contain data.

- Learn about Cal Registers.
- Learn how to View the Error Terms of a Cal Set.

To apply a Cal Set to the active channel, click a row to select that Cal Set, then click Apply Cal.

Note: A Cal Set must have been generated from the same measurement class as the active channel in order for it to Applied.

Columns click a heading to sort by that column
- **Cal Set Name** Name to identify the Cal Set.
- **Description** User-settable text to further identify the Cal Set.
- **Channels** Channel numbers that are currently using this Cal Set. A blank entry means it is not currently in use.
- **CalType / Ports** Type of Cal contained in the Cal Set. Learn about applying appropriate Cal Types.
  - **Cal Type Abbreviations:**
    - 1P, 2P, 3P, 4P... - Full n-Port calibrations
    - R - Response (instead of ports, shows the measurement type that it corrects.)
    - ER/x-y **Enhanced Response**, where x is the receive port; y is the source port.
    - VMC **Vector Mixer Cal**
    - SMC **Scalar Mixer Cal**
- **Modified** Date and time the Cal Set was last modified.

Buttons
- **Copy** Invokes the Save as User Cal Set dialog box. Type a name for the copy of the selected Cal Set data.
- **Show / Edit Properties** Invokes the Cal Set Properties dialog box. This allows you to view all of the Cal Set properties and create a duplicate User Cal Set from an existing User Cal Set or Cal Register.
- **Delete** Permanently deletes the Cal Set after you choose OK to a warning prompt.
- **Delete All** Permanently deletes ALL listed Cal Sets and Cal Registers after you choose OK to a warning prompt.
- **Apply Cal** Applies the selected Cal Set to the active channel. If the stimulus settings of the Cal Set and channel are different, a choice must be made.
- **Unselect** Available ONLY if the selected Cal Set is being used by the active channel. Click to 'Un-apply' the Cal Set, then click Close to exit with the Cal Set un-applied.
- **OK** Always APPLIES THE SELECTED CAL SET to the active channel, then closes the dialog box.
- **Close** Exit the dialog box. Performs no further action.
Cal Set Properties dialog box help

Allows you to view all of the Cal Set properties and create a duplicate User Cal Set from an existing User Cal Set or Cal Register.

- **Name**  Edit name of the User Cal Set. You can NOT change the name of a Cal Register.
- **Description**  Descriptive text to further identify the Cal Set.
- **Cal Set Properties**  Lists descriptive information and stimulus conditions of the Cal Set.

Learn how to View the Error Terms of a Cal Set.

Stimulus Setting Different between Cal Set and Measurement

Select Cal Set -- Choose Stimulus Settings

The Cal Set that you have selected has different stimulus settings than the active channel.

Please select one of the following actions:

- **A**  Do not change the active channel's stimulus settings. If necessary, interpolation will be turned on automatically.
- **B**  Change the active channel's stimulus settings to match those of the selected Cal Set.
The Cal Set contains the channel stimulus settings that were in place when the Cal Set was saved. This dialog appears when the Cal Set channel settings are different than those of the channel to which the Cal Set is being applied. Choose between the following options. (See above image).

- A. Keep the Active Channel Stimulus settings. Interpolate if possible.
  - If the Cal Set frequency range is greater than the active channel, then Interpolation will be turned ON. Learn more about Interpolation Accuracy
  - If the Cal Set frequency range is less than the active channel, then this option is not available.
- B. Keep the Cal Set Stimulus settings. The Active Channel stimulus setting are changed.

OK  Make the change.
Cancel  Cal Set will NOT be applied.

**Examples of Cal Set Usage**
The following examples show how Cal Sets increase flexibility and speed in making analyzer measurements.

- Using one User Cal Set with many Channels
  - Using one Measurement with many Cal Sets

**Using one User Cal Set with many Channels**
It is possible to do one calibration, then apply it to several channels.

An example:
During a manufacturing process, you may have many calibrated channels. You may wish to continuously cycle through the measurements and examine them individually. Occasionally, you may wish to refresh the calibration without having to recreate all the measurement state files.

Here is how: Examine the stimulus settings for each channel. Then make the User Cal Set stimulus range a super-set of the whole group. Each channel can then use the same User Cal Set. Some calibrations will be interpolated. **Note:** Make sure that interpolation is turned on.

Notice in the following image, Cal Set 78 is used on more than one channel, in this case Channel 5 and 16.
Using one Measurement with many Cal Sets

The drawback with having one very large User Cal Set associated with many instrument states could be a loss of accuracy due to interpolation. In such cases, consider using one User Cal Set for each stimulus setting. The stimulus conditions can then be changed for a channel by applying different User Cal Sets. Other settings (window setups, measurement definitions, scaling, limits, markers) will not change. This may result in faster state changes than if you saved and recalled *.cst files for each set of stimulus conditions.

Example #1: An amplifier needs to be measured at several input power levels. Calibrate at several power levels and save each calibration in a separate User Cal Set. Then, apply the User Cal Sets to the single measurement consecutively.

Example #2: Making an S21 Measurement, you need to measure both wide span and narrow span characteristics of the device. One Cal Set covers the wide span setup; another the narrow span setup.

Archiving Cal Sets using .cal or .csa files

Because User Cal Sets can easily be deleted, provide extra backup by also saving your calibration as a .cal or .csa file (see saving a .cal file).

Example:

One person performs a calibration, names and saves it as a User Cal Set. This Cal Set is available for any other person to use. A second user could accidentally delete or modify the User Cal Set requiring the originator to repeat the calibration.

Security can be provided for calibration data by saving the Cal Set to a .cal file or .csa file. At a later time, the file could be recalled and the original calibration restored.

Last modified:
17-Sep-2009  Edit wording in Appending...
3-Sep-2008   Removed legacy content
22-Apr-2008   Added link to UI Cal Set preference
              9/12/06  Added link to programming commands
Error Correction and Interpolation

Error Correction and Interpolation settings work together to provide you with the highest level of calibration accuracy possible.

- How to set Error Correction
- Error Correction
- Viewing Correction Levels
- How to set Interpolation
- Interpolation and Accuracy

See other Calibration Topics

How to set Error Correction

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press Cal</td>
<td>1. Click Cal</td>
</tr>
<tr>
<td>2. then Correction ON/off</td>
<td>2. then Correction ON/off</td>
</tr>
</tbody>
</table>

**Error Correction**

The Error Correction ON setting means that the calibration error terms are applied to the measurement. Error Correction is automatically turned ON when a calibration is performed or if a Cal Set is applied to a measurement. The PNA attempts to turn error correction ON for ALL of the measurements on the active channel. This may not always be possible when applying Cal Sets. For more information, see Applying Cal Sets.

When full 2-port error correction is ON, both forward and reverse sweeps are required to gather all 12 error terms, even if only one reflection measurement is displayed. This may result in a higher measurement speed than expected. Learn more.

You can always turn Error Correction OFF for the active measurement by clicking Correction OFF. The PNA will turn Error Correction OFF automatically when making stimulus changes under some conditions. To turn correction back ON, click Correction ON. Then:

- If Interpolation can NOT be performed, a dialog box will ask if you would like to change the stimulus settings to those of the applied calibration. Click OK or Cancel.
• If Interpolation can be performed, the stimulus setting will change and correction turned ON.

**Viewing Correction Level**

The correction level provides information about the accuracy of the active measurement. Correction level notation is displayed on the status bar for different calibration types like response, full 2-port, TRL, or power calibration.

**To View Correction Levels:**

In the **View** menu, click **Status Bar**. The status bar appears and displays the following items:

- a. Active Channel
- b. Measurement parameter
- c. Correction Level (see description below)
- d. Calibration type

<table>
<thead>
<tr>
<th>Correction Level</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>C Full</td>
<td>Highest</td>
</tr>
<tr>
<td>C* Interpolated</td>
<td>Uncertain</td>
</tr>
<tr>
<td>C△ Changed</td>
<td>Uncertain</td>
</tr>
<tr>
<td>No Cor</td>
<td>Lowest</td>
</tr>
</tbody>
</table>

**C Full Correction**

Full Correction level is displayed immediately after a calibration is performed or when a valid Cal Set is applied. If you require optimum accuracy, avoid adjusting analyzer settings after calibration so your measurement remains at this level.

**C* Interpolated Correction**

"C star" appears in the status bar when a measurement is being interpolated. See Interpolation (above) and Interpolation Accuracy.

**C△ Changed Settings**

"C-delta" appears in the status bar when one or more of the following stimulus settings change. The resulting measurement accuracy depends on which parameter has changed and how much it has changed. For optimum accuracy, recalibrate using the new settings.

- **Sweep time**
- IF Bandwidth
- Port power
- Stepped sweep enabled/disabled

**No Corr  No Correction**

The following will cause the PNA to turn Error Correction OFF for the channel:

- Decrease the start frequency
- Increase the stop frequency
- Change start frequency, stop frequency, or number of points with Interpolation OFF.
- Change sweep type

### How to set Interpolation

<table>
<thead>
<tr>
<th>Using front-panel [HARDKEY] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>Press <strong>CAL</strong></td>
<td>1. Click <strong>Cal</strong></td>
</tr>
<tr>
<td>then <strong>[More]</strong></td>
<td>2. then <strong>More</strong></td>
</tr>
<tr>
<td>then <strong>[Interpolation ON/off]</strong></td>
<td>3. then <strong>Interpolation ON/off</strong></td>
</tr>
</tbody>
</table>

### Interpolation

Calibration interpolation adjusts calibration error terms to match changes to the following settings that you make AFTER a calibration is performed or a Cal Set applied.

The Interpolation **ON** setting means that interpolation is enabled for the active measurement. This does not necessarily mean that the measurement is interpolated. When enabled (ON), if interpolation becomes necessary because you change any of the following stimulus settings, then interpolation will be applied. When stimulus settings change while interpolation is OFF, interpolation is NOT applied but instead, error correction is turned OFF.

Interpolation occurs (if enabled) when you change any of the following settings:

- Start frequency increased
- Stop frequency decreased
- Number of points
Note: Decreasing the start frequency, or increasing the stop frequency will always turn correction OFF. (Exception: Power Calibration DOES extrapolate to the start and stop frequencies.)

**Interpolation Accuracy**

When a measurement is interpolated, the accuracy of the measurements cannot be predicted. It may be affected significantly or not at all. Identifying measurement errors in these cases must be determined on a case-by-case basis.

Significant measurement inaccuracy WILL occur when the phase shift between measurement points increases more that 180 degrees. The PNA will incorrectly interpolate the new phase data. For more information, see phase accuracy.

In general, the chances of significant inaccuracy increases when interpolating measurements under the following conditions:

- when increasing, rather than decreasing, the frequency span between measurement points.
- when frequency span between measurement points becomes much greater.
- when measurement frequencies are very high, especially above 10 GHz.

---

Last modified:

- 3-Sep-2008 Removed legacy content
- Sept 12, 20/06 Added link to programming commands
Calibration Thru Methods

- What is a Non-Insertable Device
- Choosing a Thru Method
- Flush Thru
- Adapter Removal
- Swap Adapters (separate topic)
- Defined Thru
- Unknown Thru
- ECal Thru Method Choices

Other Cal Topics

What is a Non-Insertable Device
To understand the Thru method choices, you must first understand what is meant by "Non-Insertable device". These definitions also apply to ECal modules. Substitute "ECal module" for "device". Then see ECal Thru Method Choices.

A non-insertable device is one whose connectors could NOT mate together. They either do not have the same type of connector or they have the same gender. This also means that the test port cables would not mate together, as in the following diagram.

An insertable device is one whose connectors could mate together. They have the same type of connector and opposite, or no, gender. This also means that the test port cables would mate together, as in the following diagram.
Choosing a Thru Method of Calibration

The Thru method is selected from the Cal Wizard. Select the Modify checkbox in the Select DUT Connectors and Cal Kits dialog box.

Notes:
For ECal, the following choices have different meanings. See THRU methods for ECal.
For 4-port calibration, see How can we measure only 3 THRU connections?

Choice for Insertable Devices: FLUSH Thru (also known as Zero-length Thru)

When calibrating for an insertable device, the test ports at your measurement reference plane connect directly together. This is called a zero-length THRU, or Flush THRU meaning that the THRU standard has zero-length: no delay, no loss, no capacitance, and no inductance. Your calibration kit may not have a physical THRU standard because it is assumed you have an insertable device and will be using a zero-length THRU.

Choices for Non-Insertable Devices

The following methods calibrate for a non-insertable device:

- **Adapter Removal** Accurate, but least convenient.
- **Defined Thru**
- **Unknown Thru Cal** Preferred method.

Adapter Removal Calibration

This method is potentially very accurate. However, it requires many connections which increases the chances of inaccurate data.

Two full 2-port calibrations are performed: one with the adapter connected at port 1, and the other with the adapter connected to port 2. The result of the two calibrations is a single full 2-port calibration that includes accurate characterization and removal of the mismatch caused by the adapter.

Performing an Adapter Removal Cal requires:

- a THRU adapter with connectors that match those on the DUT.
- calibration standards for both DUT connectors.
To select Adapter Removal during a SmartCal, select the **Modify** checkbox in the **Select DUT Connectors and Cal Kits** dialog box. The Cal Wizard will guide you through the steps.

Learn how to perform an **Adapter Removal Cal using ECal**.

**Defined Thru** (also known as **Known Thru, Cal Kit Thru, ECal Thru, Characterized Thru**)

Defined Thru uses the THRU definition that is stored in the Cal Kit file or ECal module. The THRU standard may have worn over time, making it not as accurate as when it was new. Defined Thru is usually more accurate than Adapter Removal, but not as accurate as **Unknown Thru** method.

**Notes**

- If performing an ECal, this is the THRU standard in the ECal Module.
- If Defined Thru appears as a potential THRU method in the **SmartCal Wizard**, this means that there is a defined THRU standard in the selected Cal Kit. This could be a **Zero-length Thru**. The SmartCal Wizard will prompt you to connect the required standard when appropriate.

To define a THRU standard in a Cal Kit (not ECal module):

1. From the PNA Menu, click Calibration, **Advanced Modify Cal Kits**.
2. Select the Cal Kit
3. Click Edit Kit
4. Click Add
5. Select THRU
6. Complete the dialog box.

The next time you perform a Guided Cal, this Defined THRU standard will be available if the DUT connector types match the THRU standard.

**Unknown Thru Cal**

Unknown Thru Cal is the **preferred** THRU method of calibrating the PNA to measure a non-insertable device. The Unknown Thru calibration is also known as **Short-Open-Load-Reciprocal Thru** (SOLR) calibration.

- Very easy to perform.
- Better accuracy than **Defined Thru** and usually better than **Adapter Removal**.
- Does not rely on existing standard definitions that may no longer be accurate.
- Causes minimal cable movement if the THRU standard has the same footprint as the DUT. In fact, the DUT can often BE the THRU standard.

**About the Unknown Thru Process**
SmartCal guides you through the process. Although the following process describes ports 1 and 2, Unknown Thru can be performed on any two ports when using a multiport PNA.

1. Perform 1-port cal on port 1.
2. Perform 1-port cal on port 2.
3. Connect Unknown Thru between ports 1 and 2.
4. Measure Unknown Thru.
5. **Confirm Estimated Delay.** This estimate may be wrong if there are too few frequency points over the given frequency span. You can measure the delay value independently and enter that value in the dialog box.

**The Unknown Thru Standard**

- Can have up to about 40 dB of loss and long electrical length.
- Must be reciprocal: S21=S12.
- Must know the phase response to within 1/4 wavelength (see step 5 above).
- Can be the DUT if it meets these conditions.

**Unknown Thru Limitations**

- Unknown Thru is NOT supported during a TRL calibration.
- Beginning with PNA code release 5.25, Unknown Thru CAN be performed using a 4-port PNA-L that does NOT have a reference receiver for each test port. However, a Delta Match Calibration is usually required before the Unknown Thru is measured.
- Unknown Thru is NOT supported on E8801A, E8802A, and E8803A.

**ECal Thru Method Choices**

When the ECal module connectors exactly match the DUT connectors, choose from the following THRU methods:

**ECal Thru as Unknown Thru**  [Learn more about Unknown Thru](#)

- Measures the THRU state of the ECal module as an Unknown Thru.
- The default method when the ECal module connectors match the DUT.
- Very accurate and easy.
- May require a Delta Match Cal.

**Flush Thru (zero-length Thru)**  [Learn more about Flush Thru](#)
• Requires an insertable ECal module / DUT.
• Remove the ECal module and connect the two reference planes directly together for a zero-length thru.
• Accurate, but not as easy as ‘ECal Thru as Unknown Thru’.

**ECal (Defined Thru)**

• Measures the THRU state of the ECal module.
• Very easy, but not very accurate.

**Unknown Thru**

• Remove the ECal module.
• Then connect a Thru adapter to be measured as Unknown Thru.
• May require a Delta Match Cal.

When the ECal module connectors do NOT exactly match the DUT connectors, choose from the following two methods:

**Adapter Removal**

• Can be used with ECal when your DUT is NON-insertable. However, the ECal module MUST be insertable, and the adapter connectors must exactly match the connectors of the DUT as in the following diagram.

**Note:** With PNA release 4.8, adapter removal now performs 2-port measurements on both sides of the adapter. It previously performed 2-port measurements on one side and 1-port measurements on the other. This improves the accuracy of the adapter removal calibration.

---

**ECal User Characterization**
In cases when adapter removal cannot be performed, ECal User Characterization is ALWAYS possible if you have the right adapters. A User Characterization is performed once and stored in the ECal module. However, accuracy is compromised every time you remove, then reconnect, the adapter with the ECal module.

Last Modified:

- 15-Dec-2010  Added link to swap equal adapter
- 20-Feb-2008  Added bullet for default ECal method
Accurate Measurement Calibrations

Calibration accuracy is affected by the type of calibration, quality of the calibration standards, and the care with which the calibration is performed. This section provides additional information about how to make accurate calibrations.

- Measurement Reference Plane
- Effects of Using Wrong Calibration Standards
- Data-based versus Polynomial Calibration Kits
- Accuracy Level of Interpolated Measurement
- Effects of Power Level
- Using Port Extensions
- Isolation Portion of 2-Port Calibration
- Choosing a Thru Method

Learn how to determine the validity of your calibration.

Measurement Reference Plane

Most measurement setups will NOT allow you to connect a device under test (DUT) directly to the PNA front panel test ports. More likely, you would connect your device to test fixtures, adapters, or cables that are connected to the PNA.

A calibration takes place at the points where calibration standards are connected during the calibration process. This is called the measurement reference plane (see graphic). For the highest measurement accuracy, make the calibration reference plane the place where your DUT is connected. When this occurs, the errors associated with the test setup (cables, test fixtures, and adapters used between the analyzer ports and the reference plane) are measured and removed in the calibration process.
Effects of Using Wrong Calibration Standards

Normally, a calibration is performed using a calibration kit that contains standards with connectors of the same type and sex as your DUT.

However, your calibration kit may not always have the same connector type and gender as your device. For example, suppose your device has 3.5mm connectors, but you have a Type-N calibration kit. If you use an adapter to connect the Type-N standards to the 3.5mm test port, then the adapter becomes part of the calibration and NOT part of the test setup. This will result in significant errors in your reflection measurements.

Data-based versus Polynomial Calibration Kits

The Select DUT Connectors and Cal Kits dialog box offers a data-based model and a polynomial model for the newest high-frequency cal kits. See PNA Accessories. The data-based models provide higher accuracy for describing calibration standards than the polynomial models. It is RECOMMENDED that the data-based model be used if the most accurate results are desired.

<table>
<thead>
<tr>
<th>How accurate is the model?</th>
<th>Data-Based Model</th>
<th>Polynomial Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Provides highest calibration accuracy. Eliminates the errors that can be the result of polynomial model approximations.</td>
<td>Provides high calibration accuracy.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>How does the model define calibration standards?</th>
<th>Data-Based Model</th>
<th>Polynomial Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uses S-Parameter measurements.</td>
<td>Uses traditional four-term polynomial calibration standard modeling parameters.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>How do I manually edit the definitions of the calibration standards when using the model?</th>
<th>Data-Based Model</th>
<th>Polynomial Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use the Advanced Modify Cal Kit function.</td>
<td>Use the Advanced Modify Cal Kit function.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>How do I use the Calibration Wizard with the model?</th>
<th>Data-Based Model</th>
<th>Polynomial Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use only the SmartCal (Guided) Calibration method.</td>
<td>Use the SmartCal (Guided) or the Unguided Mechanical Calibration methods.</td>
<td></td>
</tr>
</tbody>
</table>

Learn about the “Expanded Math” feature.

Effects of Power Level

To attain the most accurate error correction, do NOT change the power level after a calibration is performed. However, when changing power within the same attenuator range at which the measurement calibration was performed, S-parameter measurements can be made with only a small degradation of accuracy. If a different attenuator range is selected, the accuracy of error correction is further degraded.

To check the accuracy of a calibration, see Validity of a Calibration.

Using Port Extensions

Use the port extensions feature after calibration to compensate for phase shift of an extended measurement.
reference plane due to additions such as cables, adapters, or fixtures.

Port extensions is the simplest method to compensate for phase shift, mismatch, and loss of the path between the calibration reference plane and the DUT.

Learn how to apply port extensions.

Learn about characterizing a test fixture.

**Isolation Portion of 2-Port Calibration**

The isolation portion of a calibration corrects for crosstalk, the signal leakage between test ports when no device is present. When performing an UNGUIDED 2-port calibration, you have the option of omitting the isolation portion of the calibration.

**Note:** Isolation can be performed on a Smart (Guided) Calibration ONLY remotely using SCPI or COM.

The uncorrected isolation between the test ports of the PNA is exceptional (typically >100dB). Therefore, you should only perform the Isolation portion of a 2-port calibration when you require isolation that is better than 100dB. Perform an isolation calibration when you are testing a device with high insertion loss, such as some filter stopbands or a switch in the open position.

The isolation calibration can add noise to the error model when the measurement is very close to the noise floor of the analyzer. To improve measurement accuracy, set a narrow IF Bandwidth.

**How to perform an isolation calibration**

Isolation is measured when the Load standards are connected to the PNA test ports. For best accuracy, connect Load standards to BOTH test ports each time you are prompted to connect a load standard. If two Loads are not available, connect the untested PNA port to any device that will present a good match.

**Choosing a Thru Method**

When calibrating for a non-insertable device, you must choose a method to calibrate for the THRU error terms. This can have a significant effect on measurement accuracy. Learn more about choosing a thru method.

---

Last Modified:

18-Feb-2011   Removed popups and added remotely to Isolation note.
Validity of a Calibration

This section helps you determine if your calibration is valid and how the analyzer displays correction level information for your measurement.

- Frequency Response of Calibration Standards
- Validating a Calibration
- Quick Check
- ECal Confidence Check
- Verification Kit

See other Calibration Topics

Frequency Response of Calibration Standards

In order for the response of a calibration standard to show as a dot on the smith chart display format, it must have no phase delay with respect to frequency. The only standards that exhibit such "perfect" response are the following:

- 7-mm short (with no offset)
- Type-N male short (with no offset)

There are two reasons why other types of calibration standards show phase delay after calibration:

1. The reference plane of the standard is electrically offset from the mating plane of the test port. Such devices exhibit the properties of a small length of transmission line, including a certain amount of phase shift.

2. The standard is an open termination, which by definition exhibits a certain amount of fringe capacitance and therefore phase shift. Open terminations which are offset from the mating plane will exhibit a phase shift due to the offset in addition to the phase shift caused by the fringe capacitance.

The most important point to remember is that all standards are measured in order to remove systematic errors from subsequent device measurements. As a result, if calibration standards with delay and fringe capacitance are measured as a device after a calibration, they will NOT appear to be "perfect". This is an indication that your analyzer is calibrated accurately and working properly.

Validating a Calibration

At the completion of a calibration or selection of a stored Cal Set, validation can accomplish the following:
**Improve Measurement Accuracy** – Once a measurement calibration has been performed, its performance should be checked before making device measurements. There are several sources of error that can invalidate a calibration: bad cables, dirty or worn calibration standards that no longer behave like the modeled standards, and operator error.

**Verify Accuracy of Interpolation** – You should validate the calibration if you are testing a device and the measurements are uncertain because of interpolation. For more information see Interpolation Accuracy.

**Verify Accuracy of Cal Standards** – To check accuracy, a device with a known magnitude and phase response should be measured.

**Quick Check**

For this test, all you need are a few calibration standards. The device used should not be one of the calibration standards; a measurement of one of these standards is merely a measure of repeatability.

The following reflection and transmission Quick Check tests can be applied to all test ports.

**To verify reflection measurements, perform the following steps:**

1. Connect either an OPEN or SHORT standard to port 1. The magnitude of $S_{11}$ should be close to 0 dB (within a few tenths of a dB).

2. Connect a load calibration standard to port 1. The magnitude of $S_{11}$ should be less than the specified calibrated directivity of the analyzer (typically less than -30 dB).

**To verify transmission measurements:**

1. Connect a THRU cable (or known device representative of your measurement) from port 1 to port 2. Verify the loss characteristics are equivalent to the known performance of the cable or device.

2. To verify $S_{21}$ isolation, connect two loads: one on port 1 and one on port 2. Measure the magnitude of $S_{21}$ and verify that it is less than the specified isolation (typically less than -80 dB).

**Note:** To get a more accurate range of expected values for these measurements, consult the analyzer's specifications.

**ECal Confidence Check**

ECal Confidence Check is a method to check the accuracy of a calibration performed with mechanical standards or an ECal module. The confidence check allows you to measure an impedance state in the ECal module (called the confidence state), and compare it with factory measured data stored in the module.

In order for this test to be valid, the test ports of the ECal module must connect directly to the calibration reference plane (without adapters).

**Note:** In the N469x series of 2-port ECal modules, from the module minimum frequency up to approximately 2 GHz, the confidence state has a very high amount of transmission loss. In this frequency range, calibrated measurements of transmission $S$-parameters for the confidence state may vary much more than expected from the Agilent-characterized data in the measurement memory trace. When comparing the measurement trace and memory trace you, ignore the data for frequencies up to 2 GHz.
How to Perform ECal Confidence Check:

1. Connect ECal module to the analyzer with the USB cable. See Connect ECal Module to the PNA. Note: Terminate any unused ECAL ports with a 50 ohm load.

2. Allow the module to warm up for 15 minutes or until the module indicates READY.

3. Do one of the following to start ECal Confidence Check

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press CAL</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [More]</td>
<td>2. then Cal</td>
</tr>
<tr>
<td>3. then [Ecal]</td>
<td>3. then More</td>
</tr>
<tr>
<td>4. then [Confidence Check]</td>
<td>4. then ECal</td>
</tr>
<tr>
<td></td>
<td>5. then Confidence Check</td>
</tr>
</tbody>
</table>

On the following ECal Confidence Check dialog box:

4. Click Read Module Data. The following occurs:
   - ECal module is set to "confidence state".
   - PNA reads and displays stored data.
   - PNA measures and displays "confidence state".

5. To view a different parameter, select Change Measurement and select the check box for the desired parameter. The default is the active channel parameter.

6. Select the viewing option in the Trace View Options block.

7. Compare the stored and measured data for each measurement parameter.

Notes:

- After exiting ECal Confidence Check, the ECal module remains in the same impedance state and the factory (or user-characterized) data is still stored in the memory trace. Therefore, you can save both the data and memory trace as a *.csv files and import them to a spreadsheet. Learn how.

- If the two traces show excessive difference, there may be a loose or dirty connection at the test ports or damage to the test cables. Carefully inspect the cables and connections. Then clean and gage each connector, and re-calibrate if needed.
The User Characterization setting selects the user-characterization data instead of the factory characterization data (available when a User-Characterization is stored in the ECal module).

**ECal Confidence Check** dialog box help

Compares the accuracy of corrected (calibrated) data with stored data in the ECal module. For the check to be valid, the module test ports must connect directly to the calibration reference plane (without an adapter). Learn more about ECal Confidence Check.

**Measurement**

Change Measurement Opens the Measure dialog box.

**Use ECal Module**

Read Module Data

- Copies stored data from the ECal module to Memory.
- Changes state of ECal module to confidence state.
- Measures and displays confidence state and Memory trace.

**User Characterization** Selects the user-characterization data (stored in the module) instead of the factory characterization data (available when a User-Characterization is stored in the ECal module).

**Scale** Opens the Scale dialog box.

**Show Prompts** Check to show a reminder for the connection (default).

**Trace View Options**

Data and Memory Trace Displays current measurement data and Memory trace.

Data / Memory Performs an operation where the current measurement data is divided by the data in memory.

Data + Memory Performs an operation where the current measurement data is added to the data in memory.
**Verification Kit**

Measuring known devices, other than calibration standards, is a straightforward way of verifying that the network analyzer system is operating properly. Verification kits use accurately known verification standards with well-defined magnitude and phase response. These kits include precision airlines, mismatch airlines, and precision fixed attenuators. Traceable measurement data is shipped with each kit on disk and verification kits may be re-certified by Agilent.

See [Analyzer Accessories](#) for a list of Agilent verification kits.

---

**Last modified:**

- **5-May-2010**  Added Note
- **26-May-2009**  Added ECal conf check save
- **3-Sep-2008**   Removed legacy content
- **9/12/06**      Added link to programming commands
Using ECal

This topic discusses all aspects of performing an ECal:

- **ECal Overview**
- **Connect ECal Module to the PNA**
- **How to Perform a Calibration Using ECal**

See Also:
- ECal User-Characterization
- Restore ECal Module Memory

---

ECal Overview

ECal is a complete solid-state calibration solution. Every ECal module contains electronic standards that are automatically switched into position during a PNA measurement calibration. These electronic standards have been measured at the factory and the data stored within the memory of the ECal module. The PNA uses this stored data, along with the PNA-measured data, to calculate the error terms for a measurement calibration.

ECal modules are available in 2-port and 4-port models and a variety of connector types, covering many frequency ranges. See [Analyzer Accessories](#) for more about available ECal modules and ordering information.

You can perform the following calibrations with ECal:

- 1-Port Reflection calibration
- Full 2-Port calibration
- Full 3-Port calibration
- Full 4-Port calibration

Verify the validity of a mechanical or ECal calibration with [ECal confidence check](#).

**Care and Handling of ECal Modules**

You can improve accuracy, repeatability, and avoid costly repair of equipment in the following ways.

- Practice proper connector care. See [Connector Care](#).
- Protect equipment against ESD damage. Read [Electrostatic Discharge Protection](#).

**Power Level into an ECal module**
NEVER exceed the following Damage levels to the ECal module.

For highest accuracy, do not exceed the following ECal Compression levels when calibrating:

<table>
<thead>
<tr>
<th>Model</th>
<th>Compression level</th>
<th>Damage level</th>
</tr>
</thead>
<tbody>
<tr>
<td>N469x series</td>
<td>-5 dBm</td>
<td>+10 dBm</td>
</tr>
<tr>
<td>N4432A series</td>
<td>-7 dBm</td>
<td>+20 dBm</td>
</tr>
<tr>
<td>N4433A series</td>
<td>+7 dBm</td>
<td>+20 dBm</td>
</tr>
<tr>
<td>N4431x series</td>
<td>+9 dBm</td>
<td>+20 dBm</td>
</tr>
<tr>
<td>8509x series</td>
<td>+9 dBm</td>
<td>+20 dBm</td>
</tr>
</tbody>
</table>

When using the PNA-X, the power level can be increased after calibration with minimal impact on measurement accuracy.

**Connect ECal Module to the PNA**

ECal modules are controlled and powered through a USB connection to the PNA. When you connect the module, the PNA automatically recognizes the type of module, frequency range, and connector type.

See Important First-time USB connection note.

ECal modules connect to the USB port on the front or rear panel of the PNA.

1. Wear a grounded wrist strap when making connections.
2. Connect the USB cable **Type B** connector to the ECal module and the USB cable **Type A** connector to the front or rear panel USB connector of the analyzer, as shown in the following graphics.
Notes:

- Unused ECal modules that have completed a calibration may remain connected to the USB port.
- You can connect and disconnect the ECal module while the analyzer is operating. However, DO NOT connect or disconnect the module while data transfer is in progress. This can result in damage or at least corrupted data.

How to Perform a Calibration Using ECal

Select an ECal module that has connectors of the same type and gender as the DUT. If such an ECal module is not available, a module with connectors different from the DUT can be used by using Advanced Settings or User Characterization.

Connect the ECal module ports to the PNA ports. During the calibration process the PNA can either automatically detect how the ECal module is connected, or the orientation can be performed manually.

1. Connect the ECal module USB cable to the analyzer USB. See Connect ECal Module USB to PNA USB.
2. Allow the module to warm up until it indicates READY.
3. Enter the analyzer settings. See Set Up Measurements.
4. Do one of the following to start the Calibration Wizard

   Using front-panel HARDKEY [softkey] buttons
   1. Press CAL
   2. then [Start Cal]
   3. then [Cal Wizard]

   Using a mouse with PNA Menus
   1. Click Response
   2. then Cal
   3. then Cal Wizard

5. In the Calibration Wizard Begin dialog box, click Use Electronic Cal (ECal).

Note: To calibrate with more than one ECal module, select SmartCal, then choose the ECal modules as your Cal Kits.
### Select Calibration Ports and ECal Module dialog box help

Allows you to select calibration type and settings.

**Cal Type Selection / Configuration** Select the number of ports to calibrate. Then select the port number configuration.

- **4 Port ECal** Available only if using a 4-port PNA. No additional configuration necessary.
- **3 Port ECal** Available only if using a 4-port or 3-port PNA.
- **2 Port ECal**
- **1 Port ECal** (Reflection) Advanced Settings are not available.

**View/Select ECal Module** Click to [Select the ECal module](#) if more than one ECal module is connected to the PNA. Also, [Select the User Characterization](#) within the module. [Learn more about User Characterization.](#)

**Show Advanced Settings** Check to display the Advanced Settings when Next is clicked.

**Back** Return to [Cal Wizard Begin](#) dialog. If checked, you can clear the Save Preferences checkbox to see the Begin page when the Cal Wizard begins.

**Note:** The PNA no longer allows ECal isolation to be performed. The inherent isolation of the PNA is better than that attained with correction using an ECal module.

Terminate any unused ECal ports with a 50 ohm load.

---

### ECal module not found dialog box help

Displays an error message indicating the ECal module is not connected or has not been recognized by the network analyzer.

- **Retry** Check the USB connections and click to continue.

**Notes:**

- If your ECal module is not detected, try to unplug, then reconnect the USB connector to the PNA.
- When the ECal module is connected to the network analyzer for the first time, it may take approximately 30 seconds for the analyzer to recognize the module and make it available for calibration.
- For best accuracy, allow the ECal module to warm-up until it indicates READY.
- Agilent 8509x and N4431 ECal modules, when first connected, draw significantly more current than other modules. This could cause the USB to stop working in certain situations. [See USB limitations.](#)
- See Connect ECal Module USB to PNA USB.

**Select Module and Characterization** dialog box help

**ECal Module**  Select one of the ECal modules that are connected to the PNA.

**Detect Connected ECals**  Click to rescan the USB for ECal modules.

**Available Characterizations**

**ECal Module Memory** - Displays the factory and [user characterizations](#) that are stored in the ECal module.

**PNA Disk Memory** - Displays the user characterizations that are stored in PNA Disk Memory. [Learn more User Characterizations in PNA Disk Memory](#).

Select either the characterization data to use for the calibration. Once selected, that characterization becomes the default selection until the PNA is turned OFF and restarted. When restarted, **Factory** again becomes the default selection.
**Error: Frequency Range** dialog box help

The current cal standards (or ECAL module) does not cover the current frequency range of the measurement. Do one of the following to correct the problem:

- **Cal Kit Class Category**  Not available with ECAL modules.
- **Frequency**  Change the frequency range of the active channel.
- **Edit**  Not available with ECAL modules.
- **Back**  Select a different characterization that covers the required frequency range.
- **Cancel**  Re-characterize the module with an increased frequency range.

**Select DUT Connectors and Cal Kits** dialog box help

If the ECAL module or selected User Characterization has more than one connector type, then the following dialog box is presented which allows you to describe the DUT connector type. Otherwise, click next to proceed to **Advanced Settings** (if checked) or **ECAL Steps**.

**Connectors**

The available connectors are listed for each DUT port.
**Advanced Settings** dialog box help

**Thru #n**
Lists the proposed Thru connections to be made during the calibration process. You can change these Thru connections to better suit your test setup.

- The proposed Thru connections are listed automatically.
- Additional Thru connections can be selected for higher accuracy. [Learn more](#).

**Add Thru**
Click to add a Thru connection. [Learn more](#).

**Remove Thru**
Select a Thru by clicking the "Thru #N" field or the "1st Port / 2nd Port" field. Then click "Remove Thru". This selection is NOT available if the selected Thru is required for the calibration.

**1st Port / 2nd Port**
Click to change the two ports to be included in the Thru connection. The order of the port numbers (1st or 2nd) is not critical.

**Thru Cal Method**
Lists the available Thru Cal methods for the specified port pairs. [Learn about ECal Thru Methods](#).

**Cal Type/ Stds**
Click to invoke the View / Modify Properties of Cal dialog box.

**Do orientation**
When this box is checked (default) the PNA automatically senses the model and direction in which an ECal module port is connected to the PNA ports. If power to the ECal module is too low, it will appear as if there is no ECal module connected. If you use low power and are having this problem, clear this check box to provide the orientation manually.

Orientation occurs first at the middle of the frequency range that you are calibrating. If a signal is not detected, it tries again at the lowest frequency in the range. If you have an **E8361A** or **E836xB** PNA and do an ECal completely within 10 - 20 MHz OR 60 - 67 GHz, you may need to do orientation manually. There may not be sufficient power to orient the ECal module at those frequencies.

**Choose delta match**
Available only when a Delta Match Cal is required.

- Check, then click **Next** to invoke the **Select Cal Set for Delta Match** dialog box.
- Clear - The Cal Wizard uses the **Global Delta Match Cal** if available.
Specify how the ECaI module is connected dialog box help

This dialog box appears when the Do orientation checkbox in the previous dialog box is cleared. Click the ECal Port that is connected to each PNA port.

Electronic Calibration Steps dialog box help

Note: Beginning in PNA Rev. 6.0, ECal can be performed with External triggers. Learn more.

Displays the instructions for each measurement required for calibration.

Measure  Measures the ECal standards.
Done    Click when last standard has been measured.

Saving an ECal Calibration

When complete, you can save the new calibration. Learn how.
28-Jun-2011   Added link to first time note
9-Jan-2009    Updated compression level
3-Sep-2008    Removed legacy content
27-Jun-2008   Added compression levels
4-Sep-2007    Added First time note
14-Sep-2007   MX Added UI
Sept. 12, 2006 MQ Modified images for multiport
ECal User Characterization

- **Overview**
- **How to Perform a User Characterization**
- **Manage Disk Memory**
- **Restore ECal Module Memory**

See also **Using ECal**

**Other Calibration Topics**

**Overview**

A user-characterized ECal module allows you to add adapters to the ECal module, re-measure the standards in the ECal module, INCLUDING the adapters, then add that data to ECal memory or save it to the PNA hard drive. This extends the reference plane from the module test ports to the adapters.

**Why perform a user characterization?**

- If you need to use adapters with your ECal module, you could characterize your ECal module with the adapters attached and perform subsequent ECals in a single step.

- If you have a 4-port ECal module, you could configure the module with adapters of different connector types, then perform a user characterization of the module. When you need to test a DUT with a pair of the connector types on your module, calibrate the PNA with a 1-step ECal using the same two connectors on the User-characterized module.

- If you test devices in a fixture, you could embed the characterization of the fixture in the characterization of the module. To do this, during the mechanical calibration portion of the user characterization, calibrate at the reference plane of the device as you would normally calibrate. Then remove the fixturing to be embedded and insert the ECal module to be characterized. When measuring the ECal module, the PNA removes the effects of the fixturing and stores the measurement results in the user characterized ECal module. Subsequent calibrations with that user-characterized module will also remove the fixture effects.

**Notes:**

- Both 2-port and 4-port ECal modules support user characterization.

- User Characterization does not delete the factory characterization data. The factory data is saved in the ECal module in addition to the user characterization data.

- The ECal Data Wipe Utility is the only way that data can be deleted from the module. Learn more at [http://na.tm.agilent.com/pna/apps/applications.htm](http://na.tm.agilent.com/pna/apps/applications.htm).

- Beginning with A.06.03, a User Characterization can be performed beyond the frequency range of the ECal
module. Although this practice is allowed, calibration accuracy with the extended user characterization is likely to be degraded. To determine the level of degradation, compare measurements of a variety of devices using a PNA with a mechanical cal kit calibration versus an ECal extended user characterization calibration.

- Beginning with A.08.33, you can save up to 12 User Characterizations in a single ECal module. Previous releases allowed up to 5. There are memory limitations. The PNA will determine if the contents of a user characterization will fit inside the module before it is performed.

- Saving a new User Characterization with PNA releases before A.08.33 will erase any User Characterizations greater than 5. This is because the PNA completely rewrites ALL ECal User Characterizations when saving a new one in order to free storage space in the ECal module.

- Beginning with A.08.33, a User Characterization can be performed remotely. See programming commands.

User Characterizations can be saved to the PNA Disk Memory beginning with A.09.00. Learn how.

This feature provides the following benefits:

- A User Characterization using connectors that are NOT included in the supported connector table can NOT be stored to the ECal module. But when stored to disk memory, ANY connector type is allowed by firmware using a description of any length for the User-characterization.

- There is NO limit to the number of data points allowed in a User-characterization stored to disk memory. When stored in the ECal module, the number of data points is limited by the PNA firmware.

- The number of User-characterizations that can be stored to disk memory is limited only by available disk space.

- User-characterizations stored to disk memory can be freely shared between PNAs that have PNA Rev. 9.00 or later.

How to Perform a User Characterization

SUMMARY (A detailed procedure follows.)

1. Select adapters for the module to match the connector configuration of the DUT.

2. Either calibrate the PNA using mechanical standards or recall an existing Cal Set.
3. Measure the ECal module, including adapters, as though it were a DUT.

4. The measurement results are the characterization data that then gets stored inside the module.

**Note**

A 2-port PNA can be used to perform a User Characterization on a 4-port ECal module. However, a 4-port ECal module has SIX different port pairs. The PNA must be recalibrated for each port pair that uses unique connector types or gender.

- If all 4 ECal module ports have the same connector type and gender, then only one PNA calibration is required to measure all six port pairs.
- If all 4 ECal module ports have different connector types or gender, then 6 calibrations are required.

When more than one PNA calibration is required during a User Characterization, then ALL calibrations must be performed using the standard Cal Wizard, saved to Cal Sets, and then recalled from Cal Sets DURING the User Characterization.
Detailed steps to Perform a User Characterization

1. Connect the ECal module to the network analyzer with the USB cable. See Connect ECal Module USB to PNA USB.
2. Allow the module to warm up until it indicates READY.
3. Preset the analyzer.
4. Set up the measurement. For best accuracy, the IF bandwidth should be set to 1 kHz or less.
5. Start and complete the Characterize ECal Module Wizard:

Using front-panel HARDKEY [softkey] buttons

1. Press CAL
2. then [More]
3. then [ECal]
4. then [Characterize ECal Module]

Using a mouse with PNA Menus

1. Click Response
2. the Cal
3. then More
4. then ECal
5. then Characterize ECal Module
Select Module and Location dialog box help

**ECal Module**  Select one of the ECal modules that are connected to the PNA.

**Detect Connected ECals**  Click to rescan the USB for ECal modules.

**Location**

- **ECal Module Memory**  Click Next to see the following dialog.

- **PNA Disk Memory**  Enter a Characterization Name. This name appears when selecting a user characterization to be used with subsequent calibrations.
  
    - Learn how to manage characterizations that are stored to PNA disk memory.
    - See the benefits of storing the User Characterization to PNA Disk Memory.

**Keyboard**  Launches a keypad that can be used to type a characterization name from the PNA front panel.

**Next**  Click to continue to the Select Connectors for the Characterization dialog box.

See note regarding extended frequency use.

Select User Number for new characterization dialog box help

Scroll to view all of the parameters of the stored characterizations. Select an empty location or select to overwrite an existing characterization.

**Next**  Click to continue to the Select Connectors for the Characterization dialog box.

See note regarding extended frequency use.
Select Connectors for the Characterization dialog box help

Connector Notes

When performing an ECaL User Characterization, do NOT use a custom connector name that you added to this list. If you need to use a custom-defined connector type, select "Type B", or one of the "Type A" variations from the list of connectors for each port.

A User Characterization using connectors that are NOT included in the supported connector table cannot be stored to the ECaL module. But when stored to disk memory, ANY connector type is allowed. Learn more about storing to PNA Disk Memory.

Select the adapters for the ECaL module test ports. Select No adapter if no adapter is used on a port.

PORT A Lists the connector types available for Port A.
PORT B Lists the connector types available for Port B.
PORT C Lists the connector types available for Port C (available with a 4-port ECaL module).
PORT D Lists the connector types available for Port D (available with a 4-port ECaL module).
Next Click to continue to the Calibrations to perform or recall dialog box.
Calibrations to perform or recall dialog box help

The PNA must be calibrated before measuring the ECal module and necessary adapters. This dialog box displays the number and types of mechanical calibrations required for the characterization.

**Guide me through this cal now** Click to perform a Guided calibration. A calibration kit is required for each connector type.

**Note:** TRL calibrations can NOT be performed on a 4-port PNA during the calibration portion of a User Characterization. However, this type of Cal can be performed using the Cal Wizard, saved to a Cal Set, then recalled at this point in the User Characterization.

If more than one calibration is required, this selection is not available. See Note.

Let me recall this cal from a cal set Click to select an existing Cal Set. You cannot select a Cal Set that is currently in use. Learn more about Using Cal Sets.

Next Click to continue to either the Select Cal Kits or the Select Cal Set dialog box.

Select Cal Kits dialog box help

Provides a list of calibration kits to perform the calibration. Select the Cal Kit you will use for each port.

**Enable Unknown Thru for characterizing the module** Check to enable. This reduces the number of steps required to characterize the THRU standard. This setting is available only on PNA models with one reference receiver per test port.

Next Click to continue to the Select Cal Set dialog box.
Select Cal Set dialog box help

The calibration that you perform will be written to a Cal Set. This dialog box allows you to select a Cal Set to overwrite, or to write to a new Cal Set. The current choice is visible below the Select Cal Set button.

Select Cal Set  Click to open the Select A Cal Set dialog box.
Create new Cal Set  Check to create a new Cal Set to store the calibration. Clear to select and overwrite a stored Cal Set.
Next  Click to continue to the Guided Calibration Steps dialog box.

Note: Remember the Cal Set name for future reference.

Guided Calibration Steps dialog box help

Instructs you to connect each calibration standard to the measurement port.

Measure  Click to measure the standard.
Back  Click to repeat one or more calibration steps.
Done  Click after a standard is re-measured and all measurements for the calibration are complete.
Next  Click to continue to the next calibration step. (Does not measure the standard.)
Cancel  Exits Calibration Wizard.

The Specify nominal delay or Guided Calibration completed dialog box appears when the steps are completed.
Specify nominal delay dialog box help

This dialog ONLY appears when Adapter Removal or Unknown Thru calibrations are performed.

The following values were estimated from the measurement. Most of the time, they are adequate. However, for CW sweep or frequency sweep with large step sizes, the accuracy of the values may be improved.

**Nominal adapter delay** To improve this value, measure and record the delay of the adapter with a dense step size. Enter that value here.

**Nominal phase offset** (Waveguide ONLY). To improve this value, measure and record the phase offset of the Waveguide adapter with dense step size. Enter that value here.

When one connector is coax and the other connector is waveguide, the phase offset has an ambiguity of 180 degrees. For consistency, the estimate provided here is always between 0 and 180 degrees. You can change this estimate to any value between -180 degrees and +180 degrees.

Guided Calibration completed dialog box help

Allows you to finish the calibration and continue to the next characterization steps.

- **No. Finish now** Select to save Cal Set data.
- **Yes** Allows selection of Save options.
- **Next** Click to continue to the Exit to Inspect Quality of Calibration dialog box.
**Exit to Inspect Quality of Calibration** dialog box help

Allows you to exit User Characterization to validate the calibration before proceeding with the characterization.

- **Back** Allows you to repeat calibration.
- **Next** Click to continue to the Characterization Steps dialog box.
- **Cancel** Exits the Calibration.

To return to the current step:

2. In the Select user number for new characterization dialog box, click **Next**.
3. In the Select Connectors for Characterization dialog box, click **Next**. (Previous entry is stored in memory.)
4. In the Calibrations to perform or recall dialog box, recall the Cal Set that you just performed.

**Characterization Steps** dialog box help

Describes the instructions for each measurement required for characterization.

- **Measure** Measures the ECal module.
- **Next** Click to continue to the Information for the New Characterization dialog box when measurements are complete.
Information for the New Characterization dialog box help

Allows you to describe the properties of the User Characterization.

**Next** Click to continue to the Write Characterized Data to the ECal module dialog box.

To minimize the number of characters, use the following 3-character codes to describe the connectors listed. A User Characterization using connectors that are NOT included on this list can NOT be stored to the ECal module. But when stored to disk memory, ANY connector type is allowed. Learn more about storing to PNA Disk Memory.

<table>
<thead>
<tr>
<th>Connector Type</th>
<th>3-Character Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0 mm female</td>
<td>10F</td>
</tr>
<tr>
<td>1.0 mm male</td>
<td>10M</td>
</tr>
<tr>
<td>1.85 mm female</td>
<td>18F</td>
</tr>
<tr>
<td>1.85 mm male</td>
<td>18M</td>
</tr>
<tr>
<td>2.4 mm female</td>
<td>24F</td>
</tr>
<tr>
<td>2.4 mm male</td>
<td>24M</td>
</tr>
<tr>
<td>2.92 mm female</td>
<td>29M</td>
</tr>
<tr>
<td>2.92 mm male</td>
<td>29F</td>
</tr>
<tr>
<td>3.5 mm female</td>
<td>35F</td>
</tr>
<tr>
<td>3.5 mm male</td>
<td>35M</td>
</tr>
<tr>
<td>7-16 female</td>
<td>16F</td>
</tr>
<tr>
<td>7-16 male</td>
<td>16M</td>
</tr>
<tr>
<td>Type F female</td>
<td>F7F</td>
</tr>
<tr>
<td>Type F male</td>
<td>F7M</td>
</tr>
<tr>
<td>N50 female</td>
<td>N5F</td>
</tr>
<tr>
<td>N50 male</td>
<td>N5M</td>
</tr>
<tr>
<td>N75 female</td>
<td>N7F</td>
</tr>
<tr>
<td>N75 male</td>
<td>N7M</td>
</tr>
<tr>
<td>APC 7</td>
<td>7MM</td>
</tr>
</tbody>
</table>
The PNA writes User Characterization and factory characterization data to either the PNA disk memory or the ECal module memory.

**Write**  Click to write data.

The Summary of new user characterization dialog box opens after data is saved to module.

- Existing data will be overwritten is you selected a User Characterization number that already has data.
  Learn more
- For more information, see Restore ECal module memory.
- The ECal Data Wipe Utility is the only way that data can be deleted from the module. Learn more at http://na.tm.agilent.com/pna/apps/applications.htm.
Summary of new user characterization dialog box help

Verify the status of the ECal User Characterization.

- ECal module model number
- summary from user characterization

Cancel  Click to exit (characterization complete).
Finish  Click to exit (characterization complete).

Manage ECal User Characterizations in Disk Memory

Normally, user characterizations that are stored in PNA disk memory can be used indefinitely without needing them to be managed. However, this dialog allows you to backup the characterizations in case they are accidentally erased, or to save them to a file that can be moved to another PNA.

How to Manage ECal User Characterizations in Disk Memory

- **Using front-panel HARDKEY [softkey] buttons**
  1. Press **Cal**
  2. then [More]
  3. then [ECal]
  4. then [Manage ECal Disk Memory]

- **Using a mouse with PNA Menus**
  1. Click **Response**
  2. then **Cal**
  3. then **More**
  4. then **ECal**
  5. then **Manage ECal Disk Memory**

Programming Commands
Manage ECal User Characterizations in Disk Memory dialog box help

This dialog allows you to save user characterizations to files in case they are accidentally erased, or so they can be moved to another PNA.

**ECal Module**  Select an ECal Module from the list for which user characterizations are currently stored in PNA disk memory.

**Save As**  Saves a User Characterization that is currently in PNA disk memory to a *.euc file. This file can be used as a backup in case the archive file is accidentally deleted, or allows you to move the file to another PNA to be used with the selected ECal Module.

**Import**  Loads a previously saved *.euc file for use on the PNA with the specified ECal module.

**Delete**  Removes a User Characterization from PNA disk memory.

---

**Restore ECal Module Memory**

When user-characterized data is written to the ECal module, the entire contents of ECal memory is also written to the PNA hard drive, including the factory ECal data. In the unlikely event that your ECal module memory is lost, you can restore all ECal data to ECal memory.

**Caution:** If a new factory cal was performed after the ECal memory was written to the PNA, the new factory cal data will also be overwritten.

**Note:** An ECal Data Wipe Utility destroys all user data per US DoD 5220.22-M. Learn more at [http://na.tm.agilent.com/pna/apps/applications.htm](http://na.tm.agilent.com/pna/apps/applications.htm)
### How to Restore ECAl Module Memory

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then <strong>[Service]</strong></td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then <strong>[Utilities]</strong></td>
<td>3. then <strong>Service</strong></td>
</tr>
<tr>
<td>4. then <strong>[Restore...]</strong></td>
<td>4. then <strong>Utilities</strong></td>
</tr>
<tr>
<td></td>
<td>5. then <strong>Restore...</strong></td>
</tr>
</tbody>
</table>

No Programming commands are available for this feature.

**Module to be restored** dialog box help

Verify the Serial number of the module to be restored. If two modules are connected to the PNA, choose the one to have data restored.

**Next**  Click to write data to the module.

Last modified:
15-Mar-2011  Added benefits of Disk Memory
22-Oct-2010  Clarified data delete
24-Aug-2009  Added PNA Disk Memory (9.0)
3-Nov-2008  Added 12 chars and remote link (8.33)
3-Sep-2008  Removed legacy content
28-Aug-2008  Clarified note about restore
13-Aug-2008  Added note about data wipe
25-Apr-2007  Added note about can NOT delete data.
20-Sept-2006  MX Modified for cross-browser
TRL Calibration

TRL (Thru, Reflect, Line) represents a family of calibration techniques that measure two transmission standards and one reflection standard to determine the 2-port 12-term error coefficients. For example, TRM (Thru, Reflect, Match), LRL (Line, Reflect, Line), LRM (Line, Reflect, Match) are all included in this family.

The traditional SOLT calibration measures one transmission standard (T) and three reflection standards (SOL) to determine the same error coefficients.

- Why Perform a TRL Cal?
- The TRL Calibration Process
- TRL Cal Kits
- Cal Standards Used in TRL
- TRL in 4-port PNA a

Why Perform a TRL Cal?

TRL calibration is extremely accurate, in most cases more accurate than an SOLT cal. However, very few calibration kits contain TRL standards. TRL Cal is most often performed when you require a high level of accuracy and do not have calibration standards in the same connector type as your DUT. This is usually the case when using test fixtures, or making on-wafer measurements with probes. Therefore, in some cases you must construct and characterize standards in the same media type as your DUT configuration. It is easier to manufacture and characterize three TRL standards than the four SOLT standards.

Another advantage of TRL calibration is that the TRL standards need not be defined as completely and accurately as the SOLT standards. While SOLT standards are completely characterized and stored as the standard definition, TRL standards are modeled, and not completely characterized. However, TRL cal accuracy is directly proportional to the quality and repeatability of the TRL standards. Physical discontinuities, such as bends in the transmission lines and beads in coaxial structures, will degrade the TRL calibration. The connectors must be clean and allow repeatable connections.

To learn more about Cal Standard requirements, see Cal Standards Used in TRL.

The TRL Cal Process

Although TRL can be performed using the Cal Wizard Unguided Cal selection, the following process uses the easier SmartCal selection. Both selections require that you already have TRL calibration standards defined and included in a PNA cal kit.

1. Preset the PNA
2. Set up a measurement and the desired stimulus settings.
3. Click Calibration / Calibration Wizard
4. Click **SmartCal (Guided Cal)**.

5. **Select the DUT connectors and Cal Kit** for each port. The LOWEST port number of each **port pair** MUST include TRL standards. TRL appears as the **Cal Method**.

6. Check **Modify Cal, Next**, then **View/Modify** to change **default TRL options** if necessary.

7. Follow the prompts to complete the calibration.

8. **Check the accuracy** of the calibration

---

### TRL Cal Kits

Agilent Technologies offers two cal kits that include the required standards to perform a TRL calibration: 85050C (APC 7mm) and 85052C (3.5mm). Both kits include the traditional Short, Open, and Load standards. (The Thru standard, not actually supplied, assumes a **zero-length Thru**). In addition, the kits include an airline which is used as the LINE standard. To use the airline, the kits include an airline body, center conductor, and insertion / extraction tools. The APC 7 kit includes an adapter to connect the airline to the APC connector.

### Cal Standards Used in TRL

These standards must be defined in your TRL cal kit:

**THRU**

**Note:** **All THRU calibration methods** are supported in a TRL Cal **EXCEPT** Unknown Thru.

- The THRU standard can be either a zero-length or non-zero length. However, a zero-length THRU is more accurate because it has zero loss and no reflections, by definition.
- The THRU standard cannot be the same electrical length as the LINE standard.
- If the insertion phase and electrical length are well-defined, the THRU standard may be used to **set the reference plane**.
- Characteristic impedance of the THRU and LINE standards defines the reference impedance of the calibration.
- If a THRU standard with the correct connectors is **NOT** available, an adapter removal cal can be performed.

**REFLECT**

- The REFLECT standard can be anything with a high reflection, as long as it is the same when connected to both PNA ports.
- The actual magnitude of the reflection need not be known.
- The phase of the reflection standard must be known within 1/4 wavelength.
- If the magnitude and phase of the reflection standard are well-defined, the standard may be used to **set the reference plane**.
**LINE**

The LINE and THRU standards establish the reference impedance for the measurement after the calibration is completed. Learn more. TRL calibration is limited by the following restrictions of the LINE standard:

- Must be of the same impedance and propagation constant as the THRU standard.
- The electrical length need only be specified within 1/4 wavelength.
- Cannot be the same length as the THRU standard.
- A TRL cal with broad frequency coverage requires multiple LINE standards. For example, a span from 2 GHz to 26 GHz requires two line standards.
- Must be an appropriate electrical length for the frequency range: at each frequency, the phase difference between the THRU and the LINE should be greater than 20 degrees and less than 160 degrees. This means in practice that a single LINE standard is only usable over an 8:1 frequency range (Frequency Span / Start Frequency). Therefore, for broad frequency coverage, multiple lines are required.
- At low frequencies, the LINE standard can become too long for practical use. The optimal length of the LINE standard is 1/4 wavelength at the geometric mean of the frequency span (square root of $f_1 \times f_2$).

**MATCH**

If the LINE standard of appropriate length or loss cannot be fabricated, a MATCH standard may be used instead of the LINE.

- The MATCH standard is a low-reflection termination connected to both Port 1 and Port 2.
- The MATCH standard may be defined as an infinite length transmission line OR as a 1-port low reflect termination, such as a load.
- When defined as an infinite length transmission line, both test ports must be terminated by a MATCH standard at the same time. When defined as a 1-port load standard, the loads are measured separately. The loads are assumed to have the same characteristics.
- The impedance of the MATCH standard becomes the reference impedance for the measurement. For best results, use the same load on both ports. The load may be defined using the data-based definition, the arbitrary impedance definition, or the fixed load definition.

**See Also**

- See [Modify Calibration Kits](#) for detailed information about creating and modifying Calibration kit definitions.
- For more information, read [Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers (Application Note 1287-11)](#)

**TRL on a 4-port PNA and with an External Test Set**

Beginning with the PNA code revision 5.25, TRL CAN be performed on a 4-port PNA and with an External Test Set enabled. Previously, a TRL calibration required a PNA with a reference receiver for each test port. With the new
TRL method, a Delta Match Calibration is first performed and applied. The accuracy of this TRL cal greatly depends on the accuracy of the Delta Match Calibration. With an accurate Delta Match Calibration, the difference in accuracy between a traditional TRL cal and this TRL cal is negligible.

**How to Perform a TRL Cal on a multiport PNA**

1. Click **Calibration, Cal Wizard**.
2. Select a TRL cal kit for the ports to be calibrated.
3. During the calibration, the Cal Wizard prompts you for a **valid Delta Match Cal**.

---

Last modified:

- 16-Sep-2010  Slightly modified for cals on apps and added link to TRL Options.
- 14-Apr-2008  Added App Note link
- 9/12/06  with Ext Test Set
Calibration Preferences Wizard

Two Cal Preferences are set from this Wizard:

1. Whether or not to show the first 'Method' Page of the Cal Wizard
2. Select and order the Cal Types that are available during a SmartCal with Mechanical Standards

To change either of these choices, you must select Yes, Enable the calibration preferences at the first Wizard page.

How to change Cal Preferences

Programming commands are NOT available for the preference settings discussed in this topic, although there are other Cal Preferences that can be set remotely.

Using front-panel HARDKEY [softkey] buttons

1. Press CAL
2. then [Start Cal]
3. then [Preferences]

Using a mouse with PNA Menus

1. Click Response
2. then Cal
3. then Start Cal
4. then Preferences

Cal Preferences Wizard dialog box help

Use this dialog to change either of the following preferences:

- Show or Hide the first page of the Cal Wizard
- Select order of calibrations that are offered.

To change either of these choices, you must select Yes, Enable the calibration preferences.
**Cal Preferences of...** dialog box help

Use this dialog to change which Cal method to perform.

After making this selection, the first page of the Cal Wizard will not be shown on subsequent calibrations.

To change ONLY the order of Cal Types that are offered, and none of these preferences, you must do the following:

1. Select one of these choices and click **Next**.
2. Select and order the Cal Types, then click **Next**.
3. Click **Finish**.
4. Click **Cal**, then **Cal Wizard**.
5. On the first Cal Wizard page that shows, click **Back**, then clear the **Preferences** checkbox.
Cal Type Preferences dialog box help

This dialog is used to set which Cal Types are available, and the order in which they are selected as the default choice, during a SmartCal with Mechanical Standards. This setting is also used to set the default Cal Type for Guided calibrations using SCPI or COM.

The specified Cal Type order should allow you to make fewer changes to the Cal Type during a SmartCal with Mechanical Standards.

For example, in the above image, the first Cal Type on the list is TRL. When doing a SmartCal with Mechanical Standards:

- If a TRL Cal Kit is available for the specified DUT connectors, then TRL becomes the default Cal Type.
- If a TRL Cal Kit is NOT available, then the second Cal Type on the list (SOLT) is evaluated for compatibility with the available Cal Kits, and so forth with the Cal Types that remain on the list.
- If TRL is removed from the list, that Cal Type is NOT available for selection during a SmartCal with Mechanical Standards.

Learn more about Cal Types.
See where you choose Cal Type during a SmartCal

Prioritized list of choices for default Cal Type  Shows the current list of Cal Types and the order in which they will be selected for Mechanical calibrations.

Change  Click to invoke the Modify list of default Cal Types dialog.

Restore factory defaults  Returns the list to the original selections and order. The factory defaults are in order of accuracy from highest (TRL) to lowest (QSOLT).

Cancel  Closes the dialog without making changes.

Notes:

- Your Cal Type settings are saved only until the PNA application is closed. When re-opened, the factory default settings are restored.

Learn more about QSOLT Calibration
Modify list of default Cal Types dialog box help

Use this dialog to Add, Remove, and re-order the available Cal Types. There must be at least ONE selected Cal Type to perform a SmartCal with Mechanical Standards.

**Unselected Cal Types**  Cal Types in this list will not be presented as a choice during a Calibration.

**Selected Cal Types**  Cal Types in this list will be presented, in order, as the default choice during a Calibration. Click a Cal Type to select it, then click the following buttons to perform that operation.

**Add / Remove buttons**  Click to Add and Remove the selected Cal Types from the Selected Cal Types list.

**Move Up / Down**  Click to re-order the Selected Cal Types list.

Cal Preferences Complete dialog box help

Either Enable or Disable Cal Preferences.

See how to select ONLY Cal Type Preferences and continue to show the first Cal Wizard page.
<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3-Sep-2008</td>
<td>Removed legacy content</td>
</tr>
<tr>
<td>19-Mar-2008</td>
<td>Small cosmetic edits</td>
</tr>
<tr>
<td>1-Jan-2007</td>
<td>MX added UI</td>
</tr>
</tbody>
</table>
Measurement Errors

You can improve accuracy by knowing how errors occur and how to correct for them. This topic discusses the sources of measurement error and how to monitor error terms.

- Drift Errors
- Random Errors
- Systematic Errors
  - 3-Port Error Terms
  - 4-Port Error Terms
- Monitoring Error Terms

See other Calibration Topics

Drift Errors

Drift errors are due to the instrument or test-system performance changing after a calibration has been done. Drift errors are primarily caused by thermal expansion characteristics of interconnecting cables within the test set and conversion stability of the microwave frequency converter and can be removed by re-calibrating.

The time frame over which a calibration remains accurate is dependent on the rate of drift that the test system undergoes in your test environment.

Providing a stable ambient temperature usually minimizes drift. For more information, see Measurement Stability.

Random Errors

Random errors are not predictable and cannot be removed through error correction. However, there are things that can be done to minimize their impact on measurement accuracy. The following explains the three main sources of random errors.

Instrument Noise Errors

Noise is unwanted electrical disturbances generated in the components of the analyzer. These disturbances include:

- Low level noise due to the broadband noise floor of the receiver.
- High level noise or jitter of the trace data due to the noise floor and the phase noise of the LO source inside the test set.

You can reduce noise errors by doing one or more of the following:

- Increase the source power to the device being measured - ONLY reduces low-level noise.
Narrow the IF bandwidth.

Apply several measurement sweep averages.

**Switch Repeatability Errors**

Mechanical RF switches are used in the analyzer to switch the source attenuator settings. Sometimes when mechanical RF switches are activated, the contacts close differently from when they were previously activated. When this occurs, it can adversely affect the accuracy of a measurement. You can reduce the effects of switch repeatability errors by avoiding switching attenuator settings during a critical measurement.

**Connector Repeatability Errors**

Connector wear causes changes in electrical performance. You can reduce connector repeatability errors by practicing good connector care methods. See Connector Care.

**Systematic Errors**

Systematic errors are caused by imperfections in the analyzer and test setup.

- They are repeatable (and therefore predictable), and are assumed to be time invariant.
- They can be characterized during the calibration process and mathematically reduced during measurements.
- They are never completely removed. There are always some residual errors due to limitations in the calibration process. The residual (after measurement calibration) systematic errors result from:
  - imperfections in the calibration standards
  - connector interface
  - interconnecting cables
  - instrumentation

**Reflection** measurements generate the following three systematic errors:

- **Directivity**
- **Source Match**
- **Frequency Response Reflection Tracking**

**Transmission** measurements generate the following three systematic errors:

- **Isolation**
- **Load Match**
- **Frequency Response Transmission Tracking**
Notes about the following Systematic Error descriptions:

- The figures for the following six systematic errors show the relevant hardware configured for a forward measurement. For reverse measurements, internal switching in the analyzer makes Port 2 the source and Port 1 the receiver. 'A' becomes the transmitted receiver, 'B' becomes the reflected receiver, and 'R2' becomes the reference receiver. These six systematic errors, times two directions, results in 12 systematic errors for a two port device.

- For simplicity, it may be stated that ONE standard is used to determine each systematic error. In reality, ALL standards are used to determine ALL of the systematic errors.

- The following describes an SOLT calibration. This does not apply to TRL, or other types of calibration.

**Directivity Error**
All network analyzers make reflection measurements using directional couplers or bridges.

With an ideal coupler, only the reflected signal from the DUT appears at the 'A' receiver. In reality, a small amount of incident signal leaks through the forward path of the coupler and into the 'A' receiver. This leakage path, and any other path that allows energy to arrive at the 'A' receiver without reflecting off the DUT, contributes to directivity error.

**How the Analyzer Measures and Reduces Directivity Error**
1. During calibration, a load standard is connected to Port 1. We assume no reflections from the load.
2. The signal measured at the 'A' receiver results from the incident signal leakage through the coupler and other paths.
3. Directivity error is mathematically removed from subsequent reflection measurements.

**Isolation Error**
Ideally, only signal transmitted through the DUT is measured at the 'B' receiver.

In reality, a small amount of signal leaks into the 'B' receiver through various paths in the analyzer.

The signal leakage, also known as crosstalk, is isolation error which can be characterized and reduced by the analyzer.
How the Analyzer Measures and Reduces Isolation Error

1. During calibration, load standards are connected to both Port 1 and Port 2.
2. The signal measured at the 'B' receiver is leakage through various paths in the analyzer.
3. This isolation error is mathematically removed from subsequent transmission measurements.

Source Match Error

Ideally in reflection measurements, all of the signal that is reflected off of the DUT is measured at the 'A' receiver. In reality, some of the signal reflects off the DUT, and multiple internal reflections occur between the analyzer and the DUT. These reflections combine with the incident signal and are measured at the 'A' receiver, but not at the 'R' receiver.

This measurement error is called source match error which can be characterized and reduced by the analyzer.

How the Analyzer Measures and Reduces Source Match Error

1. During calibration, all reflection standards are connected to Port 1. Known reflections from the standards are measured at the 'A' receiver.
2. Complex math is used to calculate source match error.
3. Source match error is mathematically removed from subsequent reflection and transmission measurements.

**Load Match Error**

Ideally in transmission measurements, an incident signal is transmitted through the DUT and is measured at the 'B' receiver. In reality, some of the signal is reflected off of Port 2 and other components and is not measured at the 'B' receiver. This measurement error is called load match error which can be characterized and reduced by the analyzer.

How the Analyzer Measures and Reduces Load Match Error

1. The Port 1 and Port 2 test connectors are mated together for a perfect zero-length thru connection. If this is not possible, a characterized thru adapter is inserted. This allows a known amount of incident signal at Port 2.

2. The signal measured at the 'A' receiver is reflection signal off of Port 2.

3. The resulting load match error is mathematically removed from subsequent transmission and reflection measurements.

**Frequency Response Reflection Tracking Error**

Reflection measurements are made by comparing signal at the 'A' receiver to signal at the 'R1' receiver. This is called a ratio measurement or "A over R1" (A/R1).

For ideal reflection measurements, the frequency response of the 'A' and 'R1' receivers would be identical. In reality, they are not, causing a frequency response reflection tracking error. This is the vector sum of all test variations in which magnitude and phase change as a function of frequency. This includes variations contributed by:

- signal-separation devices
- test cables
- adapters
• variations between the reference and test signal paths

Frequency response reflection tracking error can be characterized and reduced by the analyzer.

**How the Analyzer Measures and Reduces Frequency Response Reflection Tracking Error.**

1. During calibration, all reflection standards are used to determine reflection tracking.

2. The average 'A' receiver response is compared with the 'R1' receiver response.

3. Complex math is used to calculate Frequency Response Reflection Tracking Error (see the following diagram). This frequency response reflection tracking error is mathematically removed from subsequent DUT measurements.

**Note:** In reflection response calibrations, only a single calibration standard is measured (open or short) and thus only its contribution to the error correction is used.

**Frequency Response Transmission Tracking Error**

Transmission measurements are made by comparing signal at the 'B' receiver to signal at the 'R1' receiver. This is called a ratio measurement or "B over R1" (B/R1).

For ideal transmission measurements, the frequency response of the 'B' and 'R1' receivers would be identical.

In reality, they are not, causing a frequency response transmission tracking error. This is the vector sum of all test variations in which magnitude and phase change as a function of frequency. This includes variations contributed
by:

- signal-separation devices
- test cables
- adapters
- variations between the reference and test signal paths

Frequency response transmission tracking error can be characterized and reduced by the analyzer.

How the Analyzer Measures and Reduces Frequency Response Transmission Tracking Error.

1. During calibration, the Port 1 and Port 2 test connectors are mated together for a perfect zero-length thru connection. If this is not possible, a characterized thru adapter is inserted. This allows a known amount of incident signal to reach Port 2.

2. Measurements are made at the 'B' and 'R1' receivers.

3. Complex math is used to calculate Frequency Response Transmission Tracking Error (see the following diagram). This frequency response transmission tracking error is mathematically removed from subsequent DUT measurements.
3-Port Error Terms

The following flow diagram displays the 3-port error term model:

where:

- **E** = error term
- **DIR** = Directivity
- **MAT** = Forward Source Match and Reverse Load Match
- **TRK** = Forward Reflection Tracking and Reverse Transmission Tracking

4-Port error terms

A full 4-port calibration requires the following terms:

Learn about the port numbering convention for error terms.

<table>
<thead>
<tr>
<th>Source Port</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 DIR 1,1</td>
<td>LDM 1,2</td>
<td>LDM 1,3</td>
<td>LDM 1,4</td>
<td></td>
</tr>
<tr>
<td>RTRK 1,1</td>
<td>TTRK 1,2</td>
<td>TTRK 1,3</td>
<td>TTRK 1,4</td>
<td></td>
</tr>
<tr>
<td>SRM 1,1</td>
<td>XTLK 1,2</td>
<td>XTLK 1,3</td>
<td>XTLK 1,4</td>
<td></td>
</tr>
<tr>
<td>2 LDM 2,1</td>
<td>DIR 2,2</td>
<td>LDM 2,3</td>
<td>LDM 2,4</td>
<td></td>
</tr>
<tr>
<td>TTRK 2,1</td>
<td>RTRK 2,2</td>
<td>TTRK 2,3</td>
<td>TTRK 2,4</td>
<td></td>
</tr>
<tr>
<td>XTLK 2,1</td>
<td>SRM 2,2</td>
<td>XTLK 2,3</td>
<td>XTLK 2,4</td>
<td></td>
</tr>
<tr>
<td>3 LDM 3,1</td>
<td>LDM 3,2</td>
<td>DIR 3,3</td>
<td>LDM 3,4</td>
<td></td>
</tr>
</tbody>
</table>
Reflection terms

- DIR: Directivity
- RTRK: Reflection Tracking
- SRM: Source Match

Transmission terms

- LDM: Load Match
- TTRK: Transmission Tracking
- XTLK: Cross Talk

**How can we measure only 3 THRU connections?**

On a 4-port PNA, a full 4-port cal can be performed while measuring only 3 THRU connections. Measuring more than 3 THRU connections will give higher accuracy.

By measuring all of the reflection terms, and 3 transmission THRU connections, there is adequate information available to calculate the remaining transmission terms. The following is a high level explanation of the concept. The actual calculations are much more complex.

To simplify, let’s substitute letters (A,B,C,D) for port numbers from the diagram above so that they can be combined without confusion. Also for simplicity, let’s assume that the source match and directivity errors are zero.

- The reflection errors are all measured (AA, BB, CC, DD).
- Lets assume we measure a THRU between ports AB, AC, AD. The reverse direction for these THRUs are
also measured at the same time (BA, CA, DA).

- The terms left to calculate are BC, CB, BD, DB, CD, DC.

The following shows how the BC term is calculated from BA and AC:

$$\frac{BA \cdot AC}{AA} = \frac{B \cdot A \cdot C}{A} = BC$$

Similarly:

- CB is calculated from CA and AB
- BD is calculated from BA and AD
- DB is calculated from AB and DA
- CD is calculated from CA and AD
- DC is calculated from DA and AC

**Monitoring Error Terms using Cal Set Viewer**

You can use **Cal Set Viewer** to monitor the measured data and the calculated error term. This will help to determine the health of your PNA and the accuracy of your measurements.

By printing or saving the error terms, you can periodically compare current error terms with previously recorded error terms that have been generated by the same PNA, measurement setup, and calibration kit. If previously generated values are not available, refer to Typical Error Term Data in Appendix A, "Error Terms", of the Service Guide.

**Note**: The service guide for your PNA is available at [http://www.agilent.com/find/pna](http://www.agilent.com/find/pna). It is also on the CDROM that was shipped with your PNA.

- A stable system should generate repeatable error terms over about six months.
- A sudden shift in error terms over the same frequency range, power, and receiver settings, may indicate the need for troubleshooting system components. For information on troubleshooting error terms, see Appendix A, "Error Terms", of the Service Guide.
- A subtle, long-term shift in error terms often reflects drift or connector and cable wear. The cure is often as simple as cleaning and gauging connectors or inspecting cables.

**Viewing Cal Set Data**

- Existing measurement traces are unaffected by the Cal Set Viewer.
- The Cal Set data trace is presented in the highest unused channel number (usually 32) in the active window.
- The Cal Set data trace is labeled as S11 in the status bar regardless of the type of error term or standard.
- Only one Cal Set error term or standard data can be viewed at a time. However, a data trace can be stored into memory and then compared to other data traces.
How to access Cal Set Viewer

Using front-panel HARDKEY [softkey] buttons

1. Press **CAL**
2. then [Manage Cals]
3. then **[Cal Set Viewer]**

Using a mouse with PNA Menus

1. Click **Response**
2. then **Cal**
3. then **Manage Cals**
4. then **Cal Set Viewer**

How to use Cal Set Viewer

1. Use the down arrow to select a Cal Set. Then click either:
   
   - **Error Terms** - calculated data.
   - **Standards** - the raw measurement data of the Standard. **ONLY** available with Unguided Cal (not ECal or Guided Cal).

2. Use the down arrow to select an error term or standard to view.
3. Select the **Enable** check box to view the data on the PNA screen.

**Port numbering convention** for error terms is the same as for S-Parameters:

**E Term (Receiver, Source)** with the following exceptions:

- Load Match (2,1) - The match of port 2 which is measured by making an S11 measurement.
- Load Match (1,2) - The match of port 1 which is measured by making an S22 measurement.
- Transmission Tracking (2,1) - The port 2 receiver relative to the port 1 reference. (source=port 1).
- Transmission Tracking (1,2) - The port 1 receiver relative to the port 2 reference. (source=port 2).
- And so forth for multiport calibrations.

Last modified:
3-Sep-2008    Removed legacy content
Modify Calibration Kits

You can create or modify calibration kit files using Advanced Modify Cal Kits.

- About Modifying Calibration Kits
- Creating a New Cal Kit from an Existing Cal Kit
- Creating Custom Calibration Kits using a New Connector Family
- How to Modify Cal Kits
- Calibration Class Assignments
- Waveguide Cal Kits

Note: For a detailed discussion of Cal Kits and standards, read Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers (Application Note 1287-11)

About Modifying Calibration Kits

You can modify calibration kit files or create a custom one. You can also modify Data-based Cal Kits. Learn how.

For most applications, the default calibration kit models provide sufficient accuracy for your calibration. However, several situations exist that may require you to create a custom calibration kit:

- Using a connector interface different from those used in the predefined calibration kit models.
- Using standards (or combinations of standards) that are different from the predefined calibration kits. For example, using three offset SHORTs instead of an OPEN, SHORT, and LOAD to perform a 1-port calibration.
- Improving the accuracy of the models for predefined kits. When the model describes the actual performance of the standard, the calibration is more accurate. (Example: A 7 mm LOAD is determined to be 50.4Ω instead of 50.0Ω.)
- Modifying the THRU definition when performing a calibration for a non-insertable device.
- Performing a TRL calibration.

Creating a New Cal Kit from an Existing Cal Kit

You can create a new custom Cal Kit using a copy of an existing Cal Kit as a starting point. Here is how:

1. From the Edit PNA Cal Kits dialog, click Import Kit to load the Cal Kit you want to use as a starting point. A “Duplicate Name...” message appears. Click OK to load a duplicate copy of the Cal kit into the last position of the Edit PNA Cal Kits dialog.
2. Select the imported kit.

3. Click **Edit Kit**, then change the **Cal Kit Name and Description**.

4. Click **Installed Kits - Save As** to save the new Cal Kit to a .ckt file.

5. Recommended: Also click **Edit PNA Cal Kits - Save As** to save the entire collection of Cal Kits to a .wks file.

6. If using a new or modified connector, click **Change Family** to change the connector family.

7. Click **Add or Edit** to change connector descriptions and parameters.

8. Make modifications to your new custom Cal Kit as required. Save your work by clicking **Installed Kits - Save As**

**Note:** Custom Cal Kits must be imported after a firmware upgrade. [Learn more](#)

**Creating Custom Calibration Kits using a New Connector Family**

To create a custom calibration kit that uses a new connector type, you must first define the connector family. The connector family is the name of the connector-type of the calibration kit, such as:

- APC7
- 2.4 mm
- Type-N (50 Ω)

Although more than one connector family is allowed, it is best to limit each calibration kit to only one connector family.

If you are using a connector family that has male and female connectors, include definitions of both genders. If you are using a family with no gender, such as APC7, only one connector definition is required.

Use the following steps to create a custom calibration kit:

1. In the **Edit PNA Cal Kits dialog box**, click **Insert New** to add the new connector family.

2. In the **Edit Kit dialog box**:
   - Type the Kit Description for the custom cal kit.
   - Click **Add** in the Connectors section of the dialog box.

3. In the **Add Connector dialog box**:
   - Type a Connector Family name.
   - Type a Description of the connector.
   - Select the Gender of one of the connectors.
   - Type the minimum and maximum Frequency Range.
• Type the Impedance.
• Click the down-arrow to select the Media.
• Type the cut-off frequency.
• Click Apply.
• Click OK.

4. If you need to add another connector gender, in the Edit Kit dialog box:
• Click Add in the Connectors section again for the next connector gender.

5. If you are adding another connector gender, repeat step 3.

Note: If you have male and female versions of the connector family, you probably do NOT also have a NO GENDER version.

6. Now that the connector family is added to the custom cal kit, you are ready to add new calibration standards. In the Edit Kit dialog box:
• Under the list of standards, click Add.

7. In the Add Standard dialog box:
• Select the type of standard (OPEN, SHORT, LOAD, or THRU).
• Click OK.

8. In the Edit/Add Standards dialog box:
• Complete the information in the dialog box for the standard you selected. Note that for banded standards, the start and stop frequency may be different than the frequency range of the specified connector. Edit the start and stop frequencies as needed.
• Click OK when all the settings are correct.

9. Repeat steps 6 - 8, as necessary, to add all standards and definitions to the new custom cal kit.

10. Assign each of the standards to a calibration class. This is done through the Modify Calibration Class Assignment dialog box.

11. Click File, PrintToFile. PrintToFile will generate a .prn file (ascii file with comma delimiters) that can be imported into a spreadsheet.

12. Import the .prn file into an application such as Microsoft Excel, and print the results.

13. Use the spreadsheet to verify that each standard in the kit belongs to the same connector family and the
gender of each standard is properly specified. It is important that the connectors and genders for your standards are correctly defined and verified in order for your SmartCal (guided calibrations) to work properly.

**How to Modify Cal Kits**

The series of dialog boxes that follow allow you to modify the standard definitions or class assignments of calibration kit files.

### Using front-panel

**HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[More]**
3. then **[Cal Kit]**

### Using a mouse with PNA Menus

1. Click **Response**
2. then **Cal**
3. then **More**
4. then **Cal Kit**

**Edit PNA Cal Kits**

![Image of Edit PNA Cal Kits dialog box]

- **Open**
- **Save As**
- **Restore Defaults**
- **Installed Kits**

- **ID**
- **Kit Name**
- **Description**

- **Options**
  - **Edit Kit**
  - **Delete**
  - **Sort**
  - **Print to File**

- **Buttons**
  - **OK**
  - **Cancel**
  - **Help**
**Edit PNA Cal Kits** dialog box help

Provides access to all Agilent cal kits and allows modification of their standard definitions.

**PNA Cal Kits and Firmware Upgrades**

- The default "factory" cal kits are overwritten when new firmware is installed. Your custom cal kits (files with custom filenames) are NOT overwritten. However, the custom cal kits must be imported (click **Import Kit**) into the new firmware.

- All PNA cal kits can only be imported by the current firmware revision and later. They can NOT be imported by PAST firmware revisions. Once a Cal Kit has been imported by a later firmware revision, it cannot be imported by the previous version of firmware from which it originated.

- When a firmware upgrade takes place, ALL cal kits, both factory and custom, that are present on the PNA are saved to a single *.wks file using a unique filename. These files are NOT Excel spreadsheet files. They are opened using the **Open** button (see below). They can be used as archives of cal kits from previous firmware versions.

**Open** Opens an archive of cal kits from past firmware upgrades and 'Save As' operations.

**Save As** Saves ALL cal kits in the PNA to a *.wks file.

**Restore Defaults** Re-installs the default factory contents of all Agilent cal kits from the PNA hard drive. The factory Agilent cal kits are stored on the PNA hard drive at C:/Program Files/Agilent/Network Analyzer/PnaCalKits/factory.

**Installed Kits**

- **Import Kit** Invokes the **Import Kit** dialog box.

- **Save As** Saves the selected calibration kit and definitions (using .ckt file type).

- **Insert New** Invokes a blank **Edit Kit dialog box** to create new calibration kit definitions.

- **Print to File** Prints the contents of the selected cal kit to a .prn file.

- **Edit Kit...** Invokes the **Edit Kit dialog box** to modify selected calibration kit definitions. [Learn how to modify Data-based Cal Kits](#).

- **Delete** Deletes selected calibration kit file.

- ^ Selects previous / next calibration kit in list.

For more information see [Creating Custom Calibration Kits using a New Connector Family](#).
Imports calibration kit definitions from hard disk or other drive that are saved in the various formats. With PNA version 4.0 or later, four kit types can be imported.

**Note: See PNA Cal Kits and Firmware Upgrades**

Files of type  Select the file type of your Cal Kit

<table>
<thead>
<tr>
<th>Cal Kit Format</th>
<th>File Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current PNA Series Cal Kit</td>
<td>*.ckt</td>
</tr>
<tr>
<td>Old PNA Series Cal Kit (Version 1)</td>
<td>*.ck1</td>
</tr>
<tr>
<td>8510 Cal Kit</td>
<td>CK_*</td>
</tr>
<tr>
<td>8753, 8752, 8719, 8720, or 8722 Cal Kit</td>
<td>*.ck</td>
</tr>
</tbody>
</table>

File name  Navigate and select your cal kit file.

Open  Imports the selected file. The kit is added at the end of the list of cal kits.

**Importing Kits other than current PNA Series Kits**

Cal kit files from Agilent “legacy” network analyzers (listed above) may not contain information that the PNA requires. Therefore, the PNA may modify the cal kit name and description, the cal standards, and the cal class assignments in a best effort manner. You may need to correct these modifications after importing your legacy cal kit to meet your specific requirements.

- “Legacy” cal kit files are based on the analyzer test port sex; PNA cal kits are based on the Device Under Test (DUT) connector sex. Therefore, when the kit is imported the standard’s label and description are reversed and are noted as F- (female) and M- (male).
- When a Coaxial standard is detected in the kit file, a pair of male/female connectors is typically created.
- Waveguide standards that are created as connector have no gender.

**Edit Kit** dialog box help

**Identification**
- **Kit Number** Number of the selected calibration kit.
- **Kit Name** Allows you to change the Name of the selected calibration kit.
- **Kit Description** Allows you to change the description of the selected calibration kit.

**Connectors**

**Note:** You can NOT use a connector with a new or modified name to perform an ECal User Characterization.

The following is the list of Factory-defined connector type strings:
<table>
<thead>
<tr>
<th>Connector Type</th>
<th>Female Description</th>
<th>Male Description</th>
<th>Waveguide Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>APC 3.5</td>
<td>Type N (50) female</td>
<td>7-16 female</td>
<td>X-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type N (50) male</td>
<td>7-16 male</td>
<td>P-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type N (75) female</td>
<td>2.92 mm female</td>
<td>K-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type N (75) male</td>
<td>2.92 mm male</td>
<td>Q-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type F (75) female</td>
<td>1.85 mm female</td>
<td>R-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type F (75) male</td>
<td>1.85 mm male</td>
<td>U-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type A (50) female</td>
<td>1.0 mm female</td>
<td>V-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type A (50) male</td>
<td>1.0 mm male</td>
<td>W-band waveguide</td>
</tr>
<tr>
<td></td>
<td>Type B</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Click the down arrow to change the connector type.

**Add or Edit** Invokes the Add or Edit Connector dialog box which allows you to add new connector type to the calibration kit or edit the connector properties.

**Change Family** Invokes the Change Connector Family dialog box which allows you to rename the entire connector family name.

**Class Assignments**

Click the down arrow to change the Class Assignment.

**Edit** Invokes the Modify Calibration Class Assignments dialog box.

**Standards in Kit**

Lists the current standards and descriptions in the cal kit.

**Add...** Invokes the Add Standard dialog box that allows you to add definitions for a standard.

**Edit...** Invokes the Edit dialog box that allows you to modify standard definitions for the selected standard: either Open, Short, Load, or Thru.

**Delete** Deletes selected standard from calibration kit.
Add or Edit Connector dialog box help

**Identification**

**Note:** You can NOT use a connector with a new or modified name to perform an ECal User Characterization.

**Connector Family** Allows you to Add or Edit a specific connector name. If you change Connector Family to a unique name, the name and selected Gender is ADDED to the list of connectors in that kit.

**Note:** To change the Connector Family Name of all connectors in the Kit, click Change Family on the previous dialog box.

**Description** Displays connector type and gender.

**Frequency Range**

- **Min** Allows you to define the lowest frequency at which the standard is used for calibration.
- **Max** Allows you to define the highest frequency at which the standard is used for calibration.

**Gender**

Allows you to define the connector gender.

**Impedance**

Allows you to define the impedance of the standard. During a TRL Guided Cal, this value is also used as the System Z0 reference impedance. Learn more.

**Media**

Allows you to define the medium (or ‘geometry’) of the connector: COAX or WAVEGUIDE.

**Waveguide Cal Kits**

If modifying or creating a waveguide cal kit, be sure to make the following settings. You can create a custom waveguide cal kit using an existing factory waveguide Cal kit as a starting point. The factory cal kits already have these settings.
• Frequency Range: Min. frequency = Cutoff frequency.

• Gender: No Gender

• Impedance Z0: 1 ohm

• Media: Waveguide

• Cutoff Frequency enter the low-end cutoff frequency.

• Height/Width Ratio Used to calculate waveguide loss. This value is usually on the data sheet for waveguide devices. For more information see App Note 1287-11.

Other waveguide settings

• If performing an Unguided Cal, change System Impedance to 1 ohm.

• For waveguide, choose TRL (Thru-Reflect-Line) calibration type. These calibration types are more accurate and take fewer steps than SOLT.

---

**Change Connector Family**

<table>
<thead>
<tr>
<th>Previous Connector Family</th>
<th>APC 3.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specify New Connector Family</td>
<td>APC 3.5</td>
</tr>
</tbody>
</table>

**Change Connector Family** dialog box help

**Note:** You can NOT use a connector with a new or modified name to perform an ECal User Characterization.

Performs a text “Search and Replace” function. Within the description field of each of the standards of the current Cal Kit, it searches for the Previous Connector Name and replaces it with the New Connector Name.

**Specify New Connector Name** Allows you to replace the primary connector-family name from the selected kit with the new connector-family name. The PNA allows multiple connector-families per kit.

**Previous Connector Name** Displays the primary connector-family name. All occurrences of the previous connector name will be replaced throughout calibration dialog boxes. This includes calibration kit labels and description fields.

**Notes:**

• String replacement requires an exact match and is case sensitive. For example, "Type N" does not match "type N", and "apc 7" does not match "APC 7".

• Some calibration kits may include connector names that do not match strings within labels or description fields. You may reuse the Change Connector Name dialog to standardize the name within the kit, and then to replace the standard name with the new name.

**Example:**
Select the 85056A calibration kit. The default connector-family name is "APC 2.4". However, many
standard description files are labeled "2.4 mm". You may want to replace the connector family name with a new name and update the standard descriptions to match the new name. For this kit, use a two step procedure.

1. Use the Change Connector Name dialog to replace "APC 2.4" with "2.4 mm".
2. Use the Change Connector Name dialog to replace "2.4 mm" with the new name, "PSC 2.4 mm".

See Also Creating a New Cal Kit from an existing Cal Kit

---

### Modify Calibration Class Assignments dialog box help

Allows you to assign single or multiple standards to Calibration Classes.

There are two ways to get here:

1. Click **Calibration**
2. Click **Advanced Modify Cal Kit..**
3. Select the Cal Kit, then click **Edit Kit**
4. Under Class Assignments, select the Cal Method (SOLT, TRL), then click **Edit**

You can also get here during a SmartCal Calibration.

1. From the **Select DUT Connectors and Cal Kits dialog**, check **Modify Cal**, then click **Next**.
2. At the Modify Cal dialog, click a **Mod Stds** button.

3. At the View/Modify Properties Dialog, select the Cal Method (SOLT, TRL), then click **View/Modify**

To assign a standard to a calibration class:

1. Select the **Calibration Kit Class**
2. Select the standard from the **Unselected Standards** field
3. Click the right arrow to move the standard to the **Selected Standards** field.

**Notes:**

- During an Unguided Cal all of the **Selected Standards** are presented. You then choose which of these standards to measure.

- The MATCH standards must be assigned to the FWD MATCH, REV MATCH, and LINE classes. See [TRL calibrations](#) to learn more about TRL standards.

- Use MOVE UP and MOVE DOWN to change the ORDER of the standard. The order is used during a **SmartCal** to determine overlap priorities when:

  - **Multiple standards are valid for a frequency** - standards are presented in the order in which they appear.

  - **Using two sets of standards** - modify the order in which standards appear to reflect the configuration of your DUT. For example, for a DUT with a male connector on port 1 and a female connector on port 2, order the devices within the S11 classes (A, B, and C) such that the MALE standards are first in the list. Then order the S22 classes specifying the FEMALE standards as the first in the list.

To Add or Edit standards, click Calibration then, click **Advanced Modify Cal Kit**.

- See [TRL Class Assignments](#)

- [Learn more about Calibration Classes.](#)

**Calibration Class Label**

The label that appears on the Unguided Cal - Measure Mechanical Standards dialog box. For example, the Calibration Class Label "**Modified OPEN**" would yield the following prompt:

![Diagram of a calibration kit with labels: Short, Loads, Thru, Open, Short, Loads, with Modified OPEN highlighted.](Diagram)
The following selections in this dialog box depend on your Class Assignment selection (SOLT or TRL) in the Edit Kit dialog box.

**SOLT ONLY**

**Link FWD TRANS, FWD MATCH, REV TRANS, and REV MATCH** Check to automatically assign the standard definition for FWD TRANS to FWD MATCH, REV MATCH, and REV TRANS. Clear to separately assign FWD MATCH, REV MATCH and REV TRANS classes (SOLT calibrations only).

**Expanded Calibration**

The following two check boxes apply ONLY during Guided Calibrations. For Unguided Calibration, these check boxes are ignored, including the case where the multiple standards dialog box is presented.

- **Measure all mateable standards in class** Check this box to attain the very highest accuracy possible. For example, if a cal kit contains several load standards, during the calibration process you will be prompted to measure each of the standards. This could require a significant amount of calibration time. When checked, the "Use expanded math when possible" box is also checked automatically.

- **Use expanded math when possible** Some kits contain multiple calibration standards of the same type that together cover a very wide frequency range. (For example: multiple shorts, or a lowband load and a sliding load.) If a calibration requires more than one standard to cover the calibration frequency range, there can be regions of overlapping measurements. When this checkbox is selected, the PNA automatically computes the most accurate measurement in the overlap regions using a "weighted least squares fit" algorithm. This function improves accuracy without slowing the calibration speed.

  - Manually select this checkbox only when using a cal kit that contains multiple standards of the same type. (For example: multiple shorts, or a lowband load and a sliding load.)
  
  - The checkbox is cleared by default when a polynomial model is selected from the cal kit menu.
  
  - The checkbox is selected by default when the 85058B or 85058E data-based model is selected from the cal kit menu.

**TRL ONLY**

If TRL is selected as Class Assignment in the Edit Kit dialog box, the following changes appear in this dialog:

![Calibration Kit Class](image)

**Calibration Kit Class**

- Learn more about TRL standards.

- Isolation calibration is not usually necessary in the PNA.

**LRL line auto characterization**
**Note:** This setting ONLY applies if an LRL Cal Kit is being modified AND Testport Reference Plane is set to Thru Standard AND the TRL Thru class standard and the TRL Line/Match class standard both have the same values for Offset Z0 and Loss. Otherwise, this setting is ignored.

- Check the box to allow the PNA to automatically correct for line loss and dispersion characteristics.
- Clear the box if anomalies appear during a calibrated measurement which may indicate different loss and impedance values for the Line standards.

**Calibration Reference Z0 (TRL only)**

**System Z0**  The system impedance is used as the reference impedance. During an Unguided Cal, the value that is set at **System => Configure => System Z0** is used. During a Guided Cal, the Z0 of the Cal standard's connector definition sets the System Z0. See where this value is set.

Make this selection when the desired test port impedance differs from the impedance of the LINE standard. Also, make this selection when skin effect impedance correction is desired for coax lines.

**Line Z0**  The impedance of the line standard is used as the reference impedance, or center of the Smith Chart. Any reflection from the line standard is assumed to be part of the directivity error.

**Testport Reference Plane (TRL only)**

**Thru Standard**  The THRU standard definition is used to establish the measurement reference plane. Select if the THRU standard is zero-length or very short.

**Reflect Standard**  The REFLECT standard definition is used to establish the position of the measurement reference plane. Select if the THRU standard is not appropriate AND the delay of the REFLECT standard is well defined.

Also, select if a flush short is used for the REFLECT standard because a flush short provides a more accurate phase reference than a Thru standard.
**Add Standard** dialog box help

Allows you to add standards to the calibration kit file.

**OPEN** Adds an open to the calibration kit file.

**SHORT** Adds a short to the calibration kit file.

**LOAD** Adds a load to the calibration kit file.

**THRU** Adds a thru to the calibration kit file.

**DATA BASED STANDARD** Adds a data-based standard to the calibration kit file.

**OK** Invokes a blank Edit Standards: Open, Short, Load, Thru, or Data-Based dialog box.

For more information see Creating Custom Calibration Kits using a New Connector Family.

---

**Edit / Add Standards (Open, Short, Load, Thru, or Data-based)**

![Diagram of Edit/Add Standards dialog box](image)

**Edit / Add Standards** dialog box help

The boxed areas of the previous graphic applies to all standard types.

The other areas change depending on the type of standard selected.

**Identification**

- **Standard ID** Number in list of standards
- **Label** Type of standard.
- **Description** Description of standard.

**Frequency Range**

- **Min** Defines the lowest frequency at which the standard is used for calibration.
Max  Defines the highest frequency at which the standard is used for calibration.

Connector
Indicates the type and gender (Male, Female, None) of the standard.

Delay Characteristics
Delay  Defines the one-way travel time from the calibration plane to the standard in seconds.
Z0  Defines the impedance of the standard.
Loss  Defines energy loss in Gohms, due to skin effect, along a one-way length of coaxial cable.

The following applies to standard types Open, Short, Load, Thru, and Data-based

Open Standard

<table>
<thead>
<tr>
<th>Open Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>C0</td>
</tr>
<tr>
<td>C1</td>
</tr>
</tbody>
</table>

C0, C1, C2, C3  Specifies the fringing capacitance.

Short Standard

<table>
<thead>
<tr>
<th>Short Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>L0</td>
</tr>
<tr>
<td>L1</td>
</tr>
</tbody>
</table>

L0, L1, L2, L3  Specifies the residual inductance.

Load Standard

<table>
<thead>
<tr>
<th>Load Type</th>
<th>Arbitrary Impedance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed Load</td>
<td>Offset Load</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Complex Impedance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Real 50</td>
</tr>
<tr>
<td>Imag 0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Delay Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delay 0 pSec</td>
</tr>
<tr>
<td>Z0 50 ohms</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Offset Load Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>First Offset Standard: THRU</td>
</tr>
<tr>
<td>Second Offset Standard: 7-32 Line</td>
</tr>
<tr>
<td>Load Standard: ADPTR/LOAD -M-</td>
</tr>
</tbody>
</table>

Allows you to select the type of load.
Load Type

**Fixed Load**  Specifies the load type as Fixed. The fixed load is assumed to be a perfect termination without reflection.

**Sliding Load**  A sliding load is defined by making multiple measurements of the device with the sliding load element positioned at various marked positions of a long transmission line. The transmission line is assumed to have zero reflections and the load element has a finite reflection that can be mathematically removed using a least squares circle fitting method.

A sliding load cal can be very accurate when performed perfectly. It can also be very inaccurate when not using proper technique. **For accurate results, closely follow the users manual instructions for the sliding load.**

**Arbitrary Impedance**  Specifies the load type to be have an impedance value different from system Z0. An arbitrary impedance device is similar to a fixed load except that the load impedance is NOT perfect. Early firmware releases of the PNA series used a fixed resistance value. A complex terminating impedance has been added to allow for more accurate modeling of circuit board or on-wafer devices.

The following Complex Impedance settings are available ONLY when Arbitrary Impedance is selected.

- **Real**  The real portion of the impedance value.
- **Imaginary**  The imaginary portion of the impedance value.

**Offset Load**  In Jan 2006, Offset Load definitions were added to TRL and Waveguide Cal Kit files. Using an Offset Load standard results in a more accurate calibration than with a Broadband Load. Therefore, when performing a calibration using one of the modified Cal Kit definitions, you may be prompted to connect more standards than before this change.

**Note:** The Offset Load standard can be used ONLY during a **SMART (Guided) calibration**.

To revert to using the Broadband Load Standard without offset, do the following:

1. Click **Calibration**, then **Advanced Modify Cal Kit**
2. Select the kit, then click **Edit Kit**
3. Under Class Assignments, click **Edit**
4. Select Calibration Kit Class **S11C** (Loads)
5. Under Selected Standards, select **Broadband Load**, then click **Move Up** until the standard is at the top of the list. This will ensure that the Broadband Load is used first.

**About Offset Load**

An offset load is a compound standard consisting of a load element and two known offset elements (transmission lines) of different length. The shorter offset element can be a zero-length (Flush-thru) offset. The load element is defined as a 1-port reflection standard. An offset load standard is used when the response of the offset elements are more precisely known than the response of the load element. This is the case with waveguide. Measurement of an offset load standard consists of two measurements, one with each of the two offset elements terminated by the load element. The frequency range of the offset load standard should be set so that there will be at least a 20 degree separation between the expected response of each measurement.
To specify more than two offset elements, define multiple offset load standards. In cases where more than two offsets are used, the frequency range may be extended as the internal algorithm at each frequency will search through all of the possible combinations of offsets to find the pair with the widest expected separation to use in determining the actual response of the load element.

The following Offset Load settings are available ONLY when Offset Load is selected.

- First Offset Standard
- Second Offset Standard
- Load Standard

### Thru Standard

**Connectors**

Defines connector type and gender at both ports.

### Data-Based Standard

**Connectors**

Defines connector type and gender at both ports.

**Upload Data From File**

Click Browse to load data from a file.

### Connectors

**Uploader**

Click Browse to load data from a file.

**Connectors**

**Note:** To learn how to modify data-based standard files, visit [http://na.tm.agilent.com/pna/dbcal.html](http://na.tm.agilent.com/pna/dbcal.html)

The modified file can then be uploaded into the PNA.
One Port Standard  Currently only 1-port standards can be modified.

Port 1  Select the type of connector.

File Information  Information about the standard that is read from the uploaded file.

Last modified:

- 11-Nov-2010  Changed edit kit diag image
- 23-Sep-2010  Added clarification for TRL reference System Z0
- 3-Sep-2008   Removed legacy content
- 14-Apr-2008  Add offset load note
- 4-Jan-2008   Added limit for imported kits
- 26-Oct-2007  Added Height/Width for Add connector.
                Moved waveguide settings.
- 2-Feb-2007   MX Added UI
Power Calibration

Source and Receiver Power Calibrations work together to provide very accurate power levels from the source, and very accurate power measurements from the PNA receivers.

- **Source Power Calibration Overview**
- **Supported Power Meters and Sensors**
- **How to perform Source Power Calibration**
- **Setup**
- **Source Power Cal dialog**
  - **Source Power Calibration Options dialog**
  - **Power Meter Settings dialog**
  - **Power Loss Compensation dialog**
  - **Power Sensor Settings dialog**
- **Copy a Source Power Calibration to other Channels**
- **Saving a Source Power Calibration**
- **Reducing Time to Complete a Source Power Calibration**
- **Receiver Power Calibration**
- **Saving Receiver Cals**

Other Source Power Cal choices

- **Guided Power Cal** can be performed during an S-parameter Guided Calibration. [Learn more](#).
- **Receiver Leveling** can be used to provide ‘real-time’ source power cal. [Learn more](#).
- **See Also:** [Configure an Power Meter As a Receiver (PMAR)](#)

See other Calibration Topics

Source Power Calibration Overview

Perform Source Power Calibration when you need accurate power levels at some point in the measurement path between the PNA test ports. For example, you need to characterize the gain of an amplifier across a frequency range at a specified input power. You would perform a source power cal at the input of the amplifier to ensure the exact power level into the amplifier across the frequency range.
Using a Source Power Cal, you can expect the power at the point of calibration to be within the range of the uncertainty of the power meter and sensor that is used.

**Source Power Calibration...**

- Is independent of measurement type. It corrects the PNA source regardless of which receivers are being used in a measurement. Therefore, it can be used with both ratio or non-ratio measurements.

- Applies ONLY to those measurements on the selected channel that use the test port that was specified as the Source for the calibration. For example, if you specify Channel 1 and Port 1 as the source to be calibrated, only those measurements on channel 1 that use port 1 as the source will be corrected.

- Can be used in conjunction with other measurement calibrations, such as a full 2-port calibration. For highest accuracy, perform the measurement calibration AFTER the source calibration.

- Can be used with Power Sweep type. Source Power Cal will correct the power at all power levels across the power sweep.

- Can be used with Port Power Uncoupled.

- Forces sweep mode to Stepped on measurements with source power correction turned ON.

- Beginning with PNA Rev. 7.50, an external source can be calibrated using Source Power Cal.

**Overview of How it works:**

See Important First-time USB connection note.

Click to see the detailed procedure

1. Specify the measurement settings (frequency range, IFBW and so forth).


   **Note:** When using an Agilent 848X power sensor (sensors that do NOT have built-in calibration factors), enter the Cal Factors using the Power Sensor Settings dialog, because the PNA instructs the power meter to NOT use the Cal Factor tables internal to the power meter.

3. Connect a power meter sensor to the point at which you want a known power level. This may be at the input or output of your device, or some other point between the test ports.

4. The PNA source is stepped through the specified frequency range, and power is measured with the power meter. At each data point, the source power is adjusted until the measured power is within your specified accuracy level.

5. When complete, the power meter is preset. The source power calibration can be saved as part of the instrument state.

6. The power meter is removed and the measurement path reconnected.

7. The calibration is automatically applied to the channel. All measurements on that channel using that source port benefit from the source power cal.
8. Perform an S-parameter calibration AFTER a Source Power Cal. The S-parameter cal is performed using the corrected stimulus power levels for the relevant ports.

**Verify** the source power calibration using the following procedure.

1. Connect the power meter as it was during the source power calibration.
2. Set the PNA to **Point Trigger** mode.
3. Trigger the PNA across the trace. Read about the behavior of the **sweep indicator**.
4. At each data point, the power meter should read the corrected power level within the specified tolerance.

**Supported Power Meters and Sensors**

See Agilent's Power Meters and Sensors Webpage.

**Power Meters**

The following Agilent Power Meters are supported:

- HP 437B / 438A power meters
- E-Series power meters (E4418 and E4419) and all supported sensors.
- P-Series power meters (N1911A and N1912A) and all supported sensors.
- EPM Series power meters (E4416A and E4417A) and all supported sensors.
- N1913A and N1914A power meters and all supported sensors.
- U2000 Series USB power sensors. See **USB Power Sensors** (below)

**Notes**

- N1911A, 12A, 13A, and 14A power meters have a ‘**device-side**’ USB connector and are controlled by the PNA exactly like a USB sensor. See **USB Power Sensors** (below). Although these meters may also have a front-panel USB port, USB power sensors must be connected directly to one of the PNA USB ports.
- The **82357A USB/GPIB Interface** can be used to control power meters.
- **Create a Custom Power Meter Driver** for use with other power meters.

**Power Sensors**

You can perform a Source Power Calibration with ALL power sensors that are supported by the above power meters. However, Source Power Calibration operates slowly with the Agilent E930x and E932x power sensors. Multiple power sensors can be used to cover the frequency span of the measurement. **Learn how.**
**USB power sensors** are supported beginning with PNA Rev 7.50.

- Only one USB power sensor can be used to cover the entire frequency span.

- To select a USB power sensor:
  1. Connect the sensor directly to one of the PNA USB ports.
  2. From the **Source Power Cal** dialog, click **Power Meter Config**.
  3. On the **Power Meter Settings** dialog, select **USB**.

- Or select a USB power sensor that is configured as a PMAR Device. [Learn how]

See Important First-time USB connection note.
See note about Zeroing USB Power Sensors.
See also: Power Meters as Receivers (PMAR)
3. Apply power to the power meter and allow 30 minutes warm-up time before beginning calibration.

4. Select Source Power Cal as follows:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press CAL</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [Power Cal]</td>
<td>2. then Cal</td>
</tr>
<tr>
<td>3. then [Source Cal]</td>
<td>3. then Power Cal</td>
</tr>
<tr>
<td></td>
<td>4. then Source Cal</td>
</tr>
</tbody>
</table>

5. Complete the Source Power Cal dialog box (below), including Options, Loss Compensation and Power Sensor Settings, as needed.

   **Note:** When using an Agilent 848X power sensor (sensors that do NOT have built-in calibration factors), enter the Cal Factors using the Power Sensor Settings dialog, because the PNA instructs the power meter to NOT use the Cal Factor tables internal to the power meter.

6. When complete, click Take a Cal Sweep in the Source Power Cal dialog box.

7. Follow the prompts to connect the sensors as required.

8. At this time you can change the Source Port setting and perform a Source Power Cal on a different port.

9. When calibration is finished, click OK. Correction is then applied and turned ON for the relevant ports on the active channel.

10. Remove sensor.

11. SrcPwrCal is displayed in the status bar when Source Power Correction is applied to the Active Measurement.

12. Perform a S-parameter calibration, which would use the corrected stimulus power levels for the relevant ports.

---

**To turn Source Power Correction OFF:**

- On the Calibration menu, point to Power Calibration, then click Source Power Correction on/OFF.

- ONLY correction for the source port of the ACTIVE MEASUREMENT is turned OFF (regardless of port power coupling setting.)
Interpolation or Extrapolation

If the original stimulus settings are changed, Interpolation or EXTRAPOLATION is applied and SrcPwrCal* is displayed in the status bar. This is different from measurement calibration interpolation. For example, if the frequency span is increased, the PNA will extrapolate new correction values rather than turn correction off. This is to protect your test device from being overpowered by the source. If the original settings are restored, then source power calibration returns to full correction.

Source Power Cal dialog box help

![Source Power Calibration dialog box](image)

**Note:** Be sure that the frequency range of your power sensor covers the frequency range of your measurement. This does NOT occur automatically.

**Power**

- **Cal Power** The calculated power (in dBm) at the calibration point. This value is the specified PNA source power plus the Power Offset value.

- **Power Offset** Allows you to specify a gain or loss (in dB) to account for components you connect between the source and the reference plane of your measurement. These components will remain during a measurement. For example, specify 10 dB to account for a 10 dB amplifier in the path to your DUT. Following the calibration, the PNA power readouts are adjusted to this value.

To account for components that will be removed when the calibration is complete, use the Loss Compensation table.

**Channel and Port Selection**

- **Channel** Specifies the channel on which to perform the calibration. This setting defaults to the active channel.

- **Source Port** Specifies the source port to be corrected. This setting defaults to the source port for the active measurement.

Beginning with PNA Rev. 7.22, external sources can be calibrated using his dialog. Learn more.
### Accuracy
At each data point, power is measured using the specified Power Meter Settling Tolerance and adjusted, until the reading is within this Accuracy Tolerance or the Max Number of Readings has been met. The last power reading is plotted on the screen against the Tolerance limit lines.

- **Tolerance**: Sets the maximum desired deviation from the specified Cal Power level in 0.005 dB increments from 0 to 5 dB.
- **Max Number of Readings**: Sets the maximum number of readings to take at each data point for iterating the source power.

### Calibration Status
Allows you to turn Source Power Cal ON | OFF and view Cal data for each port, regardless of the active measurement. This feature allows the Internal Second Source to be calibrated and turned ON | OFF, even when being used as an incidental source in a measurement, such as an LO.

- **Calibration ON**: Check to turn Source Power Calibration ON for the specified source port.
  
The displayed text indicates when interpolation is applied for the calibration.

### Buttons
- **Options**: Invokes the Source Power Cal Options dialog. Label to the left of the button displays the current 'Options' setting.
- **Power Meter Config**: Invokes the Power Meter Settings dialog box
- **Take Cal Sweep**: Begins source power calibration measurement.
- **OK**: Applies calibration. This button is disabled until the Take Cal Sweep has been pressed.
- **Cancel**: If a sweep is in progress, cancels the sweep. Press again to close the dialog.

### Pass / Fail Limits
Limit lines are drawn on the Source Power Cal measurement graticule area. These lines are at the Cal Power +/- the current setting of Accuracy Tolerance. A FAIL during the Source Power Cal sweep means that the PNA was unable to measure power to within the Accuracy Tolerance. Tight tolerances are more difficult to achieve at lower Cal Power levels. When a FAIL indication appears, increase the Max Number of Readings. If this does not cause a PASS condition, then decrease the Accuracy Tolerance value.

### See Also
- Learn more about **Source Power Cal**
- Learn about **External Testsets** and Source Power Cal.
Provides options for measurement of the source power.

**Note:** At low power levels (less than -30 dBm) most power meters are not as accurate as a PNA receiver.

**Calibration Reference** Choose power meter/PNA receiver to use to measure power.

- **Use a power meter.** Traditional source power calibration using only a power meter to measure the source power at each data point. Most accurate (at higher power levels) and slowest method.

  **Note:** Because the following two settings use PNA receivers to make power measurements, they do NOT work correctly when a Frequency Offset value is being used.

- **Use a power meter once, then use the PNA reference receiver.** When checked, the first reading at each data point uses a power meter to calibrate the reference receiver. Subsequent readings, if necessary to meet your accuracy requirement, are measured using the reference receiver. This technique is much faster than using the power meter, and more accurate when measuring low power levels.

  **Note:** Do NOT use this setting if there is a component before the power sensor that exhibits non-linear behavior, such as a power amplifier in compression. Use a power meter, and **Calibrate the source at multiple power levels**.

- **Use a receiver.** Select a PNA Receiver or a PMAR (Power Meter as Receiver).

  **PNA receiver** - For highest accuracy, first calibrate the receiver by performing a source power cal using a power meter, then a **receiver cal**. That receiver can then be used to quickly calibrate other PNA source ports, or used on another channel with different stimulus settings. This would be useful, for example, if the power level of the measurement was below the sensitivity of the power sensor. Calibrate the PNA receiver using a source power cal that is within the sensitivity of the sensor. Then, use the calibrated receiver to perform a second source power cal at the reduced power level.
The PNA receiver is specified using either standard receiver notation or logical receiver notation.

It is best to use the reference receiver for the source port to be calibrated. For example, if calibrating source port 2, specify the "B" or "a2" receiver.

To ensure an accurate source power cal, the frequency range over which the receiver was calibrated must be the same or larger than the "receiver only" source power calibration.

All accuracy and settling tolerance and number of reading settings apply just as they do with a power meter reading.

**PMAR Device** - The power meter/sensor must first be configured. [Learn how to Configure a PMAR device.](#)

- **Use a power table and the PNA reference receiver** Used to provide power leveling with mmWave test set and modules. [Learn more.](#)

**Calibrate the source at multiple power levels** Used primarily with mmWave measurements.

This feature can also be used with standard PNA measurements when a component is used in the source path such as a booster amp which does not have NOT linear gain or loss over frequency. If this is not true for your setup but want to improve your source power accuracy, consider using the [Receiver Leveling](#) feature.

When checked, source power is measured using the specified 'Cal Reference' device (power meter/sensor or PNA receiver) and iterated on a sweep-to-sweep basis to construct a 2-dimensional power table: Power IN, Power OUT, over all frequencies.

- Click **Power Levels** to launch the [Source Cal Power Levels dialog box](#) to set the power levels at which source power is to be measured.

  - The source power cal is saved, but the power table is NOT accessible.

**Note:** If your measurement requires more or less source power than you specified in the Power Levels dialog, then the minimum or maximum source power correction will be used. The correction values are NOT extrapolated. This WILL result in source power inaccuracy.

**Calibrate the PNA reference receiver** Check to calibrate the appropriate reference receiver to the power level that is measured at the calibration plane. Do this to make very accurate measurements using the calibrated reference receiver. This cal is done in addition to the standard source power cal using the any of the methods listed above. At the end of the source power cal measurement sweep, you can optionally save the reference receiver cal to a Cal Set to be recalled at a later time. The Cal is saved when the **OK** button is clicked to close the Source Power Cal dialog.
Source Cal Power Levels dialog box help

This dialog appears when you click **Power Levels** on the [Source Power Cal Options dialog](#).

Specify the power levels at which the Source Power will be calibrated.

**Note:** If your measurement requires more or less source power than specified here, then the minimum or maximum source power correction will be used. The correction values are NOT extrapolated. This WILL result in source power inaccuracy.

- **Max Power** - The highest power level at which to calibrate.
- **Min Power** - The lowest power level at which to calibrate.
- **Power Step** - Calibrate at every incremental power level, between the Max and Min Power settings.

Power Meter Settings dialog box help

This dialog appears when you click the **Power Meter Config** button on many dialog boxes.
Communication

- **GPIB / Address** Select GPIB power meter. Then select the address for the power meter. Default is 13. The PNA will search VISA interfaces that are configured in the Agilent IO Libraries on the PNA.

- **USB** PNA scans the USB for connected power sensors. Select a power sensor from the list. Only ONE USB power sensor can be configured to cover the entire frequency range of the calibration. See USB sensor accuracy note.

- **LAN** Specify the Hostname or IP address of the Power Meter.

- **Any** For future use.

**Sensors** Invokes the power sensor settings dialog box.

**Settling**

These Settling settings do not apply when a PNA receiver is the power measurement device. Each power meter reading is "settled" when either:

- two consecutive meter readings are within this **Tolerance** value or

- when the **Max Number of Readings** has been met.

The readings that were taken are averaged together to become the "settled" reading. The settled reading is then compared to the **Accuracy Tolerance requirements** (tolerance and max readings) specified on the Source Power Cal dialog box.

**Tolerance** When consecutive power meter readings are within this value of each other, then the reading is considered settled.

**Max Number of Readings** Sets the maximum number of readings the power meter will take to achieve settling.

**Sensor Loss Compensation**

- **Use Loss Table** Select this checkbox to apply loss data to Source Power calibration correction (such as for an adapter on the power sensor).

- **Edit Table** Invokes the Power Loss Compensation dialog box.
To Add a Row to the table, click on a row in the table and press the down arrow on either the PNA front panel or keyboard.

To Edit a value, double-click in the cell to be edited.

Beginning with A.08.33, these values can be loaded from an S2P file using the Characterize Adaptor Macro. Compensates for losses that occur when using an adapter or coupler to connect the power sensor to the measurement port. These components will be removed when the calibration is complete. To account for components that will remain during the measurement, use the Power Offset setting.

The Frequency / Loss pairs define the amount of loss for the entire frequency range. For example, using the entries in the above dialog image:

- 0.5 dB is used to compensate power sensor measurements up to 1 GHz.
- Each data point between 1 GHz to 2 GHz is linearly interpolated between 0.5 dB and 1 dB.
- 1 dB is used above 2 GHz.
- A single frequency/loss segment is applied to the entire frequency range.

Enter up to 100 segments to achieve greater accuracy.

**Frequency** Enter a frequency in Hz.

**Loss** Enter a loss as a POSITIVE value in dB. To compensate for gain, use NEGATIVE values.

**Delete Table Segment** Deletes row indicated in the field.

**Delete All** Deletes all data in the table.

The Power Loss Compensation table survives PNA Preset and Power OFF. To NOT use Loss compensation, clear the Use Loss table checkbox on the Power Meter Settings dialog.
This dialog appears when you click the Sensors button on the Power Meter Settings dialog.

**Note:** Be sure that the frequency range of your power sensor covers the frequency range of your measurement. This does NOT occur automatically.

**Sensor A (B)** Displays one of the following messages depending on type of sensor.

- **Not connected** The PNA is not detecting a power sensor.

- **Cal factors are contained within this sensor** Internal Reference Cal Factor and Cal Factor data are loaded automatically.

- **Sensor Data** Allows the following entries for power sensor data:
  
  - **Reference Cal Factor** Specifies the sensor's Reference Cal Factor.
  
  - **Cal Factor Table** Specifies the frequency and corresponding Cal Factor for the sensor.
  
  - **Delete Cal Factor** Deletes the indicated row in the table.
  
  - **Delete All** Deletes all data in the table.
  
  - **To Add a Row** to the table, click on a row in the table and press the down arrow on either the PNA front panel or keyboard. A row is added to the bottom of the table. The table is automatically sorted by frequency when OK is pressed.

**Use this sensor only** Check this box to use this sensor over the entire frequency span of the measurement, even if two sensors are connected to power meter.

Clear this box to allow entry of minimum and maximum frequencies for the sensor. Only ONE of the two sensors can have this box checked. You will be prompted to connect the appropriate sensor during the power calibration.

**Minimum Frequency** Specifies the minimum frequency range for the sensor when using dual sensors.

**Maximum Frequency** Specifies the maximum frequency range for the sensor when using dual sensors.
Perform Sensor Zeroing and Calibration  Zero and/or calibrate the power sensor before measuring data. Follow prompts that may appear. Press Zero/Calibrate Sensor to perform required calibration steps. If the following settings are 'greyed', Internal or External zeroing is selected automatically based on the power meter/sensor model.

- **Internal Zero** - A switch inside the power sensor removes the zero circuit from the incident power.
- **External Zero** - Requires that you physically remove the sensor from incident power.

**Note for the U2000 Series USB power sensors**
Select External Zero ONLY when the power to be measured is below the specified level. Otherwise, the U2000 series performs internal zeroing automatically when needed. See your power sensor documentation for more details.

- U200xA - below -30 dBm
- U200xH - below -20 dBm
- U200xB - below 0 dBm

If your U2000 power sensor 'hangs' when external zeroing, upgrade the power sensor firmware to Rev. A.01.02.00 or higher to fix this problem.

**Copy a Source Power Calibration to other Channels**
A macro application is now available that copies a Source Power Calibration to other channels. Once downloaded and installed on a PNA, the macro is automatically configured up. To learn more, click Help on the application main dialog. Get the application from http://na.tm.agilent.com/pna/apps/applications.htm.

**Saving a Source Power Calibration**
Because Source Power Cal calibrates source hardware, the calibration data is saved as part of the Instrument State, in either a .sta file or a .cst file. This correction is applied to all measurements on the channel that uses the calibrated source. See Save Instrument State.

**Reducing Time to Complete a Source Power Calibration**
The time required to perform a Source Power Calibration depends on source power, number of points, and number of readings taken. You can reduce this measurement time with the following methods:

- **Reduce number of points before calibration.** You can reduce the number of points before the measurement, then return the number of points to its original value after calibration is complete and correction is ON. The analyzer will perform a linear interpolation, although with some loss in accuracy.
- **Use an Agilent E-Series sensor.** You can obtain 40+ readings per second over GPIB with this type of
sensor on the PNA.

- **Increase power to the sensor.** Lower power may have longer settling time with some sensors.

- **Check** Use Reference Receiver for Iteration.

**Receiver Power Calibration**

**Note:** Beginning with PNA firmware rev A.09.30, a Guided Power Cal can be performed during an S-parameter Guided Calibration. [Learn more.]

Receiver power calibration mathematically removes frequency response errors in the specified PNA receiver, and adjusts readings to the same, or a value offset from, the source power calibration level. It is the same as doing a **Response Cal** or **Data / Memory, (Normalization)** but with the data shifted to the **Cal Power** value.

Use Receiver Power Calibration to make very accurate absolute power (amplitude) measurements.

**Receiver Power Calibration:**

- Is ONLY allowed when making absolute power (**unratioed**) measurements.

- Is most accurate when a source power calibration was performed first.

- Applies to all unratioed measurements in the active channel using that receiver.

- Can be saved in a Cal Set and later reapplied to a like measurement.

**Interpolation**

Like other calibration types, if the original stimulus settings are narrowed, interpolation is applied and **C* Rcvr Pwr** is displayed in the status bar. If the original stimulus settings are made wider, the PNA will turn Receiver Power Correction **OFF**.

If the original settings are restored, then receiver power calibration returns to full correction.

**How to perform a Receiver Power Calibration**

1. Perform a [Source Power Calibration](#).

2. Set the active measurement to unratioed. [Learn How](#).

3. Connect a THRU line from the source port to the receiver port.

   - When performing a receiver power cal on a reference receiver, no connection is necessary as the receiver is internally connected to the source.

   - When the receiver port and the source port are the same (receiver A, source port 1), then connect an open or short to get maximum power to the receiver. This practice is not recommended. It is best to use different ports for the source and receiver.

4. Ensure correction for Source Power Calibration is ON as indicated by **Src Pwr Cal** or **Src Pwr Cal** in
5. Start the **Calibration Wizard**

### Using front-panel HARDKEY [softkey] buttons

1. Press **CAL**
2. then **[Power Cal]**
3. then **[Receiver Cal]**

### Using a mouse with PNA Menus

1. Click **Response**
2. then **Cal**
3. then **Power Cal**
4. then **Receiver Cal**

---

### Select Calibration Type for Unratioed Measurement dialog box help

**Cal Type Selection**  Select **Receiver Power**

**Receiver Power Configuration**

**Cal Power**  Specifies the power level to be displayed on the measurement when complete. (Source Port Power + Power Offset).

**Source Port Power**  Test port Power set for the measurement.  [Learn how to change Test Port Power](#)

**Power Offset**  Allows you to specify a gain or loss (in dB) to account for components you connect between the source and the reference plane of your measurement AFTER a source power cal has been performed. Following the calibration, the PNA power readouts are adjusted to the Cal Power value.

**Next**  Click to continue the Calibration Wizard.

**Notes:**

- When Receiver Power Cal is finished, **C RcvrPwr** is displayed in the status bar and correction data is applied to subsequent sweeps.

- To turn correction **OFF**, click **Calibration**, point to **Power Calibration**, then set **Receiver Power Correction** to **OFF**.

[Learn more about Receiver Power Cal (scroll up)](#)
**Saving a Receiver Power Calibration**

Beginning with PNA Revision 5.0, Receiver Power Cal is saved to a [Cal Register](#) and optionally to a [User Cal Set](#). It can be applied to measurements in the same way as other Cal Types. Previously, Receiver Power Cal data was saved as part of an Instrument State and was only applied to the measurement on which it was performed.

[Learn more about Saving PNA files types.](#)

Last modified:

- 28-Jun-2011 Modified First time note
- 10-Jun-2011 Removed LAN restrictive note
- 7-Mar-2011 Minor support edits
- 22-Oct-2010 Add N914x meters
- 21-Oct-2010 Add Calibrate at multiple power levels
- 30-Jan-2009 Added USB sensor note
- 11-Dec-2008 Added tolerance increment.
- 8-Dec-2008 Added persistence of power loss table
  - Link to characterize Adaptor Macro A.08.33
- 14-Aug-2008 Clarified a few concepts from BH.
- 24-Jun-2008 Updated diag image
- 21-Feb-2008 Added 848x note
- 4-Jan-2008 Added Cal note for USB sensors
- 30-Oct-2007 Added link to supported Power meters/ sensors
- 20-Jul-2007 Added USB / LAN support and Apply macro
- 21-Jan-2007 MX Added UI
- 14 Sept-2006 MQ Added Receiver-only SPC.
Fixture Simulator

The following features allow you to mathematically add (embed) or remove (de-embed) circuits to, or from, your PNA measurements. The mathematical models are applied to specific ports for all measurements on the channel.

Notes

- Beginning with PNA Rev. A.09.20, the following features are available in GCA, GCX, Swept IMD, Swept IMDX, Noise Figure, and NFX Apps:
  - Port Extensions (Not available in Swept IMD or IMDX)
  - 2 Port De-embedding
  - Port Matching
  - Port Z Conversion
- All other Fixturing features are available ONLY in a standard channel.
- When fixturing is enabled, all of the enabled fixturing features are applied when snp files are saved.

See Also

- Procedures: To Embed or De-embed?
- Characterize Adaptor Macro can be used to create S2P files from Cal Sets.
- "De-embedding and Embedding S-Parameter Networks Using a Vector Network Analyzer" App note, for more conceptual information on Fixture Simulation.
- See an example of how these functions can be used to de-embed unwanted effects of a test fixture, and then mathematically embed the DUT in the circuit in which it is used.

Order of Fixture Operations

- The fixturing operations are applied to the measurement results. The order of operations 1 through 4 can be changed using the SCPI command: CALC:FSIM:SEND:OORD. Learn how to send this command from the GPIB Command Processor Console.
- The order of the operations 5 through 8 can NOT be changed.
- In the PNA data processing chain, the Fixture Simulator functions occur at the same time as the Apply Error Terms block.

First, the following Single-ended measurement functions are processed in this order:

1. Port Extensions
2. **2-Port De-embedding**

3. **Port Matching Circuit Embedding**

4. **Port Z (Impedance) Conversion**

5. **4-Port Network (single-ended) Embed/De-embed**

Then, **Balanced** measurement functions are processed in this order:

6. **Balanced Conversion**

7. **Differential / Common Mode** Port Z Conversion

8. **Differential Matching Circuit Embedding**

Then, **Source power compensation** is optionally applied to compensate for the aggregate loss through all enabled fixturing operations.

**Note:** Port Z (Impedance) conversion uses values in the following prioritized order:

1. Balanced (Differential or Common Mode) - if enabled, these values are always used.
2. **Single Port Impedance** - if enabled, this value is used if Balanced is not enabled.
3. **System Impedance** - if neither balanced or single port is enabled, this value is used.

---

**How to select Fixturing Simulator**

**About Fixturing ON/off**

**BOTH** of the following must occur to turn a fixturing selection **ON**.

**EITHER ONE** will turn a fixturing selection **OFF**.

1. Check **Fixturing ON/off**

   Port Extensions is **NOT** affected by Fixturing ON/off.

2. Check **Enable** on the individual fixturing selection dialog box.

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[More]**
3. then **[Fixtures]**

**Using a mouse with PNA Menus**

1. Click **Cal**
2. then **More**
3. then **Fixtures**
Port Matching dialog box help

**Note:** Beginning with PNA Rev. A.09.20, this feature is available to the following measurement classes: GCA, Swept IMD, Swept IMDx, Noise Figure, NFX, and standard (S-Parameter) channels.

This function specifies a circuit to embed (add) to the measurement results. [See Order of Fixture Operations.](#)

**Enable Port Matching** Check to apply the settings to the measurement results. Must also enable Fixturing.
**ON/off.**

**Port** - Select Port in which to apply simulation.

**Circuit Model for Matching** - Choose one of the following that best emulates your fixture at the selected PNA port:

<table>
<thead>
<tr>
<th>Circuit Model</th>
<th>Diagram</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Series L - Shunt C</strong></td>
<td>![Diagram]</td>
</tr>
<tr>
<td><strong>Shunt C - Series L</strong></td>
<td>![Diagram]</td>
</tr>
<tr>
<td><strong>Shunt L - Series C</strong></td>
<td>![Diagram]</td>
</tr>
<tr>
<td><strong>Series C - Shunt L</strong></td>
<td>![Diagram]</td>
</tr>
<tr>
<td><strong>Shunt L - Shunt C</strong></td>
<td>![Diagram]</td>
</tr>
</tbody>
</table>

**User Defined (S2P File)** Load a file that is specified with User S2P File button.

**None** Use no circuit model.

**User S2P File** Click to specify an S2P file of the circuit model to embed at the selected port. If the normalized impedance value in a recalled User .S2P file is different from the port reference impedance setting of the PNA, the PNA setting is used. **Characterize Adaptor Macro** can be used to create S2P files from Cal Sets.

**Circuit Values**

- **Capacitance (C), Inductance(L), Resistance(R), Conductance(G)** Values for the specific components of the circuit type that models your fixture.

**Reset** Restores the default values.
De-Embed when you have performed a calibration and then added a fixture (an adapter, an attenuator, a longer cable, etc.) that connects between the Cal reference plane and your DUT. This function removes the effects of a test fixture from the measurement results.

The de-embedding operation recalls an .s2p file (Touchstone format) which includes the electrical characteristics of a 2-port fixture or device. The file can be in any standard format (real-imaginary, magnitude-angle, dB-angle).
Enable De-embedding  Check to apply the settings to the measurement results. Must also enable Fixturing ON/off.

Enable Extrapolation  Check to apply a simple extrapolation when the S2P file has a narrower frequency range than the channel. The values for the first and last data points are extended in either direction to cover the frequency range of the measurement. The frequency ranges of both the channel and the S2P file are displayed at the bottom of the dialog.

When extrapolation is necessary and enabled, a message is displayed showing the frequency range to be extrapolated. When extrapolation is necessary and disabled, a message is displayed offering to enable extrapolation.

This setting also causes 4-port Extrapolation to be enabled and disabled.

Port  The PNA port to which the recalled de-embedding file is applied.

From the drop-down menu, select User Defined (S2P File).

Reverse Adaptor Ports  Check to cause the Fixture/Adapter to be configured with Port 2 connected to the PNA and Port 1 to be connected to the DUT. The image in the dialog reflects that change.

User S2P File  Click to specify an existing .S2P file. If the normalized impedance value in a recalled User .S2P file is different from the port reference impedance setting of the PNA, the PNA setting is used. Characterize Adaptor Macro can be used to create S2P files from Cal Sets.

Port Z (Impedance) Conversion dialog box help

Note: Beginning with PNA Rev. A.09.20, this feature is available to the following measurement classes: GCA, Swept IMD, Swept IMDx, Noise Figure, NFX, and standard (S-Parameter) channels.

This function corrects the measurement and displays the results as if the measurement had been made into the specified impedance value. However, the physical port termination is still approximately 50 ohms.

The specified impedance value is applied to all of the measurements on ONLY the active channel.

See Order of Fixture Operations.

Enable Port Z Conversion  Check to apply the settings to the measurement results. Must also enable Fixturing ON/off.

R  Real part of the impedance value.

jX  Imaginary part of the impedance value.

Close  Applies the entries and closes the dialog box

See note about Port Impedance priority.
4-Port Embed/De-embed dialog box help

This function specifies a single-ended 4-port circuit (*.S4P file) to embed (add) or de-embed (remove) from the measurement results. Computation takes place BEFORE Balanced conversion. See Order of Fixture Operations.

There is a single normalized impedance value for each port in the *.S4P file. This impedance value must match the impedance of the previous Port Z setting, or the PNA port impedance.

The PNA will interpolate if the number of data points that are read is different from the current PNA setting.

**Enable 4-Port Embed/De-embed** Check to apply the settings to the measurement results. Must also enable Fixturing ON/off.

**Enable Extrapolation** Check to apply a simple extrapolation when the S4P file has a narrower frequency range than the channel. The values for the first and last data points are extended in either direction to cover the frequency range of the measurement. The frequency ranges of both the channel and the S4P file are displayed at the bottom of the dialog.

When extrapolation is necessary and enabled, a message is displayed showing the frequency range to be extrapolated. When extrapolation is necessary and disabled, a message is displayed offering to enable extrapolation.

This setting also causes 2-port Extrapolation to be enabled and disabled.

**Topology:** Select a DUT topology.

Refer to the images on the 4-port embed/De-embed dialog box.

- **A** - 2 PNA/DUT Ports
- **B** - 3 PNA/DUT Ports
- **C** - 4 PNA/DUT Ports

**The following 2 Topology configurations are NOT addressed with standard images in dialog box:**

1. If you have a 4-port DUT; 4-port network on one side; None on the other side.
• Specify **Topology C**.
  
  • Use 4-port Network on one side.
  
  • Use 4-port Network on the other side; set to **None**.

2. If you have a 3-port DUT and networks as follows:

   ![Diagram](image)

   • Specify **Topology B**.
   
   • Use 4-port Network1 on one side.
   
   • Use **2-port network** on the other side.

---

**NA Ports** - Select the PNA Port that is connected to each circuit port.

**Network Ports** Select the network ports that represent the configuration of the S4P file. By default, ports 1 and 2 are connected to the PNA and ports 3 and 4 are connected to the DUT.

**None, Embed, De-embed** For Network1 and Network2, select:

  - **None** - The same as disabling.
  
  - **Embed** - Add the specified network circuit to the measurement results. [See 2-port Embed image](image).
  
  - **De-embed** - Remove the specified network circuit from the measurement results. [See 2-port De-embed image](image).

**Browse** For both Network1 and Network2, navigate to find the .S4P file to embed or de-embed.

**OK** Applies the changes and closes the dialog box.

**Cancel** Does NOT apply the changes and closes the dialog box.
Differential Impedance Conversion dialog box help

This function sets the Differential impedance value for each balanced port.
The default value for $R$: is the SUM of the impedance values for both ports that make the logical port. If Port Z Conversion is not enabled, then System $Z_0$ values for both ports are summed.

See Order of Fixture Operations.

Enable Differential Z Conversion Check to apply the settings to the measurement results. Must also enable Fixturing ON/off.

Logical Port Select the logical (balanced) port to receive impedance value. To see logical port numbers, see the measurement topology.

R Real part of the impedance value.

jX Imaginary part of the impedance value.

Close Closes the dialog box.

See note about Port Impedance priority.

Common Mode Impedance Conversion dialog box help

This function sets Common Mode Impedance value for each balanced port.
The default value for $R$: is calculated as follows.

$$\frac{(Z_1 \times Z_2)}{(Z_1 + Z_2)}$$

Where ports 1 and 2 comprise the logical port:

$Z_1$ = the Port Impedance values for port 1
$Z_2$ = the Port Impedance values for port 2

If Port Z Conversion is not enabled, then System $Z_0$ values for port 1 and 2 are used in the calculation.

See Order of Fixture Operations.

Enable Common Mode Z Conversion Check to apply the settings to the measurement results. Must also enable Fixturing ON/off.

Logical Port Select the logical (balanced) port to receive impedance value. To see logical port numbers, see the measurement topology.

R Real part of the impedance value.
jX  Imaginary part of the impedance value.

**Close** Closes the dialog box.

See note about Port Impedance priority.

---

### Differential Port Matching dialog box help

This function allows the embedding of a differential matching circuit at a balanced port. See *Order of Fixture Operations*.

**Enable Differential Port Matching** Check to embed the selected matching circuit to the measurement results. Must also enable **Fixturing ON/off**.

**Logical Port** Choose **Logical DUT port** to receive the selected matching circuit. To see logical port numbers, see the **measurement topology**.

**Select Circuit** Select a matching circuit. Choose from:

- **Shunt L - Shunt C** Predefined circuit.

### Circuit Values

Choose from:

- **C** Capacitance value
- **G** Conductance value
- **L** Inductance value
- **R** Resistance value
- **User defined**  Select an *.S2P file that represents the matching circuit. Then click **Browse** to navigate to the *.S2P file.

  **Note:** For the *.S2P file:
  - Port 1 of the circuit is assumed to be connected to the PNA
  - Port 2 of the circuit is assumed to be connected to the DUT.

- **None**  No embedded circuit on selected port.

**Close** Closes the dialog box.

---

**Power Compensation** dialog help

This function adjusts the source power at the specified port to compensate for the combined amount of gain or loss through ALL enabled fixturing operations. Use this function to set the power level at the DUT input.

For example:

- Your DUT requires a fixture on the input port which is connected to PNA port 1.
- The fixture description (such as an S2P file at the [2-port De-embed function](#)) indicates the fixture has approximately 2 dB of loss across the frequency span.
- You set source power to 0 dBm. But you want 0 dBm at the DUT input (the fixture output).
- Check Power Compensation on Port 1 and enable **Fixturing**.
- Power Compensation causes the source power to be increased by approximately 2 dB so that the power at the fixture output plane will remain at 0 dBm.

Power Compensation affects all measurements in the channel.

Enable **Fixturing** to use Power Compensation.

**Note:** Use caution when applying power compensation. Always test your setup without a DUT in a place. If you
are using S2P files, Recall your S2P file into the PNA so you can verify that the device your S2P file describes is what you intended it to be. It is too easy to misalign data in S2P files if they are constructed by hand.

**Fixture Simulator Example**

The following example shows a DUT and the matching circuit with which the DUT will be used in its intended application. When the DUT is tested in a high-volume manufacturing environment, multiple test fixtures are often required. The most accurate way to test the DUT and ensure measurement consistency between the different test fixtures is to use a simple, repeatable, test fixture without the actual matching elements.

To get the desired performance data, the parasitic effects of the fixture must first be removed (de-embedded) from the measured data. Then a perfect "virtual" matching circuit must be simulated and added mathematically (embedded) to the corrected, measured data. The result is an accurate display of the DUT as though it was actually tested with a physical matching circuit, but without the uncertainties of using real components.

![Test Device and the circuit in which it will be used.](image)

Rs = 50 Ω
RL = 200 Ω

**Circuit Simulation**

Port Z Conversion Port Extensions/ De-embed Balanced Conversion

Port Matching Diff Port Match Balanced Z Conversion

1. **Create a balanced measurement** using single-ended to balanced (SE-Bal) topology. Include all relevant measurement settings (IFBW, number of points, and so forth). Once the measurement is created and
calibrated, the measurement parameter can be easily changed. For example, Sdd22 to Sds21.

2. Calibrate the measurement at the point where the simple test fixture is connected to the PNA. Use accurate calibration standards and definitions.

3. Remove the effects of the three uncalibrated transmission lines of the simple test fixture. This can be done in several different methods. The easiest is to use manual or automatic **Port Extensions** to move the calibration reference plane to the DUT. This removes the electrical length and loss of the fixture’s transmission lines, but does not account for fixture mismatch. Another method is to de-embed previously-created *.S2p files of the 3 transmission lines. The files can be created using external ADS modeling software. Another alternative is to create the *.S2P files by independently measuring all 3 ports of the test fixture and saving the results of each to an S2P file.

4. With the test fixture connected to the PNA and a DUT inserted, the measurement results now appear as though calibration was performed at the connections to the DUT, and the device was measured in a 50-ohm single-ended test environment. The following steps will cause the results to reflect the performance of the device as though the device is embedded in the circuit in which it will be used.

5. Port 1 of the device is a single-ended port and sees a source impedance the same as the PNA system impedance, so no change is required. However, if Rs were a value other than 50 ohms, **Port 1 Impedance Conversion** would be used to simulate the different impedance.

6. **Port Matching** is used to simulate L1 inductance. Select any of the Shunt L circuits to embed (add) to the measurement results. Enter the value of L and R. The C and G values can be entered as 0 (zero).

7. **Port Matching** is used to simulate C1 and C2 capacitance. For both port 2 and port 3, select any of the **Series C** circuits to embed (add) to the measurement results. Enter the value of C and G. The L and R values can be entered as 0 (zero).

8. **Balanced Conversion** mathematically simulates the measurement in balanced mode.

9. **Differential Port Matching** is used to simulate L2 inductance. Select Shunt L- Shunt C and enter the inductance/resistance value. The C and G values can be entered as 0 (zero).

10. Finally, **Differential Z Conversion** is used to simulate a circuit termination of 200 ohms. If you are making Common Mode measurements, specify **Common Mode Z Conversion**.

Last modified:
12-Nov-2010  Added extrapolation and reverse (A.09.33)
12-Apr-2010  Updated for A.09.20
11-Nov-2009  Add SNP note
2-Feb-2009   Reorder and add CAN change first four
29-Sep-2008  Added note about standard channel
3-Sep-2008   Removed legacy content
Sept 12, 2006 Added link to programming commands
Port Extensions

Port extensions allow you to electrically move the measurement reference plane after you have performed a calibration.

Note: Beginning with PNA Rev. A.09.20, this feature is available to the following measurement classes: GCA, Noise Fixture, NFX, and standard (S-Parameter) channels.

- Why and How to use Port Extensions
- Manual Port Extensions Procedure
- Port Extensions dialog and Toolbar
- Step Size dialog
- Automatic Port Extension dialog

See Also
- PNA Data Flow Map
- Fixture Compensation features
- Phase Accuracy
- Comparing the PNA Delay Functions

Why use Port Extensions

1. You are unable to perform a calibration directly at your device because it is in a test fixture. Perform a calibration at a convenient place, then use port extensions to compensate for the time delay (phase shift), and optionally the loss, caused by the fixture.

2. You have already performed a calibration, and then decide that you need to add a length of transmission line in the measurement configuration. Use port extensions to "tell" the analyzer you have added the length to a specific port.
Important Note: Port Extensions and PNA Data Flow

See PNA Data Flow diagram

Normally, Port Extensions are applied to individual S-parameters in the Phase Correction process and only applies to displayed S-parameters.

However, when Fixturing is ON or when making a Balanced Measurement, Port Extension compensation is applied in the Apply Error Terms process which affects ALL S-parameters, whether displayed or not. This allows all underlying S-parameters to have proper extensions applied.

Therefore, when using Port Extensions with features that require more than a single S-parameter (such as k-factor in equation editor), do one of the following:

- Enable Fixturing - Individual Fixturing features are NOT required to be enabled.
- Use 8510 Mode Data Processing.

When Port Extension compensation is applied in the Apply Error Terms process, after a Data-to-Memory operation has been performed, further changes to Port Extensions settings will NOT be applied to the Memory trace.

How to use Port Extensions

- If you know the electrical length of the fixture or additional transmission line, enter the value directly to the Time setting.
- If you know the physical length of the fixture or additional transmission line, enter the value directly to the Distance setting.
- If you do NOT know either the electrical or physical length of the fixture or additional transmission line, you must be able to connect an OPEN or SHORT to the new reference plane - in place of the DUT. In most cases, removing the DUT will leave a suitable OPEN at the new reference plane.
- Port Extensions can then be added manually (as follows), or by using Automatic Port Extensions.

Manual Port Extensions Procedure

1. Select a calibrated S11 measurement.
2. Select Phase format.
3. With an OPEN or SHORT at the calibration reference plane, verify that the phase across the frequency span is at or near zero.
4. Connect the fixture or added transmission line and attach an OPEN or SHORT in place of the DUT. In most cases, removing the DUT will leave a suitable OPEN at the new reference plane. On the Port Extension toolbar or dialog, increase either Time or Distance until the phase response is flat across the frequency span of interest.
5. If you know the loss of the additional transmission line, enter the Loss Compensation values using either one or two data points.

**Note:** Most OPEN and SHORT standards have delay. Therefore, adjusting delay with this method results in a delay equal to two times the delay of the OPEN or SHORT.

### How to access Port Extensions settings

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[Port Extensions]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Cal**
3. then **Port Extension**

---

**Port Extensions dialog and Toolbar help**

**Note:** Beginning with PNA Rev. A.09.20, this feature is available to the following measurement classes: GCA, Noise Fixture, NFX, and standard (S-Parameter) channels.

Port extensions settings affect all measurements on the active channel that are associated with a particular port.

- **Learn Why and How to use Port Extensions** (scroll up).

- **Port Extension** Turns ON and OFF port extensions on all ports.

- **Port** Select a PNA port for delay and loss values. Port Extensions settings affect ALL measurements on the active channel that are associated with a particular port.

- **Show Toolbar** Check to show the Port Extensions toolbar. The toolbar allows you to make adjustments to the port extensions while showing more of the PNA screen. Beginning with PNA rev A.08.50, this is the only way to
Delay settings

Enter delay in either Distance or Time by entering a value or clicking the up/down arrows. Click to start the Step Size dialog.

Time  The amount of port extension delay in time. Enter a positive value.

Distance  The amount of port extension delay in physical length. Enter a positive value.

Distance Units  (Dialog ONLY) Select from Meters, Inches, or Feet. The Step Size setting will not change automatically. Learn more.

Loss Compensation

The following settings allow the entire frequency span to be corrected for loss.

Loss at DC  Offsets the entire frequency span by this value. Loss1 or Use1 must also be checked. To compensate for loss at DC, enter a positive value which causes the trace to shift in the positive (up) direction.

Loss @Frequency  Check the box, and enter values for Loss and Frequency

When Loss1 or Loss1/Loss2 are used, a curved-fit algorithm is used as follows:

Loss1 ONLY:

\[ \text{Loss}(f) = \text{Loss1} \times \left(\frac{f}{\text{Freq1}}\right)^{0.5} \]

Loss1 and Loss2:

Set the lower frequency to Loss1, and the higher frequency to Loss2.

\[ \text{Loss}(f) = \text{Loss1} \times \left(\frac{f}{\text{Freq1}}\right)^{n} \]

Where:

\[ n = \frac{\log_{10} \left| \text{abs(\text{Loss1}/\text{Loss2})} \right|}{\log_{10} (\text{Freq1}/\text{Freq2})} \]

Note: \text{abs} = \text{absolute value}

Velocity

Velocity Factor  For each port, sets the velocity factor that applies to the medium of the device that was inserted after the measurement calibration. The value for a polyethylene dielectric cable is 0.66 and 0.7 for PTFE dielectric. 1.0 corresponds to the speed of light in a vacuum.

Couple to system Velocity Factor  When unchecked, the Velocity Factor is set for only the specified port and only for Port Extensions. When checked, sets the Velocity Factor for all ports. In addition, changing this value also changes this setting for the Electrical Delay and Time Domain Distance Marker features.

Media

For each port, select the media of the added transmission line or fixturing.

Coax  Select when the fixture or added transmission line is coax. Also specify the velocity factor of the coax.

Waveguide / Cutoff Frequency  Select when the fixture or added transmission line is waveguide. Also enter cutoff (minimum) frequency of the waveguide.

Note: when using a Waveguide cal Kit, set System Z0 to 1 ohm before calibrating.

Couple to system Media Definition.  When unchecked, the Waveguide Cutoff Frequency is set for only the specified port and only for Port Extensions.  When checked, sets the Waveguide Cutoff Frequency for all

433
ports. In addition, changing this value also changes this setting for the Electrical Delay feature.

**Reset** All port extensions settings are changed to preset values. The Port Extension ON / OFF state is NOT affected.

**Auto Ext.** Starts the Automatic Port Extensions dialog box.

**Note:** Individual receiver port extensions (A,B, and so forth) can no longer be set. (Sept. 2004)

---

**Step Size** dialog box help

Changes the step size that occurs when the Time or Dist up/down arrows are pressed on the Port Extension toolbar. The Units for step size are changed on the Port Extension dialog.

**Auto** Step Size is set to the default value.

**User Defined** Enter a step size value, then click OK.

This value remains the same when the units are changed. For example if a step size of 12 is entered on this dialog, then you change the units from Inches to Feet, the step size of 12 inches becomes 12 feet, not 1 feet. Therefore, change the units first, then set the step size.

Learn about Port Extensions (scroll up)

---

**Automatic Port Extension** dialog box help

Automatic Port Extension AUTOMATICALLY performs the same operation as Manual Port Extension. By
connecting a SHORT or OPEN, the reference plane is automatically moved to the point at which the standard is connected. In addition, Automatic Port Extension will optionally measure and compensate for the loss of the additional transmission line.

Auto Port Extension is NOT available when:

- Sweep type is set to power sweep
- Frequency Offset is ON
- Media is set to Waveguide

**Note:** Turn OFF Equations that may exist on the active trace when using Automatic Port Extensions.

**Auto Port Extensions Procedure**

1. Connect the added transmission line or fixture. Attach an OPEN or SHORT to all affected ports at the new reference plane. In most cases, removing the DUT will leave a suitable OPEN at the new reference plane.

2. On the Port Extension toolbar, click **Auto Port Ext**. Click **Show Configuration** to make additional settings.

3. Click **Measure** to perform the port extension calculations. The resulting delay and loss settings are entered into the port extension toolbar. These settings are saved with Instrument Save or you can manually record the values and enter them again when required.

**Settings**

**Measure either OPEN, SHORT, or both** Press a button to make the measurement of the reflection standard. Measure either OPEN or SHORT depending on which is most convenient. An ideal OPEN and SHORT, with zero loss and delay, is assumed. Therefore, accuracy is most affected by the quality of the standard. In most cases, removing the DUT will leave a suitable OPEN at the new reference plane. When measuring both OPEN and SHORT standards, the average of the two is used and will slightly improve accuracy.

**Selected Ports** Indicates the ports that currently have automatic port extension enabled. By default, ALL PNA ports are enabled. To disable a port, see **Measure on Port Number** below.

**Note:** Port Extensions settings affect ALL measurements on the active channel that are associated with a particular port.

**Show/Hide Configuration** Press to either show or hide the following configuration settings in the dialog box.

**Measure on Port Number**

Select port number to enable or disable automatic port extension.

- **Enable** Check to enable the specified port. All enabled ports will have their reference plane automatically adjusted after performing Automatic Port Extension.

- **Include Loss** Check to automatically measure the loss in the additional transmission line and apply compensation. To calculate loss compensation, frequencies at 1/4 and 3/4 through the frequency range are usually used as Freq1 and Freq2 values. Learn more about Loss Compensation.

- **Adjust for Mismatch** Only available when **Include Loss** is checked. During the measurement of the OPEN or
SHORT standard, mismatch could cause ripple in the magnitude (loss) response. The Loss compensation curved-fit algorithm allows half of the ripple to be positive and half negative. When measuring low-loss devices, it is possible that some magnitude responses could become slightly positive, indicating gain rather than loss.

Check - Offsets the trace to cause all of the data points to be at or below zero.

Clear - Most accurate application of the curve-fit calculation, but allows positive responses.

**Prompt for Each Standard** Check to invoke a prompt when the Measure OPEN or SHORT button is pressed. The prompt will indicate which standard to connect to which port.

**Method**

Select the span of data points which will be used to determine correction values for phase and loss (optional). If a portion of the current frequency span does not have flat or linear response, you can eliminate this portion from the calculations by using a reduced User Span.

To calculate loss compensation, Current Span and User Span methods usually use frequencies at 1/4 and 3/4 through the frequency range as Freq1 and Freq2 values. See Loss Compensation to learn more about how loss is calculated.

- **Current Span** Use the entire frequency span to determine phase and loss values.
- **Active Marker** Use only the frequency at the active marker, and one data point higher in frequency, to calculate phase and loss values. If a marker is not present, one will be created in the center of the frequency span.
- **User Span** Use the following User Span settings to determine phase and loss values.

**User Span**

- **Start** Enter start frequency of the user span.
- **Stop** Enter stop frequency of the user span.

Learn about **Port Extensions** (scroll up).

See also **Comparing the PNA Delay Functions**

---

Last modified:

- 1-Jun-2011  Edited data flow note
- 12-Apr-2011  Added note about data flow
- 18-May-2009  Added note about equations
- 5-Feb-2009  Added many features (A.08.50)
- 3-Sep-2008  Removed legacy content
- 9/12/06  Added link to programming commands
Characterize Adaptor Macro

This external Macro application creates an S2P file that models a device such as an adaptor, the input OR output side of a test fixture, or an on-wafer probe head. This is done by calculating the four S-parameters of the device from two 1-port calibrations. Such S2P files can be used for embedding or de-embedding the device from S-parameter measurements and FCA calibrations.

This application, along with the FCA Embed/De-embed feature, can be especially useful when performing FCA calibrations.

- An SMC calibration requires a power meter measurement at the port 1 reference plane. This could be very difficult in on-wafer applications where the measurement reference plane is at the tip of a probe. This macro, in conjunction with the FCA Embed/De-embed feature, enables you to model the probe and connect the power sensor at the coax connector where the probe connects.

- Likewise, a VMC calibration requires that a calibration mixer be used for the Thru standard. Again, this can be very difficult in on-wafer applications where the measurement reference plane is at the tip of a probe. This macro, in conjunction with the FCA Embed/De-embed feature, enables you to model the probe and connect the calibration mixer at the coax connectors where the probe connects.

New: Characterize Adaptor Macro - Version A.02.10

- Reverse S2P files
- Loads the PNA Power Loss Table from an existing S2P file.
- This macro update requires PNA firmware version A.08.20.04 or higher

For S-Parameter measurements:

- To Embed or De-embed
- Procedures

How to start the Characterize Adaptor Macro

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press MACRO</td>
<td>1. Click Utility</td>
</tr>
<tr>
<td>2. then [AdaptorChar]</td>
<td>2. then Macro</td>
</tr>
<tr>
<td></td>
<td>3. then AdaptorChar</td>
</tr>
</tbody>
</table>

Programming Commands
Characterize 2-Port Adaptors, Probes, Fixture Paths (S2P) dialog box help

Important Notes

- The device to be characterized (probe, adapter...) MUST be reciprocal (S21 = S12).
- Two 1-port cals must be performed and saved to Cal Sets BEFORE using the Characterize Adaptor application.
- The frequencies and number of points of the two Cal Sets MUST be identical.
- **CRITICAL**: The calculations that are performed to create the S2P file require that **Calset 1** ALWAYS be from the side closest to the PNA and **Calset 2** ALWAYS be from the other side of the device. If your application that uses the resulting S2P file requires that the ports be reversed, check **Reverse Port Order**. Or learn how to reverse the port order on an existing S2P file.
- The majority of this topic describes the characterization of a 2-port device from two 1-port cal sets. However, you can also use the cal sets from two 2-port calibrations, or one 2-port and two 1-port calibrations. See procedures for both.

Learn more about the Characterize Adaptor Macro.(scroll up)

**Connected** <PNA host name>  The two 1-port Cal Sets can reside in another PNA. Click to connect to another PNA that is DCOM configured. Learn how to configure DCOM.

**Adaptor Type**  Select the type of device to be characterized.

**Note**: The image that appears in the macro does not influence the calculations. It only appears to help you visualize the measurement reference plane of the Cal Sets.
Select Calset 1 and Calset-2  Select a 1-port Cal Set from each list. Although all Cal sets are listed, only the Cal Sets that have error terms to satisfy a 1-port calibration may be used.

Cal Port  Select the port within the selected Cal Set which represents the modeled device. The Cal Ports must be the same for both selected Cal Sets.

Characterize and Save  Calculates four S-parameters, then invokes the Save As dialog with S2P file type. This button is not available until valid Cal Sets and Cal Ports are selected.

Reverse Port Order  By default, the calculations that are performed to create the S2P file require that Calset 1 ALWAYS be from the side closest to the PNA and Calset 2 ALWAYS be from the other side of the device. Check Reverse Port Order to cause the ports to be reversed in the resulting S2P file.

Reverse S2P
Click Actions, then Reverse S2P File
This action causes ports to be reversed on an existing S2P file. For example:

- The data for S11 becomes the data for S22 and vice versa.
- The data for S21 becomes the data for S12 and vice versa.

The resulting file is written in the standard PNA S2P file format.

1. Navigate to the S2P file to be reversed.
2. Navigate to where the new reversed S2P file will be saved. By default, the file is saved to the same folder as: [old filename]_Reversed.s2p.
3. Click OK.

Write to Power Loss Table
Click Actions, then Write to PNA Pwr Loss Table.
**Power Loss Table Setup** dialog box help

Loads the S2P Frequency / Loss pairs into the PNA Power Loss Compensation table to compensate for losses that occur when using the device to connect a power sensor to the measurement port during a Source Power Cal.

**Note:** PNA firmware revisions prior to A.08.33.07 ONLY support up to 100 segments in the power loss table. Therefore, to save data to the power loss table in one of the earlier firmware revs, then the Cal Sets that created the S2P file must contain 100 data points or less. Revision A.08.33.07 and above allows any number of segments.

**Use Selected Calsets** Computes the transmission loss of the fixture based on the selected Cal Sets. This choice is NOT available until two valid Cal Sets are selected.

**Use S2P File** Uses the S21 data in an existing S2P file to build the PNA’s power loss table. Select, then click ... then navigate to the S2P file, then click OK.

**Note:** In the PNA Power Loss Compensation table, loss is expressed as a positive number. The macro assumes that any negative S21 value in the S2P file is a loss and therefore multiplies the S21 values in the file by -1 to express that value as a positive number. This ensures proper handling of the offset during a source power cal.

---

**Change Phase Pivot Point**

Click **File**, then Set Phase Pivot Point.

![Change Phase Pivot Point](image)

- Degrees
- Radians

Enter the Phase Pivot Point Below:

```
0
```

[OK] [Cancel]
Change Phase Pivot Point dialog box help

Before the A.01.50 revision (see Help, About) the Characterize Adaptor macro projected the phase of S21 at DC to cross the X-axis between 0° and -180°, based on the delay of the adapter. For most adapters the value is 0°.

However, when characterizing electrically long cables, cables with significant mismatch, or high noise in the measurements, it is possible that the projection of phase goes above 0°. This results in a 180° phase difference between the results computed by the macro versus the results you might get by measuring the same adapter with a 2-port calibration.

This revision allows you to adjust the center of the 180° phase pivot window. The default value 0° yields a window between +90° to -90° which should be adequate for the majority of adapters.

Enter the value in degrees or radians. This value remains while the application is running.

Restarting the application will result in the Phase Pivot Point to revert back to 0°, the default value.

To Embed or De-embed S-Parameter Measurements

- For FCA measurements, see FCA Embed or De-embed.
- In this section, the term “adapter” can mean any type of 2-port device and its associated S2P file.

To make an accurate measurement, the setup configuration during the DUT measurement MUST exactly match the setup configuration during Calibration. In other words, if you calibrate with an adapter, you must also measure the DUT with the adapter.

However, the PNA provides some flexibility by allowing you to ‘Virtually’ add (embed) or remove (de-embed) an adapter from the measurement. Knowing how to do this can be confusing.

- To perform a calibration WITHOUT the adapter, but make DUT measurements WITH the adapter, then De-embed (remove) the adapter from the DUT measurement.
- To perform a calibration WITH the adapter, but make DUT measurements WITHOUT the adapter then Embed (add) the adapter during the DUT measurement.

Procedures

- Create an S2P file using Characterize Adaptor Macro
- De-embed the S2P file from DUT measurement
- Embed the S2P file in DUT measurement
- De-Embedding a Fixture that has a THRU Standard
- De-Embedding a Fixture with No THRU Standard
Create an S2P file using the Characterize Adaptor Macro

1. Configure your PNA measurement (frequency span, power level, IF bandwidth, and number of points).

2. Perform a 1-port SmartCal at the reference plane. Save the cal to a User Cal Set using a descriptive name (for example, Ref Plane).

3. Connect the adapter to be characterized at the reference plane.

4. Perform another 1-port SmartCal at the end of the adapter. Save it to a User Cal Set using a different descriptive name (for example, Adapt End).

5. Start the Characterize Adaptor Macro.

6. In the Select Calset1 field of the dialog box, select the Cal Set for the reference plane (from step 2 above).

7. In the Select Calset2 field of the dialog box, select the Cal Set for the end of the adapter (from step 4 above).

8. Click Characterize and Save. In the resulting dialog box, enter the .S2P file name and location.

9. Click Close.

De-embed the adapter (S2P file) from subsequent S-parameter measurements

See Fixture De-embedding

Note: Subsequent measurements must have the same or smaller frequency range (within the Start / Stop frequencies) as that of the S2P file.

1. Perform a 2 port SOLT calibration without the adapter/fixture.

2. Select 2-port De-embedding: click Response, then Cal, then More, point to Fixtures, then click 2 port De-embedding.

3. Select the Port to add the adapter to, then select User Defined (S2P file).

4. Click Use S2P file and select the S2P file created using the Characterize Adaptor macro.

5. Check Enable De-embedding, then click Close.

6. Enable Fixturing: click Response, then Cal, then More, point to Fixtures, then click Fixturing on/OFF.

7. Sim appears in the Status Bar to indicate that Fixture Simulation is ON.

Embed the adapter (S2P file) into subsequent S-parameter measurements

The adapter (S2P file) can also be a matching network. Learn more.

1. Perform a 2 port SOLT calibration including the adapter. Note the port number on which the adapter is calibrated.

2. Select Port Matching: click Response, then Cal, then More, point to Fixtures, then click Port Matching
3. Under Choose Circuit Model for Matching, select the Port that the adapter was on during calibration, then select **User Defined (S2P file)**.

4. Press **Use S2P File** and navigate to the S2P file created using the Characterize Adaptor macro.

5. Check **Enable Port Matching**, then click **Close**.

6. Enable Fixturing: click **Response**, then **Cal**, then **More**, point to **Fixtures**, then **Fixturing on/OFF**.

7. **Sim** appears in the Status Bar to indicate that Fixture Simulation is ON.

**De-Embedding a Fixture that has a THRU Standard**

A test fixture is generally regarded as a single 'bed' in which a DUT is placed. However, for modeling purposes such as this, it is separated into two circuits: Fixture A on the input of the DUT, and Fixture B on the output.

Use this procedure to perform calibrations **WITHOUT** the test fixture while making measurements **WITH** the test fixture. A calibration is performed **once WITH** the test fixture, and then again as it wears with use and electrical performance changes. The fixture is de-embedded from subsequent measurements to match the regular calibrations that are performed without the fixture.

If you have a THRU standard for your test fixture, you can perform a full 2-port calibration in the fixture, and from that create the required S2P files for de-embedding.

![Diagram of test fixture and PNA connection](image)

1. Perform a full 2-port **CAL 1** at the connections of the PNA to the fixture as shown above. Save to **MyCalSet1**.

2. Perform a full 2-port **CAL 2** where the DUT is inserted (reference plane). Save to **MyCalSet2**.

Follow the Create an S2P file procedure, beginning with step 6, using the following selections:

1. Create #1 S2P file:
   1. For CalSet1, choose **MyCalSet1** and select **CalPort=1**
   2. For CalSet2, choose **MyCalSet2** and select **CalPort=1**
   3. Save to **FixtureA.s2p**

2. Create #2 S2P file:
   1. For CalSet1, choose **MyCalSet1** and select **CalPort=2**
2. For CalSet2, choose **MyCalSet2** and select **CalPort=2**

3. Save to **FixtureB.s2p**

Follow steps in *To De-embed the adapter...*

Perform these steps TWICE; once for each of the following S2P files:

1. For PNA Port 1, select **FixtureA.s2p**
2. For PNA Port 2, select **FixtureB.s2p**

**De-Embedding a Fixture with No THRU Standard**

This procedure is a slight modification of the above. Cal 2 is performed from two 1-port cals when a THRU standard for the fixture is not readily available.

1. Perform a full 2-port **CAL 1** at the connections of the PNA to the fixture as shown above. Save to **MyCalSet1**.

2. **CAL 2** is performed using two 1 port cals

   - **Cal2A** at the Fixture A / DUT plane. Save to **MyCalSet2A**
   - **Cal2B** at the Fixture B / DUT plane. Save to **MyCalSet2B**

Follow steps in *Create an S2P file...Step 6* above, except:

1. Create #1 S2P file:
   1. For CalSet1, choose **MyCalSet1** and select **CalPort=1**
   2. For CalSet2, choose **MyCalSet2A** and select **CalPort=1**
   3. Save to **FixtureA.s2p**

2. Create #2 S2P file:
   1. For CalSet1, choose **MyCalSet1** and select **CalPort=2**
   2. For CalSet2, choose **MyCalSet2B** and select **CalPort=2**
   3. Save to **FixtureB.s2p**

Follow steps in *To De-embed the adapter* above, except:

1. For PNA Port 1, select **FixtureA.s2p**
2. For PNA Port 2, select **FixtureB.s2p**
Last modified:

25-Mar-2010  Added support for version 2.1
23-Mar-2009  Fixed error in "De-embedding a fixture..." procedures
8-Dec-2008   Added support for version 2.0
3-Sep-2008   Removed legacy content
10-Mar-2008  Added Change Pivot dialog
13-Mar-2008  Moved FCA embed or de-embed
12-Feb-2008  New procedures
24-Jan-2008  Fixed error in procedures and added section
30-Nov-2007  Clarified and highlight order of calsets.
26-Feb-2007  Fixed " Note: Subsequent measurements...".
12-Sept-2006 Added link to programming commands
CalPod (Opt 301 or 302)

CalPod is a system that simplifies the process of recalibrating the PNA without requiring the removal of the DUT or the physical connection of standards. This allows recalibration from a remote location such as when the DUT is in a temperature chamber.

In this topic:

- Overview
- How to start the CalPod dialog
- CalPod dialog
- CalPod Setup dialog
- CalPod Operational Check

Other Calibration topics

Process Overview

Note: The following overview assumes the CalPod system has been installed and configured. See the CalPod User's Guide for installation instructions at: http://cp.literature.agilent.com/litweb/pdf/85523-90001.pdf

The following process assumes a 2-port DUT connected to the PNA ports 1 and 2 through CalPod modules as follows:

1. After configuring and assigning CalPod modules to PNA ports 1 and 2, connect the CalPod modules to the PNA, directly or using short cables. Learn how to configure CalPod.

2. Setup measurements on a channel. An IFBW of 1 kHz or lower with eight averages is recommended.
CalPod does not support measurements below 100 MHz.

3. Perform a full 2-port calibration for the channel with the CalPod outputs as the reference plane.

4. Click **Initialize Channel** to automatically perform the following steps:

   a. The OPEN, SHORT, AND LOAD states of both Calpod modules are switched in and S11/S22 are measured.

   b. The resulting measurements are stored in the channel's Cal Set as additional standard measurements. These measurements are used to characterize the Calpod states - they are NOT used at this time to change the error correction.

**Notes:**

- Because the OPEN, SHORT, AND LOAD states in the CalPods are measured, it is not important what is connected to the CalPod when Initialize is pressed. Therefore, for highest accuracy, click Initialize IMMEDIATELY and ONLY ONCE after performing the calibration - before causing ANY cable movement.

- If an adapter is required to connect the DUT to a CalPod, use a high-quality adapter. Any temperature drift due to the adapter is NOT recorrected.

- Always connect the DUT as close as possible to the CalPod modules.

5. Connect the DUT to the CalPod outputs.

6. Click **Recorrect Channel** or **Recorrect All Channels** whenever necessary. Any of the following actions will cause the current calibration to become invalid and require recorrection:

   a. Moving the CalPod modules to the ends of long cables.

   b. Changing the cables.

   c. Extreme temperature variations.

   d. Measurement drift over long time periods.

7. The following steps occur automatically during recorrection for the active channel:

   a. The OPEN, SHORT, AND LOAD states of both CalPod modules are switched in and S11/S22 are measured.

   b. Additional (de-embedded) error terms are computed to compensate for changed conditions from the Initialize measurements.

   c. Another Cal Set is created using the original name with the CalPod number appended. The modified error terms are saved to that Cal Set and applied to the channel. The measurements are now fully corrected.
**How to start the CalPod dialog**

**Using front-panel HARDKEY [softkey] buttons**
1. Press `CAL`
2. then `[More]`
3. then `[CalPod]`
4. then `[CalPod...]`

**Using a mouse with PNA Menus**
1. Click `Response`
2. then `Cal`
3. then `CalPod`
4. then `CalPod`

---

**CalPod dialog box help**

**Learn all about the CalPod process (Scroll up)**

**Initialize Channel**  Calibrated measurements of the CalPod states are performed as initial reference data points for the active channel.

**Initialize All Channels**  Calibrated measurements of the CalPod states are performed as initial reference data points for all current channels. This command is not recommended, it is generally preferable to initialize each channel immediately following calibration.

**Recorrect Channel**  Recorrects the active channel Cal Set to match the initial reference.

**Recorrect All Channels**  Recorrects the Cal Sets on ALL channels that were initialized.

**Correct Power**  When checked, the power output at the PNA port is adjusted to compensate for any change in path loss when Recorrect is performed. For example, if the path loss between the PNA port and the CalPod was increased by two dB following initialization, then the PNA output power will be increased by two dB upon recorrection. Do this when you add a significant amount of loss in the calibration path, or when the power level at the DUT is important.

If a Source Power Cal is not present, one is created and the measured loss values are entered to correct the source power. After recorrection, `SrcPwrCal` will appear on the PNA status bar.

When a significant amount of loss is introduced in the calibration path, it may not be possible to increase the source power enough to overcome the loss. In this case, an **Unleveled source** message may appear on the PNA screen.
When the checkbox is cleared, the source power level is not corrected.

**CalPod Assignments**
For each PNA port, select a CalPod module.

**CalPod Setup** Starts the CalPod Setup dialog

**Delete All CalPod Cal Sets** Deletes all recorrection Cal Sets and reinstates the Initialization Cal Set.

---

**CalPod Setup dialog box help**

To start this dialog, click **CalPod Setup** in the CalPod dialog box.

**CalPod Serial Number** Type the CalPod module (without 'sn'), then click **Add CalPod**. The new module is added to the list of available CalPod modules. (The list of available CalPod modules is limited to four entries when option 301 is the only PNA CalPod option.)

**Serial # and CalPod Types**

Shows the list of available CalPod modules. A CalPod module type may be STANDARD or THERMAL. A CalPod module will be listed as a STANDARD type unless two conditions are met: the PNA has option 302 and the thermal characterization data has previously been loaded into the PNA from a USB flash drive.

**Buttons**

**Test** Click to test the connection between the controller and the selected CalPod module. The message box displays the connection status and temperature for both Ambient and Thermal modules. Only the Thermal module will apply test temperature for recorrection.

**Delete** Removes the selected STANDARD CalPod module from the list.
To delete a THERMAL CalPod from the list:

1. Navigate to the c:\e-trak\adapters\itm directory.
2. Delete the .xml file associated with the CalPod serial number.
3. Exit all CalPod dialog boxes and restart the CalPod dialog.
4. The CalPod may now be removed using the **Delete** button.

**About** Shows the CalPod software version information.

For more CalPod Setup information, see the CalPod web site: [http://na.tm.agilent.com/pna/calpod](http://na.tm.agilent.com/pna/calpod). Click **CalPod Controller Configuration**.

### CalPod Operator's Check

This program is provided as a convenience to help determine the operational status of each 855xxA Series CalPod and its associated CalPod Controller. While this check is not intended to be a complete test, it does check each unit enough to provide greater than 95% confidence that the CalPod is functioning properly.

- When the max frequency of the CalPod is higher than the max frequency of the PNA, the full frequency range of the CalPod is not tested.
- Up to four CalPod modules may be checked at once. All four devices must be of the same frequency range.
- The software revision for the Operator’s Check code is displayed in the upper left-hand corner of the window.

### Before running Op Check

The CalPod system must be installed and configured on the PNA. The PNA must have Option 301 or 302 installed. See the **CalPod User's Guide** for instructions at: [http://cp.literature.agilent.com/litweb/pdf/85523-90001.pdf](http://cp.literature.agilent.com/litweb/pdf/85523-90001.pdf)

**Required equipment:**

- An appropriate ECal or mechanical Cal Kit.
- A high-quality cable.
- A female-female adapter of the calibration connector type.
- A fixed attenuator up to 10 dB (3 dB preferred) or other frequency insensitive device with similar loss.

### How to perform CalPod Operators Check

Click **Utilities**, then **System**, then **Service**, then **CalPod**, then **OpCheck**.

For PNA firmware version earlier than A.09.50, create a shortcut on the PNA desktop to C:\Program Files\Agilent\Network Analyzer\Service\CalPodOpChk.exe. (To create a shortcut first navigate to the desired file, right click and hold on the file name, drag to the desktop, release and select “Create Shortcuts Here”.)
Click **Setup Info** to learn more about this dialog.

Also, click **Cal Method** or **Connector** for additional explanation for these areas.

**Configure**

1. Enter information in the “Configure” area.

2. Each time a 2-port cal is performed, the results are saved in a file. The “Use Prior” selection uses the saved calibration.

3. When the calibration connector type does not mate with the CalPod connectors, perform the calibration and then use adapters to connect to the CalPod module.

4. Click **Begin** to start the Op Check.

5. Follow the prompts in the gray box.

**Op Check Results**

- The Results area shows Op Check progress.

- Click a test label for test information.

- When the check has finished, the results are saved to a text file. The default path and filename is: C:\Program Files\Agilent\Network Analyzer\Service\calpodopchklog.txt. To save multiple results, rename the file or save it to a different location.

- For assistance in troubleshooting CalPod Operator's Check failures or for additional information, see the appropriate FAQ at the CalPod web site: [http://na.tm.agilent.com/pna/calpod](http://na.tm.agilent.com/pna/calpod)

---

**Last Modified:**

15-Jun-2011 Updated per LH

5-May-2011 New topic
Delta Match Calibration

A TRL, QSOLT, or Unknown Thru Cal requires a reference receiver for each test port. The 4-port PNA-L model does NOT have a reference receiver for each test port.

A Delta Match Calibration can be thought of as a software method which provides a reference receiver for each test port when not otherwise available in the hardware. The Delta Match Calibration measures the source match and load match of the PNA test ports, and then calculates the differences, or "delta", of the two match terms. The results are then used to correct subsequent TRL, QSOLT, or Unknown Thru calibrations.

There are several ways to acquire the Delta Match Calibration:

1. **From an existing User Cal Set** that meets the following Delta Match criteria: (Not allowed for use with external test sets.)
   - Must have been performed using ECal or as a guided mechanical Cal (not Unguided).
   - Must have the same start frequency, stop frequency, and number of points as the channel being calibrated.
   - Must calibrate the ports that require the delta match terms.

2. **From a Global Delta Match Calibration**.

3. **From a 'Self Delta Match'** when other portions of the calibration fully characterize all ports using SOLT with Defined Thru or Flush Thru. For example, when calibrating all four ports of a PNA-L, perform a SOLT between ports 1 and 2, and also between ports 3 and 4, then Unknown Thru could be used between any combination of the remaining ports. This is allowed with an external test set.

**Which to use?** A Self Delta Match Cal will always be used when possible. Otherwise, the Cal Wizard will use a Global Delta Match Cal when available unless you select Choose Delta Match.

**Global Delta Match Cal**

A Global Delta Match Cal is an "all-inclusive" calibration that can be applied whenever the delta match terms are required.

A Global Delta Match Cal differs from a standard SOLT Cal in the following ways:

- It is always performed using a Flush Thru, a Known Thru, or an insertable ECal module. You can NOT use an Unknown Thru in the calibration process.

- Only two Thru connections are required to characterize the delta match terms on a 4-port PNA. This is less than the minimum number of Thrus of a standard 4-port Cal.

- Upon completion, the Global Delta Match Cal is stored as a special type of Cal Set and should be used ONLY as a Delta Match Cal. It provides Delta Match error terms, but does NOT provide all of the standard error correction terms.

- To attain the highest accuracy, the following settings are automatically used to perform a Global Delta Match Cal. When applied, it will likely be interpolated.
- Performed over the entire frequency range of the PNA.
- Uses very dense data points, particularly at low frequencies.
- Uses 100 Hz IF Bandwidth.

**Note:** For highest accuracy, perform Global Delta Match Cal using an insertable ECal module and select Flush-thru as the Calibration Thru method.

### How to perform a Global Delta Match Cal

These selections will only be available if the PNA hardware requires a Delta Match Calibration.

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[Start Cal]**
3. then **[Global Delta Match]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Cal**
3. then **Start Cal**
4. then **Global Delta Match**

### Delta Match Calibration. Select DUT Connectors and Cal Kit dialog box help

- Only one Cal Kit is specified and necessary to perform a Delta Match Cal. However, ALL of the PNA test ports are calibrated in a Delta Match Cal.

- You must configure ALL test ports to terminate in the specified connector / gender using the necessary adapters. The errors from adapters are removed during calibration, but the Thru connections must be made as specified.

- If you select an ECal module that does NOT cover the entire frequency range of the PNA your selection will change to a different Cal Kit. The Global Delta Match Cal covers the entire frequency range of the PNA. Your selected Cal Kit or ECal module must also cover the frequency range of the PNA.
Guided Calibration Steps dialog box help

Click **Measure** for each standard.
When all standards have been measured, click **Done** to complete the measurement steps.

Delta Match Calibration Complete dialog box help

Click **Finish** to store the Global Delta Match Calibration as a special type of Cal Set.
By default, it will be used when a Delta Match Calibration is required.
It should ONLY be used as a Delta Match Cal. It does NOT provide all of the standard error correction terms.

Last modified:

- 3-Sep-2008   Removed legacy content
- 9-Nov-2007   Edits for requirements
- 23-Feb-2007  Modified requirements for multiport
- 9/12/06      Added link to programming commands
Markers

Markers provide a numerical readout of measured data, a search capability for specific values, and can change stimulus settings. There are 9 regular markers and one Reference marker (used with Delta markers) available per trace. This topic discusses all aspects of markers.

Note: Marker Readout can be turned ON / OFF and customized from the View/Display menu. Learn more.

- Creating and Moving Markers
- Marker Dialog
- Searching with Markers
- Marker Functions (Change Instrument Settings)
- Marker Display
- Marker Table

Other Analyze Data topics

How to Create Markers

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press MARKER</td>
<td>1. Click Marker/Analysis</td>
</tr>
<tr>
<td>2. then [Marker n]</td>
<td>2. then Marker</td>
</tr>
<tr>
<td></td>
<td>3. select a marker number</td>
</tr>
</tbody>
</table>

Moving a Marker

To move a marker, make the marker active by selecting its number in any of the previous 3 methods. The active marker appears on the analyzer display as V. All of the other markers are inactive and are represented on the analyzer display as Δ. Then change the stimulus value using any of the following methods:

- Type a value.
- Scroll to a stimulus value using the up / down arrows. The resolution can not be changed.
- Click the stimulus box, then use the front-panel knob.
Click and Drag Markers using a finger (touchscreen) or by left-clicking and holding a marker symbol. Then drag the marker to any point on the trace. This feature is NOT allowed in Smith Chart or Polar display formats or with a Fixed Marker type.

---

**Marker dialog box help**

**Marker**  Specifies the current (active) marker number that you are defining.

**On**  Check to display the marker and corresponding data on the screen.

**Stimulus**  Specifies the X-axis value of the active marker. To change stimulus value, type a value, use the up and down arrows, click in the text box and use the front-panel knob, or drag the marker on the screen.

**Delta Marker**  Check to make the active marker display data that is relative to the reference (R) marker. There is only one reference marker per trace. All nine other markers can be regular markers or delta markers. When a delta marker is created, if not already displayed, the reference marker is displayed automatically. A delta marker can be activated from the Marker dialog box or the Marker Toolbar.

**Discrete Marker**  Check to display values at only the discrete points where data is measured. Clear to display values that are interpolated from the data points. The interpolated marker will report y-axis data from ANY frequency value between the start and stop frequency.

**Fixed**  Check to cause the marker to have a fixed X-axis and Y-axis position based on its placement on the trace when it was set to fixed. It does NOT move with trace data amplitude. It can be scrolled left and right on the X-axis by changing the marker stimulus value. Use this marker type to quickly monitor "before and after" changes to your test device. For example, you could use fixed markers to record the difference of test results before and after tuning a filter.

Clear the box to create a **Normal** marker, which has a fixed stimulus position (X-axis) and responds to changes in data amplitude (Y-axis). It can be scrolled left and right on the X-axis by changing the marker stimulus value. Use this marker type with one of the marker search types to locate the desired data.

**Coupled Markers**  Check to couple markers by marker number, 1 to 1, 2 to 2 and so forth. The markers will remain coupled until this box is unchecked. Learn more about coupled markers.

**Format**  Displays the marker data in a format that you choose. The marker format could be different from the grid format. The Trace Default setting has the marker and grid formats the same.

**All Off**  Switches OFF all markers on the active trace.

---

**Searching with Markers**

You can use markers to search and return data for the following trace criteria:
• **Max and Min**: find the highest or lowest points on the trace

• **Peak**, then move to other peaks (left, right, next highest)

• **Target Value**: find a specific Y-axis value

• **Bandwidth** (Filters)

• **Compression Point** (Amplifiers)

• **About PSAT and PNOP Markers**
  
  • **Power Saturation** (Amplifiers)
  
  • **Power Normal Operating Point** (Amplifiers)

• **What is a Peak?**

• **Search Domain**

If there is no valid data match for any of the search types, the marker will not move from its current position.

---

**How to Search with Markers**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
</table>

For **PNA-X and 'C' models**  

1. Press **SEARCH**  
2. Then **Marker Search**

---

**Marker Search** dialog box help

**Marker** Specifies the marker that you are defining. Not available for search types that deploy specific markers.

**Search Domain** Defines the area where the marker can move or search. For full span, the marker searches for specified values within the full measurement span. For user span, the marker searches for specified values within a measurement span that you define. [Learn more about Search Domain](#).

**Search Type**
Note  You must either press **Execute** or check **Tracking** to initiate all search types.

- **Execute** Click to cause the marker to search for the specified criteria.

- **Tracking** Check to cause the marker to search for the specified criteria with each new sweep. The searches begin with the first sweep after Tracking has been checked, based on the current search type and domain information. Therefore, make sure that the search criteria are in the desired state before using the data. You cannot manually change the stimulus setting for a marker if Tracking is selected for that marker.

**Maximum** Marker locates the maximum (highest) data value.

**Minimum** Marker locates the minimum (lowest) data value.

**Next Peak** Marker locates the peak with the next lower amplitude value relative to its starting position.

**Peak Right** The marker locates the next valid peak to the right of its starting position on the X-axis.

**Peak Left** The marker locates the next valid peak to the left of its starting position on the X-axis.

- **Threshold** - Minimum amplitude (dB). To be considered valid, the peak must be **above** the threshold level. The valley on either side can be below the threshold level.

- **Excursion** The vertical distance (dB) between the peak and the valleys on both sides. To be considered a peak, data values must “fall off” from the peak on both sides by the excursion value.

For more information, see **What is a Peak?**

**Target** Enter the Target value. The marker moves to the first occurrence of the Target value to the right of its current position. Subsequent presses of the Execute button cause the marker to move to the next value to the right that meets the Target value. When the marker reaches the upper end of the stimulus range, it will "wrap around" and continue the search from the lower end of the stimulus range (left side of the window).

- If **Discrete Marker** is OFF, the marker locates the interpolated data point that equals the target value.

- If **Discrete Marker** is ON and there are two data points on either side of the target value, the marker locates the data point closest to the Target value.

---

**Bandwidth Markers**

Four markers are automatically deployed to find the first negative or positive bandpass in the selected search domain.

To create Bandwidth markers:

1. Press **SEARCH**, then [Bandwidth]

2. Specify the level in dB from the peak or valley where bandwidth is measured.

- Bandwidth Search can be used ONLY with **Log Mag display format**.

- To use Bandwidth Search on a peak or valley other than the maximum or minimum values, change the
Search Domain.

Enter a **Negative** number to search for a **Peak** bandpass, such as a filter S21 response:

- Marker 1: Maximum value within the **Search Domain**.
- Marker 2: Specified level DOWN the left of the peak.
- Marker 3: Specified level DOWN the right of the peak.
- Marker 4: Center frequency between markers 2 and 3.

Enter a **Positive** number to search for a **Valley** bandpass, such as a filter S11 response:

- Marker 1: Minimum value within the **Search Domain**.
- Marker 2: Specified level UP the left of the valley.
- Marker 3: Specified level UP the right of the valley.
- Marker 4: Center frequency between markers 2 and 3.

The following four values are displayed for Bandwidth Search:

- **BW**: (Marker 3 x-axis value) - (Marker 2 x-axis value) = width of the filter.
- **Center**: Mathematical midpoint between markers 2 and 3.
- **Q**: Ratio of Center Frequency to Bandwidth (Center Frequency / Bandwidth).
- **Loss**: Y-axis value of Marker 4. This is the loss of the filter at its center frequency. The ideal filter has no loss (0 dB) in the passband.

**Compression Markers**

Uses the active marker to find the specified gain **Compression Level**. Learn more about Gain Compression.

**Note**: Valid ONLY for S21 (Gain) measurements with a **Power Sweep**.

To create Compression markers:

1. Press **SEARCH**, then **[Compression]**
2. Specify the compression level in dB,
3. Optionally press **Tracking** to search for the specified compression level with each sweep.

Linear gain is defined as the Y-axis value (gain) of the first data point of the **Search Domain** (Full Span by default).
Marker > N: X-axis value and Y-axis value

Comp Pin: Input power (marker X-axis value)
Comp Pout: Output power (Pin + gain)
Comp Level: Compression level found.

- When **Discrete** is OFF (default setting) the marker finds the exact specified compression, interpolated between the two closest data points and calculates the Comp Pin and Comp Pout value for that point.
- When Discrete is ON (not interpolated), the marker resides on the closest data point to the requested compression level.

Comp. Not Found: Displayed when the requested compression level is not found.

### About PSAT and PNOP Markers

Compression measurements based on the Pout vs Pin curves are common in the satellite test industry. In the case of Travelling Wave Tube (TWT) amplifiers, PSAT markers identify the normal operating point near saturation, and the amplifiers are operated with the power slightly backed-off approximately 0.03 to 0.1 dB. For TWT amplifiers, the saturation curve always "folds over" and produces a maximum power out.

For Solid State Power Amplifiers (SSPA), the saturation is not as well defined. A common reference is the Normal Operating Point, which is a power backed-off by 8 to 10 dB from the maximum power. In this case, the normal operating point marker replaces the Psat with the PNOP values. Also, because the backoff is important, the backoff output and input powers are displayed (PBO Out), (PBO in) as well as gain at back off (PBO Gain).

#### Power Saturation (PSAT) Markers

Uses Markers 1, 2, and 3 to quickly identify output power saturation parameters of an amplifier.

**Note:** Valid ONLY for Power IN vs Power OUT measurements. These markers can also be used on a CompOut trace with **Compression Analysis** mode in the Gain Compression Application.

To make a Power IN (X-axis) vs Power OUT (Y-axis) measurement:

1. **Preset**
2. Set **Sweep Type:** Power Sweep
3. Set Trace Meas to "B" Receiver

4. Connect DUT input to port 1

5. Connect DUT output to port 2

To create PSAT markers:

1. Press search, then [Search...]

2. From Search Type, select Power Saturation

3. For PMax Back-Off, enter the Y-axis (Power OUT) difference between the Max Power marker (3) and the Back-off marker (2).

4. Press Execute or check Tracking. Learn more.

This setting uses three markers to calculate and display 10 values.

The three markers:

- Marker 1: Linear gain; the first data point in the sweep.

- Marker 2: Specified output power Back-off from max power.

- Marker 3: Max Power output; usually the last data point.

The 10 displayed values:
Power Normal Operating Point Marker

Uses Markers 1, 2, 3, and 4 to quickly identify Normal Operating Point parameters of an amplifier.

**Note:** Valid ONLY for Power IN vs Power OUT measurements.

These markers can also be used on a CompOut trace with Compression Analysis mode in the Gain Compression Application.

See Power Saturation to learn how to make a Power IN (X-axis) vs Power OUT (Y-axis) measurement.

To create PSAT markers:

1. Press SEARCH, then [Search...]
2. From Search Type, select Normal Operating Pt
3. For Back-Off, enter the Y-axis (Power OUT) difference between the Max Power marker (3) and the Back-off marker (2).
4. For Pin Offset, enter the X-axis (Power IN) difference between Back-off marker (2) and PNOP marker (4).
5. Press Execute or check Tracking. Learn more.
This setting uses four markers to calculate and display 12 values.

The four markers:

- Marker 1: Linear gain; the first data point in the sweep.
- Marker 2: Max Output Power MINUS the specified Output (Y-axis) Back-off value in dB.
- Marker 3: Max Output Power; usually the last data point in the sweep.
- Marker 4: X-axis value of Back-off (Marker 2) plus the Pin Offset (X-axis) value in dB.

The 12 displayed values:

- >Mkr 4: Marker 4 - X-axis and Y-axis values
- Pnop Out: Marker 4 Y-axis value
- Pnop In: Marker 4 X-axis value
- Pnop Gain: Pnop Out - Pnop In
- Pnop Comp: Pnop Gain - Linear Gain*
- PMax Out: Marker 3 Y-axis value
- PMax In: Marker 3 X-axis value
- Gain Max: PMax Out - PMax In
- Comp Max: Gain Max - Linear Gain*
- PBO Out: Marker 2 Y-axis
- PBO In: Marker 2 X-axis
- PBO Gain: PBO Out - PBO In

*Linear Gain (not shown): Marker 1 - Y-axis value MINUS X-axis value

PNOP Not Found: Displayed when the requested back-off level is not found.

When Discrete marker is NOT selected (the default setting), the four markers each find an interpolated value between the two closest data points.

When Discrete marker is selected (NOT interpolated), the four markers each reside on the closest data point.
What Is a "Peak"?

You define what the analyzer considers a "peak" by selecting the following two peak criteria settings:

- **Threshold** - Minimum amplitude (dB). To be considered valid, the peak must be **above** the threshold level. The valley on either side can be below the threshold level.

- **Excursion** - The vertical distance (dB) between the peak and the valleys on both sides. To be considered a peak, data values must "fall off" from the peak on both sides by the excursion value.

**Example:**

Threshold Setting: -10dB  
Excursion Setting: 1dB  
Scale = 1 dB / Division

*Mouse over the graphic to find a valid peak.*

- **Peak A** = Valid Peak (Above Threshold and Excursion Settings)  
- **Peak B** = Invalid Peak (Below Excursion Setting)  
- **Peak C** = Invalid Peak (Below Threshold Setting)

**Search Domain**

Search domain settings restrict the stimulus values (X-axis for rectangular format) to a specified span. Set the Start and Stop stimulus settings of these **User** spans. If Start is greater than Stop, the marker will not move. Learn how to set Search Domain.

- The default domain of each new marker is "full span".
- There are 16 user-defined domains for every channel.
- The user-defined domains can overlap.
- More than one marker can use a defined domain.
- Search Domain settings are shared with Trace Statistics User Ranges.
The graphic below shows examples of search domains.

Marker Functions - Change Instrument Settings

The following settings change the relevant PNA settings to the position of the active marker.

How to change instrument settings using markers

<table>
<thead>
<tr>
<th>How to change instrument settings using markers</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Using front-panel</strong></td>
</tr>
<tr>
<td>HARDKEY [softkey] buttons</td>
</tr>
<tr>
<td>1. Press <strong>Marker</strong></td>
</tr>
<tr>
<td>2. then <strong>[Marker Function]</strong></td>
</tr>
</tbody>
</table>

[Diagram of search domains with markers labeled: User 1, User 3, User 2]
**Marker Function** dialog box help

Note: Marker Functions do not work with channels that are in CW or Segment Sweep mode.

**Marker =>Start** Sets the start sweep setting to the value of the active marker.

**Marker =>Stop** Sets the stop sweep setting to the value of the active marker.

**Marker =>Center** Sets the center of the sweep to the value of the active marker.

**Marker =>Ref Level** Sets the screen reference level to the value of the active marker.

**Marker =>Delay** The phase slope at the active marker stimulus position is used to adjust the line length to the receiver input. This effectively flattens the phase trace around the active marker. Additional Electrical Delay adjustments are required on devices without constant group delay over the measured frequency span. You can use this to measure the electrical length or deviation from linear phase.

This feature adds phase delay to a variation in phase versus frequency; therefore, it is only applicable for ratioed measurements. See Measurement Parameters.

**Marker =>Span** Sets the sweep span to the span that is defined by the delta marker and the marker that it references. Unavailable if there is no delta marker.

**Marker =>CW Freq** Sets the CW frequency to the frequency of the active marker. NOT available when the channel is in CW or Power Sweep. Use this function to first set the CW Frequency to a value that is known to be within the current calibrated range, THEN set Sweep Type to Power or CW.

**Coupled Markers**

The coupled markers feature causes markers on different traces to line up with the markers on the selected trace. Markers are coupled by marker number, 1 to 1, 2 to 2, 3 to 3, and so forth. If the x-axis domain is the same (such as frequency or time), coupling occurs across all channels, windows, and traces. Trace markers in a different x-axis domain will not be coupled. If a trace marker has no marker to couple with on the selected trace, the marker remains independent.

**Coupled Markers Model**

This model simulates the use of coupled markers in the PNA:
1. Click **Trace A** or **Trace B**

2. Click **Coupled Markers**

3. Notice the following:
   
   - Markers on the unselected trace move to the x-axis position of the selected trace.
   - If a marker number on the unselected trace has no corresponding marker on the selected trace, no movement occurs for that marker.

4. Click **Reset** to run the model again. There is no Reset for coupled markers on the PNA.

**Marker Display**

The marker display dialog allows you to change how markers and the associated readout is displayed on the PNA screen. Several marker display features also apply to **Statistics** display.

**How to change marker display settings**

Right-click on a marker readout, then click **Marker Display** or:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>MARKER</strong></td>
<td>1. Click <strong>Marker/Analysis</strong></td>
</tr>
<tr>
<td>2. then <strong>[Properties]</strong></td>
<td>2. then <strong>Marker</strong></td>
</tr>
<tr>
<td>3. then <strong>[Marker Display]</strong></td>
<td>3. then <strong>Marker Display</strong></td>
</tr>
</tbody>
</table>
The following settings apply to readouts of ALL currently-displayed marker, bandwidth, and trace statistics. These settings revert to their defaults on Preset but ARE stored with Instrument State and User Preset.

**Marker Readout**
- Checked - Shows readout information.
- Cleared - Shows NO readout information.

**Large Readout**
- Checked - Shows the marker readout in large font size for easy reading. However, all readout lines may not be visible.
- Cleared - Shows the marker readout in normal font size.

**Readouts Per Trace** Choose the quantity of marker readouts to show in the window for each trace. Choose to display up to 10 readouts per trace, up to 20 readouts per window. When more markers are present than the specified quantity of readouts, the marker numbers for which readouts are displayed can change depending on the marker number that is active. Readouts Per Trace can be set independently for each window.

**Symbol** - Choose from the following marker symbols.

- Triangle
- Flag
- Line

Line symbols are NOT used on Smith or Polar display formats.

Symbols can be set independently for each window.

**Decimal Places** Choose the marker readout resolution to display. These values also apply to the readouts that are displayed in the marker table. Decimal Places can be set independently for each window.
- Stimulus (X-axis) - Choose from 2 to 6 places after the decimal point. Default is 3.
- Response (Y-axis) - choose from 1 to 4 places after the decimal point. Default is 2.
**Readout Position** Choose where to place the marker readouts. Marker readouts are right-justified on the specified X-axis and Y-axis position. The default position (10.0, 10.0) is the upper-right corner of the grid. Position (1.0,1.0) is the lower-left corner. Readout position can be set independently for each window.

**Note:** Readout Position can also be changed using a mouse by left-clicking on the top readout and dragging to the new position.

**Marker Colors** Starts the Display Colors dialog with only the marker colors available. [Learn more.](#)

---

**Marker Table**

You can display a table that provides a summary of marker data for the active trace. The marker data is displayed in the specified format for each marker.

**How to view the Marker Table**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **DISPLAY**
2. then **[More]**
3. then **[Tables]**
4. then **[Marker Table]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Display**
3. then **Tables**
4. then **Marker Table**

---

Last Modified:

- 7-Jun-2011  Linked Search domain to User Range
- 5-Aug-2010  Added marker display
- 9-Feb-2010  Added PSAT, PNOP, and new dialog
- 6-Feb-2009  Added compression marker
- 3-Oct-2008  Added Marker->CW function
- 3-Sep-2008  Removed legacy content
- 4-Jan-2008  Added bookmark to move marker
- 17-Jul-2007  Clarified bandwidth search
- 2-Feb-2007  MX Added UI
Using Math / Memory Operations

You can perform four types of math on the active trace versus a memory trace. In addition three statistics (Mean, Standard Deviation and Peak to Peak) can be calculated and displayed for the active data trace.

- **Trace Math**
- **Trace Statistics**

Note: Trace Math (described here) allows you to quickly apply one of four math operations using memory traces. Equation Editor allows you to build custom equations using several types of traces from the same, or different channels.

Other Analyze Data topics

Trace Math

To perform any of the math operations, you must first store a trace to memory. You can display the memory trace using the View options.

Trace math is performed on the complex data before it is formatted for display. See the PNA data processing map. Markers can be used while viewing a memory trace.

How to select Trace Math

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press MEMORY</td>
<td>1. Click Marker/Analysis</td>
</tr>
<tr>
<td></td>
<td>2. then Memory</td>
</tr>
<tr>
<td></td>
<td>3. then Memory</td>
</tr>
</tbody>
</table>

Normalize, available only from the Memory menu, (not on the Math / Memory dialog), performs the same function as Data=>Memory, then Data / Memory.

Math / Memory dialog box help
Normalize, available only from the Memory menu, (not on the Math / Memory dialog), performs the same function as Data=>Memory, then Data / Memory.

Data=>Memory  Puts the active data trace into memory. You can store one memory trace for every displayed trace.

Note: Many PNA features are NOT allowed on Memory traces. For example, Memory traces can NOT be saved to any file type (PRN, SNP, CTI, CSV, MDF). However, you can restore a memory trace to a data trace using the Memory-to-Data utility at the http://na.tm.agilent.com/pna/apps/applications.htm website.

Data Math
All math operations are performed on linear (real and imaginary) data before being formatted. See the PNA Data flow (below).

Data (or OFF)  Does no mathematical operation.

Data / Memory - Current measurement data is divided by the data in memory. Use for ratio comparison of two traces, such as measurements of gain or attenuation. Learn more.

Data – Memory - Data in memory is subtracted from the current measurement data. For example, you can use this feature for storing a measured vector error, then subtracting this error from the DUT measurement. Learn more.

Data + Memory - Current measurement data is added to the data in memory. Learn more.

Data * Memory - Current measurement data is multiplied by the data in memory. Learn more.

8510 Mode - Learn more.

Trace View Options

Data Trace  Displays ONLY the Data trace (with selected math operation applied).

Memory Trace  Displays ONLY the trace that was put in memory.

Data and Memory Trace  Displays BOTH the Data trace (with selected math operation applied), and the trace that was put in memory.

Learn more about Trace Math (scroll up)
(Data / Memory) and (Data - Memory)

(Data / Memory) and (Data - Memory) math operations are performed on linear data before it is formatted. Because data is often viewed in log format, it is not always clear which of the two math operations should be used. Remember: dividing linear data is the same as subtracting logarithmic data. The following illustrates, in general, when to use each operation.

Use Data / Memory for normalization purposes, such as when comparing S21 traces “before” and “after” a change is made or measurement of trace noise. In the following table, the Data/Mem values intuitively show the differences between traces. It is not obvious what Data-Mem is displaying.

<table>
<thead>
<tr>
<th>S21 values to compare</th>
<th>Data/Mem</th>
<th>Data-Mem</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5 dB and 0.6 dB</td>
<td>0.1 dB</td>
<td>-39 dB</td>
</tr>
<tr>
<td>0.5 dB and 0.7 dB</td>
<td>0.2 dB</td>
<td>-33 dB</td>
</tr>
</tbody>
</table>

Use Data - Memory to show the relative differences between two signals. Use for comparison of very small signals, such as the S11 match of two connectors.

In the following table, Data/Mem shows both pairs of connectors to have the same 2 dB difference. However, the second pair of connectors have much better S11 performance (-50 and -52) and the relative significance is shown in the Data-Mem values.

<table>
<thead>
<tr>
<th>S11 values to compare</th>
<th>Data/Mem</th>
<th>Data-Mem</th>
</tr>
</thead>
<tbody>
<tr>
<td>-10 dB and -12 dB</td>
<td>2 dB</td>
<td>-24 dB</td>
</tr>
<tr>
<td>-50 dB and -52 dB</td>
<td>2 dB</td>
<td>-64 dB</td>
</tr>
</tbody>
</table>

Data * Memory and Data + Memory

Use Data * Memory and Data + Memory to perform math on an active data trace using data from your own formulas or algorithms rather than data from a measurement. For example, if you want to simulate the gain of a theoretical amplifier placed in series before the DUT, you could do the following:

1. Create an algorithm that would characterize the frequency response of the theoretical amplifier.

2. Enter complex data pairs that correspond to the number of data points for your data trace.

3. Load the data pairs into memory with SCPI or COM commands. The analyzer maps the complex pairs to correspond to the stimulus values at the actual measurement points.

4. Use the data + memory or data * memory function to add or multiply the frequency response data to the measured data from the active data trace.

Note: The data trace must be configured before you attempt to load the memory.

Trace Statistics
You can calculate and display statistics for the active data trace. These statistics are:

- Mean
- Standard deviation
- Peak-to-peak values

You can calculate statistics for the full stimulus span or for part of it by using User Ranges.
You can define up to 16 user ranges per channel. These user ranges are the same as the Search Domain specified for a marker search in that same channel. They use the same memory registers and thus share the same stimulus spans.
The user ranges for a channel can overlap each other.
A convenient use for trace statistics is to find the peak-to-peak value of passband ripple without searching separately for the minimum and maximum values.
The trace statistics are calculated based on the format used to display the data.

- **Rectangular data formats** are calculated from the scalar data represented in the display
- **Polar** or **Smith Chart** formats are calculated from the data as it would be displayed in Log Mag format

See how to make Trace Statistics display settings.

### How to activate Trace Statistics

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>ANALYSIS</strong></td>
<td>1. Click <strong>Marker/Analysis</strong></td>
</tr>
<tr>
<td>2. then <strong>[Statistics]</strong></td>
<td>2. then <strong>Analysis</strong></td>
</tr>
<tr>
<td>3. then <strong>[Trace Statistics]</strong></td>
<td>3. then <strong>Trace Statistics</strong></td>
</tr>
</tbody>
</table>

[Programming Commands]
Trace Statistics dialog box help

See how to make Trace Statistics display settings.

Statistics  Check to display mean, standard deviation, and peak to peak values for the active trace.

Span  Specifies the span of the active trace where data is collected for a math operation. You can select Full Span, or define up to 16 user spans per channel with Start and Stop. You can also define the user spans from the Search Domain selector on the Marker Search dialog box.

Start  Defines the start of a user span.

Stop  Defines the stop of a user span.

Learn more about Trace Statistics (scroll up)

Last Modified:

7-Jun-2011  Moved 8510 mode to separate topic
3-Sep-2008  Removed legacy content
28-Aug-2008  Added Memory trace utility note
22-Apr-2008  Added 8510 preference link
27-Aug-2007  Edited trace display settings
2-Feb-2007  MX added UI
Equation Editor

Equation Editor allows you to enter an algebraic equation that can mathematically manipulate measured data. The results are displayed as a data trace. Data that is used in the equation can be from the same or different channels.

**Note:** Equation Editor is now available with ALL PNA Apps.

See Using Noise Power Traces in Equation Editor

- Overview
- How to start Equation Editor
- Using Equation Editor
- Data that is used in Equation Editor
- Trace Settings, Error Correction, and an Example
- Functions and Constants
- Operators
- Example Equations
- Saving Equation Editor Data

**See Also**

Equation Editor Import Functions - create custom functions for use in Equation Editor.

---

**Other 'Analyze Data' topics**

---

**Overview**

Equation Editor allows you to enter an algebraic equation of standard mathematical operators and functions, referencing data that is available in the PNA. Once a valid equation is entered and enabled, the display of the active trace is replaced with the results of the equation, and updated in real-time as new data is acquired. For equations that can be expressed with Equation Editor's supported functions, operators, and data, there is no need for off-line processing in a separate program.

For example, enter the equation S21 / (1 - S11). The resulting trace is computed as each S21 data point divided by one minus the corresponding S11 data point. For a 201 point sweep setup, the computation is repeated 201 times, once for each point.

As another example, suppose you want the PNA to make a directivity measurement of your 3-port DUT. This is not a native PNA measurement, but can be achieved using the Equation Editor. The desired result is the sum and difference of LogMag formatted traces, expressed as: S12 + S23 - S13.

Because Equation Editor operates on unformatted complex data, the required equation is:

\[ \text{DIR} = \frac{S12 \times S23}{S13} \]

DIR becomes a display label to help you identify the computed data trace.
On the equation trace, set the format to LogMag.

### How to start Equation Editor

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>ANALYSIS</strong></td>
<td>1. Click <strong>Marker/Analysis</strong></td>
</tr>
<tr>
<td>2. then <strong>[Equation Editor]</strong></td>
<td>2. then <strong>Analysis</strong></td>
</tr>
<tr>
<td></td>
<td>3. then <strong>Equation Editor</strong></td>
</tr>
</tbody>
</table>

![Equation Editor dialog box help](image)

**Equation Editor dialog box help**

**Notes**
- **Double-click**, or type, the Functions, Operators, and Data to build an Equation.
- Scroll down to learn more about Using Equation Editor

**Equation:** The field in which equations are built. Click the down arrow to the right to use or modify equations that have been previously saved. This is where equations are saved when you press ‘Store Equation’.

**Enabled** Check this box to enable the equation that is currently in the Equation field. If the Enabled box is not available, then the equation is not valid. If a data trace is used that is from a different channel than the Equation trace, the channels MUST have the same number of data points to be valid.

**<-Backspace** Moves the cursor to the left while erasing characters.

**<-** Moves the cursor to the left without erasing characters.

**->** Moves the cursor to the right without erasing characters.
**Store Equation**  Press to save the current equation. To later recall the equation, click the down arrow to the right of the equation.

**Delete Equation**  Removes the current equation from the drop-down list.

**Functions/Constants:**  See descriptions of Functions.

Select the "library" of functions to view. The "built-in" library appears by default which includes the standard functions of equation editor. Other functions that can appear here are functions that you have written and imported.  Learn more.

**Operators:**  See descriptions of Operators.

**Trace Data:**  Select from ALL of the currently displayed traces on ALL channels.

**Ch Param Data:**  Select from undisplayed data that is available ONLY from the active channel (same channel as the equation trace).

**Note:**  With an external test set enabled, only parameters involving ports 1 through 4 are listed. However, all available parameters can be typed directly into the **Equation** field.

See Data that is used in Equations.

**Keypad:**  Provided to allow navigation of the entire dialog with a mouse.

**Import Functions**  Click to launch the Import Functions Dialog box.

### Using Equation Editor

1. **Pick a trace in which to enter the equation**
   
   - Equation Editor works on the active trace.
   
   - Either create a new trace, or click the **Trace Status** button on an existing trace to make the trace active.

2. **Enter an equation**

   Start Equation Editor See how.

   - The equation text can be in the form of an expression \( \frac{S21}{1-S11} \) or an equation \( \text{DIR} = S12 * S23 / S13 \).
     This topic refers to both types as equations.
   
   - Either type, or double-click the Functions, Operators, and Data to build an equation.
   
   - Functions and Constants ARE case-sensitive; Data names are NOT case sensitive.
   
   - Learn more about referring to data traces.

3. **Check for a valid equation**

   When a valid equation is entered, the Enabled checkbox becomes available for checking. When the Enabled box is checked:

   - The Equation Trace becomes computed data.
The equation is visible on the **Trace Status** (up to about 10 characters).

The equation is visible in the trace **Title** area (up to about 45 characters) when the Equation trace is active.

The equation is visible in the **Status Bar** at the bottom of the display. This is updated only after the equation is entered and the **Trace Status** button is clicked.

If an equation is NOT valid, and a trace from a different channel is used, make sure the number of data points is the same for both channels.

Learn more about the **Functions**, **Operators**, and **Data** that are used in Equation Editor.

**Data that is used in Equation Editor**

**Definitions**

- **Equation trace**  A trace in which an equation resides.
- **Referred trace**  A trace that is used as data in an equation.

  **Example:** \( eq = Tr2 + S11 \) is entered into **Tr1**.
  
  **Tr1** becomes an equation trace.
  
  **Tr2** and **S11** are both referred traces because they are used in the equation trace.

**Notes**

- Referred traces are processed one data point at a time. For example, the expression \( S11/S21 \) means that for each data point in **S11** and **S21**, divide point N of **S11** by point N of **S21**.
- Once an equation is enabled, the trace is no longer identified by its original measurement parameter. It becomes an equation trace.
- An equation trace can NOT refer to itself. For example, an equation in **Tr1** cannot refer to trace **Tr1**.
- Referred traces can be selected from S-Parameters, Receiver data, and **Memory traces**.

  **See note regarding External Test Sets.**

  **See** Using Noise Power Traces in Equation Editor

**There are three ways to refer to traces:**

The following distinction is important when discussing the three ways to refer to traces/data.

- **Trace** - a sequential collection of data points that are displayed on the PNA screen.
- **Data** - PNA measurements that are acquired but not displayed. When an equation trace refers to data that is not displayed, the PNA will automatically acquire the data.
1. Using **TrX** Trace notation (for example, Tr2).

When a trace is created, check "Show Tr Annotation" to see the Tr number of that trace.

- **Simple** - ALWAYS refers to displayed traces.
- Must be used for referring to traces in a different channel as the equation trace.
- All trace settings are preserved in the equation trace. If you do NOT want a trace setting to be used in the equation trace, you must disable it in the referred trace.
- If the referred trace is error corrected, then that data is corrected in the equation trace.
- Used to refer to a memory trace (it must already be stored in memory). Append .MEM to the TrX trace identifier. For example, *Tr2.mem* refers to the memory trace that is stored for Tr2.

2. Using **S-parameter** notation (for example, S11/S21)

- **Convenient** - ALWAYS refers to data that is NOT displayed.
- Refers to data that resides in the same channel as the equation.
- NOT the same as referring to a displayed S11 trace using TrX notation. See Example.

  - The referred data includes NO trace settings.
  - If the channel has error correction available, then it can be applied by turning error correction ON for the Equation trace.

3. Using **Receiver** notation (for example AB_2); NOT case sensitive.

At least one receiver is required, followed by an underscore and a number.

- The **letters** before the underscore refer to the receivers.
  - Letters alone refer to physical receivers.
  - Letters immediately followed by numbers refer to logical receivers. Learn more.
  - If two receivers are referenced, they are ratioed.
- The **number** after the underscore refers to the source port for the measurement.

Examples

- `AR1_2` = physical receiver A / physical receiver R1 with 2 as the source port.
- `a3b4_1` = reference receiver for port 3 / test port receiver for port 4 with 1 as the source port.

Learn more about ratioed and unratioed receiver measurements.

Receiver notation is like S-parameter notation in that:
• Refers to data that is NOT displayed and resides in the same channel as the equation.

• The referred data includes NO trace settings.

• If the channel has error correction available for that receiver, then it can be applied by turning error correction ON for the Equation trace.

**Referring to Traces in a different channel**

When the equation trace refers to a trace on a different channel:

• The trace must already be displayed.

• Must refer to the trace using **TrX** notation.

• The Equation trace and the referred trace MUST have the same number of data points or the Enable checkbox will not be available.

• The Equation trace is updated when the last referred data in the same channel is acquired. Therefore, to prevent 'stale' data from being used, the Equation trace must be on a higher numbered channel than the referred trace. This is because the PNA acquires data in ascending channel number order - first channel 1, then channel 2, and so forth. If the Equation trace is on channel 1, and it refers to a trace on channel 2, the Equation trace will update after channel 1 is finished sweeping, using 'old' data for the channel 2 trace.

**Port Extensions and Equation Editor**

When using port extension with an equation, turn Fixturing ON to ensure that the underlying parameters have port extension properly applied. Learn more.

**Trace Settings, Error Correction, and an Example**

This discussion highlights the differences between using **S-parameter / Receiver** notation and **TrX** notation when referring to traces. The key to understanding the differences is realizing that **S-parameter / Receiver** notation ALWAYS refers to data that is NOT displayed.

- **Trace Settings** Normalization, Trace Math, Gating, Phase and Mag Offset, Electrical Delay, Time Domain.
- **Equation Editor** processing occurs on the *equation trace* immediately after error correction.

- **Referred Data/Trace** (used in the equation) is taken from the following locations:
  - When using **TrX** notation, data is taken immediately before formatting. These traces are always displayed and include **Trace Settings**.
  - When using **S-parameter / Receiver** notation, data is taken immediately after error correction. This data is NOT displayed and includes **NO** trace settings (see example).

See Equation Editor Notes at [GetData Method](#) or [GetDataByString Method](#).

**Error-correction and Equation Editor**

Using **TrX** notation:

- The Trace Settings and Error-correction on the referred trace are used in the Equation trace.
- If error correction is **NOT** ON, then the raw, uncorrected data is used in the equation trace.
- To see if error correction is ON, make the trace active, then see the [Correction level in the status bar](#).
- Turning error correction ON/OFF on the equation trace has no meaning. The referred data that is used in the equation is **ALWAYS** what determines its level of correction.

Using **S-parameter** and **Receiver** notation:

- Because the data is not displayed, NO trace settings are used in the Equation trace.
- Correction can be turned ON/OFF if corrected data is available for the referred data. Exception: When using **S-parameter** and **Receiver** notation to refer to a trace on a channel that has been calibrated with a **Response Cal** or **Receiver Cal**, correction can **NOT** be turned ON, even though the Status Bar indicates otherwise. For example: Tr1 is an S11 measurement with a Response Cal. Tr2 is an equation trace that refers to S11. The Tr2 equation trace is **NOT** corrected, even though the Status Bar may indicate that it is corrected. However, if Tr2 refers to Tr1 (not S11), the Tr2 equation trace is corrected.

**Example**

This example illustrates the differences when referring to a trace using **S-parameter** notation and **TrX** notation:
• **Tr1** is an S11 measurement with no equation, 2-port correction ON, and Time Domain transform ON.

• **Tr2** is an equation trace that refers to **Tr1**. Tr2 is corrected because Tr1 is corrected. Tr2 is transformed because Tr1 is transformed. If transform is turned ON for Tr2, the data will be transformed AGAIN, which results in "unusual" data.

• **Tr3** is an equation trace that refers to **S11**. This is NOT the same as referring to Tr1. The S11 trace that is referred to is a different instance of S11 that is NOT displayed, and has NO trace settings. Notice that Tr3 data is NOT transformed, although Tr1 is transformed. Correction for **Tr3** can be turned ON and OFF because a calibration was performed on the channel in which the S11 trace resides.

• **Note**: X-axis annotation of the Equation trace is completely independent of the data that is presented. ONLY the data values from a referred trace are used. For example, notice that the Equation trace **Tr2** has Frequency on the X-axis although the referred trace **Tr1** is presented in Time.

### Functions and Constants used in Equation Editor

ALL trace data that is used in Equation Editor is unformatted, complex data.

In the following table,

- Function(scalar x) means that an automatic conversion from a complex number to its scalar magnitude is performed before passing the value to the function.

- Function(complex x) means that the entire complex value is used.

- **a, b, c, d** are arguments that are used in the function.

<table>
<thead>
<tr>
<th>Function/Constant</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>acos(scalar a)</td>
<td>returns the arc cosine of a in radians</td>
</tr>
<tr>
<td>asin(scalar a)</td>
<td>returns the arc sine of a in radians</td>
</tr>
<tr>
<td>atan(scalar a)</td>
<td>returns the arc tangent of a in radians</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>atan2</td>
<td>returns the phase of complex a = (re,im) in radians</td>
</tr>
<tr>
<td>has the following two argument sets:</td>
<td></td>
</tr>
<tr>
<td>- atan2(complex a) - returns the phase in radians</td>
<td></td>
</tr>
<tr>
<td>- atan2(scalar a, scalar b)</td>
<td></td>
</tr>
<tr>
<td>conj(complex a)</td>
<td>takes a and returns the complex conjugate</td>
</tr>
<tr>
<td>cos(complex a)</td>
<td>takes a in radians and returns the cosine</td>
</tr>
<tr>
<td>cpx(scalar a, scalar b)</td>
<td>returns a complex value (a+b) from two scalar values</td>
</tr>
<tr>
<td>e</td>
<td>returns the constant =~ 2.71828...</td>
</tr>
<tr>
<td>exp(complex a)</td>
<td>returns the exponential of a</td>
</tr>
<tr>
<td>getNumPoints()</td>
<td>returns the number of points for the current sweep</td>
</tr>
<tr>
<td>im(complex a)</td>
<td>returns the imag part of a as the scalar part of the result (zeroes the imag part)</td>
</tr>
<tr>
<td>kfac(complex a, complex b, complex c, complex d)</td>
<td>k-factor:</td>
</tr>
<tr>
<td>when entered in EE: kfac(S11,S21,S12,S22)</td>
<td>k = (1 -</td>
</tr>
<tr>
<td>returns a scalar result - the imaginary part of the complex result is always 0</td>
<td></td>
</tr>
<tr>
<td>ln(complex a)</td>
<td>returns the natural logarithm of a</td>
</tr>
<tr>
<td>log10(complex a)</td>
<td>returns the base 10 logarithm of a</td>
</tr>
<tr>
<td>mag(complex a)</td>
<td>returns sqrt(a.re<em>a.re+a.im</em>a.im)</td>
</tr>
<tr>
<td>max(complex a, complex b, ...)</td>
<td>returns the complex value that has the largest magnitude of a list of values.</td>
</tr>
<tr>
<td>max_hold(complex a)</td>
<td>New - holds the current maximums of the sweep. Disable the equation to reset. See example</td>
</tr>
<tr>
<td>median(complex a, complex b,...)</td>
<td>returns the median of a list of complex values</td>
</tr>
<tr>
<td>- The median is determined by sorting the values by magnitude, and returning the middle one.</td>
<td></td>
</tr>
<tr>
<td>- If an even number of values is passed, then the smaller of the two middle values is returned.</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>-------------</td>
</tr>
<tr>
<td>min(complex a, complex b, ...)</td>
<td>returns the complex value that has the smallest magnitude of a list of values.</td>
</tr>
<tr>
<td>min_hold(complex a)</td>
<td>New - holds the current minimums of the sweep. Disable the equation to reset. See example</td>
</tr>
<tr>
<td>mu1(complex a, complex b, complex c, complex d)</td>
<td>mu1 = (1 -</td>
</tr>
<tr>
<td>when entered in EE: mu1(S11,S21,S12,S22)</td>
<td></td>
</tr>
<tr>
<td>mu2(complex a, complex b, complex c, complex d)</td>
<td>mu2 = (1 -</td>
</tr>
<tr>
<td>when entered in EE: mu1(S11,S21,S12,S22)</td>
<td></td>
</tr>
<tr>
<td>for both mu1 and mu2 (Usually written with the Greek character µ)</td>
<td>• conj is the complex conjugate. For scalars a and b, conj(a+ib) = (a-ib)</td>
</tr>
<tr>
<td></td>
<td>• returns a scalar result - the imaginary part of the complex result is always 0</td>
</tr>
<tr>
<td>phase(complex a)</td>
<td>returns atan2(a) in degrees</td>
</tr>
<tr>
<td>PI</td>
<td>returns the numeric constant pi (3.141592), which is the ratio of the circumference of a circle to its diameter</td>
</tr>
<tr>
<td>pow(complex a, complex b)</td>
<td>returns a to the power b</td>
</tr>
<tr>
<td>re(complex a)</td>
<td>returns the scalar part of a (zeroes the imag part)</td>
</tr>
<tr>
<td>sin(complex a)</td>
<td>takes a in radians and returns the sine</td>
</tr>
<tr>
<td>sqrt(complex a)</td>
<td>returns the square root of a, with phase angle in the half-open interval (-pi/2, pi/2]</td>
</tr>
<tr>
<td>tan(complex a)</td>
<td>takes a in radians and returns the tangent</td>
</tr>
<tr>
<td>traceDataArray(complex a)</td>
<td>returns the entire set of points from a sweep. Function is intended to be used as an argument in a custom function to allow access for data array processing.</td>
</tr>
<tr>
<td>xAxisArray()</td>
<td>New returns the x-axis values for the entire sweep.</td>
</tr>
<tr>
<td>xAxisIndex()</td>
<td>returns the current index in the sweep.</td>
</tr>
<tr>
<td>xAxisValue()</td>
<td>returns the current value of the x-axis index.</td>
</tr>
</tbody>
</table>
Operators used in Equation Editor

<table>
<thead>
<tr>
<th>Operator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Addition</td>
</tr>
<tr>
<td>-</td>
<td>Subtraction</td>
</tr>
<tr>
<td>*</td>
<td>Multiplication</td>
</tr>
<tr>
<td>/</td>
<td>Division</td>
</tr>
<tr>
<td>(</td>
<td>Open parenthesis</td>
</tr>
<tr>
<td>)</td>
<td>Close parenthesis</td>
</tr>
<tr>
<td>,</td>
<td>Comma - separator for arguments (as in S11, S22)</td>
</tr>
<tr>
<td>=</td>
<td>Equal (optional)</td>
</tr>
<tr>
<td>E</td>
<td>Exponent (as in 23.45E6)</td>
</tr>
</tbody>
</table>

Example Equations

The following examples may help you get started with Equation Editor.

Offset each data point in Tr2 from Tr1 by 2dB

Use the function: pow(complex \(a\), complex \(b\)) -- returns \(a\) to the power \(b\).

\[
\begin{align*}
20\log(a) + 2 &= 20\log(x) \\
\log(a) + 2/20 &= \log(x) \quad \text{// divide all by 20.} \\
x &= 10^{\log(a) + 2/20} \quad \text{// swap sides and take 10 to the power of both sides} \\
x &= 10^{\log(a) \times 10^{(2/20)}} \\
x &= a \times 10^{(2/20)}
\end{align*}
\]

The equation is entered into Tr2 as:

\[
\text{Offset=Tr1*\text{pow}(10, 2/20)}
\]

To offset by 5 dB

\[
\text{Offset=Tr1*\text{pow}(10, 5/20)}.
\]

Balanced Match using a 2-port PNA

\[
S_{DD11} = \frac{(S_{11} - S_{21} - S_{12} + S_{22})}{2}
\]

Conversion loss

\[
B_1/\text{pow}(10, -15/20)
\]

- \(B_1\) is a receiver measurement;
-15 is the input power in dBm

**Third-order intercept point (IP3 or TOI)**

$$ TR1 \cdot \sqrt{Tr1/Tr3} $$

- $Tr1$ = input signal power
- $Tr3$ = intermodulation power (both traces measured with single receivers)

**Harmonics in dBc**

$$ B_1/Tr2 $$

- $B_1$ is tuned to a harmonic frequency
- $Tr2$ = power at fundamental frequency, measured with $B_1$ receiver

**PAE (Power Added Efficiency)**

\[ PAE = 100 \times \frac{(.001 \times \text{pow(mag(Tr1),2)}) - (.001 \times \text{pow(mag(Tr1),2)}/\text{pow(mag(Tr2),2)})}{(Tr3 \times Tr4)} \]

Where:

- $Tr1$ - a trace that measures unratioed B receiver.
- $Tr2$ - a corrected S21 trace (amplifier gain)
- $Tr3$ - a trace that measures ADC voltage (A11) across a sensing resistor.
- $Tr4$ = an equation trace containing Isupp = (Tr3 / value of sensing resistor).

Data is displayed in Real format with units actually being watts.

**1-port Insertion Loss**

When it is not possible to connect both ends of a cable to the PNA, a 1-port insertion loss measurement can be made. However, the measured loss must be divided by 2 because the result includes the loss going down and coming back through the cable. This assumes that the device is terminated with a short or open to reflect all of the power. The 'divide by 2' operation (for dB) is performed as follows using Equation Editor:

- $Tr1$ - an S11 trace in log mag format.
- $Tr2$ - an equation trace containing $\sqrt{Tr1}$

**Max and Min Hold**

These two functions allow you to capture and display either the Maximum or Minimum values for each data point.
over multiple sweeps.

**Maxhold (S21)** - displays the maximum value for each data point until reset. Reset by disabling, then enabling the equation. This example refers to an S21 trace that is not displayed.

### Saving Equation Editor Data

Equation data can be saved to the PNA hard drive in the following formats:

- **Citifile (.cti)** - Equation data is saved and recalled. The file header indicates the "underlying" s-parameter trace type.
- **PRN** - read by Spreadsheet software. Can NOT be recalled by the PNA.
- **CSV** - read by Spreadsheet software. Can NOT be recalled by the PNA.
- **MDIF** - compatible with Agilent ADS (Advanced Design System). Can NOT be recalled by the PNA.
- **Print to File** (bmp, jpg, png) - saves image of PNA screen.

Equation data can NOT be saved in .SnP file format. When attempting to save an Equation trace in .SnP format, the "underlying" S-parameter data is saved; not Equation data.

Last Modified:

- 6-Apr-2011  Replaced 1-port example with JV ex.
- 5-Nov-2010  Edited TR annotation
- 20-Sep-2010  Added file types to save data
- 2-Sep-10  Added link to NF
- 11-Jun-2009  Added link to GetData
- 3-Sep-2008  Removed legacy content
- 22-Aug-2008  Added xAxisArray() Maxhold() and Minhold() function
- 6-May-2008  Added Import functions
- 3-Jul-2007  Added PAE and other notes
- 18-Jun-2007  Added examples
Using Limit Lines

Limit lines allow you to compare measurement data to performance constraints that you define.

- **Overview**
- **Create and Edit Limit Lines**
- **Display and Test with Limit Lines**
- **Testing with Sufficient Data Points**

**Other Analyze Data topics**

**Overview**

Limit lines are visual representations on the PNA screen of the specified limits for a measurement. You can use limit lines to do the following:

- Give the operator **visual guides** when tuning devices.
- Provide **standard criteria** for meeting device specification.
- Show the **comparison** of data versus specifications.

Limit testing compares the measured data with defined limits, and provides optional **Pass or Fail** information for each measured data point.

You can have up to **100** discrete lines for each measurement trace allowing you to test all aspects of your DUT response.

Limit lines and limit testing are NOT available with **Smith Chart** or **Polar** display format. If limit lines are ON and you change to Smith Chart or Polar format, the analyzer will automatically disable the limit lines and limit testing.

**Create and Edit Limit Lines**

You can create limit lines for all measurement traces. The limit lines are the same color as the measurement trace.

Limit lines are made up of discrete lines with four coordinates:

- BEGIN and END stimulus - X-axis values.
- BEGIN and END response - Y-axis values.
How to create, edit, and test with Limit Lines

All limit line settings are made with the limit table. Use one of the following methods to show the limit table:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Analysis</strong></td>
<td>1. Click <strong>Marker/Analysis</strong></td>
</tr>
<tr>
<td>2. then <strong>Limits</strong></td>
<td>2. then <strong>Analysis</strong></td>
</tr>
<tr>
<td>3. then <strong>Limit Test</strong></td>
<td>3. then <strong>Limit Test</strong></td>
</tr>
</tbody>
</table>

Limit Table

<table>
<thead>
<tr>
<th>Type</th>
<th>BEGIN STIMULUS</th>
<th>END STIMULUS</th>
<th>BEGIN RESPONSE</th>
<th>END RESPONSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>MIN</td>
<td>1.90000 MHz</td>
<td>5.00000 dB</td>
<td>0.00000 dB</td>
</tr>
<tr>
<td>2</td>
<td>MAX</td>
<td>1.90000 MHz</td>
<td>60.00000 dB</td>
<td>0.00000 dB</td>
</tr>
<tr>
<td>3</td>
<td>MAX</td>
<td>2.00000 MHz</td>
<td>5.00000 dB</td>
<td>0.00000 dB</td>
</tr>
<tr>
<td>4</td>
<td>OFF</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** To ADD a limit line to the table, change the last limit line to either MAX or MIN.

1. In the **Type** area of the Limit Table, select **MIN** or **MAX** for Limit Line 1.
   - The **MIN** value will fail measurements BELOW this limit.
   - The **MAX** value will fail measurements ABOVE this limit.

2. Click **BEGIN STIMULUS** for Limit Segment 1. Enter the desired value.

3. Click **END STIMULUS** for Limit Segment 1. Enter the desired value.

4. Click **BEGIN RESPONSE** for Limit Segment 1. Enter the desired value.

5. Click **END RESPONSE** for Limit Segment 1. Enter the desired value.

6. Repeat Steps 1-5 for each desired limit line.

Displaying and Testing with Limit Lines

After creating limit lines, you can then choose to **display** or **hide** them for each trace. The specified limits remain valid even if limit lines are not displayed.

Limit testing cannot be performed on memory traces.

You can choose to provide a visual and / or audible **PASS / FAIL** indication.

With limit testing turned ON:

- Any portion of the measurement trace that **fails** is **displayed in red**.
Any portion of the measurement trace that does **NOT fail** remains unchanged and silent.

**Display failed trace points or trace segments**

Beginning with PNA Rev. A.09.00, the new default behavior is to display the data points that fail limit line testing as red dots. This can be changed to the old default behavior (the failed limit line segment is colored red) with a Preference setting. Learn how.

![New default behavior vs Old default behavior](image)

**PASS is the default mode of Pass / Fail testing.**

A data point will **FAIL** only if a measured point falls outside of the limits.

- If the limit line is set to OFF, the entire trace will **PASS**.
- If there is no measured data point at a limit line stimulus setting, that point will **PASS**.
Limit Test dialog box help

**Show Table**  Shows the table that allows you to create and edit limits.

**Hide Table**  Makes the limits table disappear from the screen.

**Note:** To ADD a limit line to the table, change the last limit line to either MAX or MIN

**Limit Test**

- **Limit Test ON**  Check the box to compare the data trace to the limits and display PASS or FAIL.
- **Limit Line ON**  Check the box to make the limits visible on the screen. (Testing still occurs if the limits are not visible.)
- **Sound ON Fail**  Check the box to make the PNA beep when a point on the data trace fails the limit test.

**Global Pass/Fail**

The Pass/Fail indicator provides an easy way to monitor the status of ALL measurements.

- **Global pass/fail display ON**  Check to display the Global Pass/Fail status.

**Policy:**  Choose which of the following must occur for the Global Pass/Fail status to display PASS:

- All Tests (with Limit Test ON) Must Pass - This setting reads the results from the Limit Tests. If all tests (with Limit Test ON) PASS, then the Global Pass/Fail status will PASS.

- All Measurements Must Pass -  This more critical setting shows FAIL unless all measured data points fall within established test limits and Limit Test is ON. **Note:** In this mode, if one measurement does NOT have Limit Test ON, Global Pass/Fail will show FAIL.

Learn more about displaying and testing with Limits (scroll up)

**Testing with Sufficient Data Points**

Limits are checked only at the actual measured data points. Therefore, it is possible for a device to be out of specification without a limit test failure indication if the data point density is insufficient.

The following image is a data trace of an actual filter using 11 data points (approximately one every vertical graticule). The filter is being tested with a minimum limit line (any data point under the limit line fails).

Although the data trace is clearly below the limit line on both sides of the filter skirts, there is a PASS indication because there is no data point being measured at these frequencies.
The following image shows the exact same conditions, except the number of data points is increased to 1601. The filter now fails the minimum limit test indicated by the red data trace.
Save and Recall a File

The PNA allows you to save and recall files to and from an internal or external storage device in a variety of file formats.

- **How to Save PNA Instrument State**
- **How to Save PNA Measurement Data**
- **How to Recall a File**
- **About Instrument State and Calibration Data** (.csa, .cst, .sta, .cal)
- **About Measurement Data Files** (.prn, .snp, .cti, .csv, .mdf)
- **Managing Files without a Mouse**

### Other Data Outputting topics

#### How to Save Instrument State and Calibration Files

Use one of the following methods:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Save</strong></td>
<td>1. Click <strong>File</strong></td>
</tr>
<tr>
<td>2. then <strong>[Save], [Save As], or [Auto Save]</strong></td>
<td>2. then <strong>Save</strong> or <strong>Save As</strong></td>
</tr>
</tbody>
</table>

**Learn all about PNA Instrument State files.**

**Save**  Immediately saves the PNA state and possibly calibration data to the filename and extension you used when you last performed a Save. This file will be overwritten the next time you click **Save**. To prevent this, use one of the following methods.

**Save As**  Starts the **Save As** dialog box.

**Auto Save**  NOT available from the PNA menu. Saves state and calibration data to the internal hard disk in the C:/Program Files/Agilent/Network Analyzer/Documents folder. A filename is generated automatically using the syntax "atxxx.csa"; where xxx is a number that is incremented by one when a new file is Auto Saved.
Save (State and Calibration) As dialog box help

**Save** Allows you to navigate to the directory where you want to save the file.

**File name** Displays the filename that you either typed in or clicked on in the directory contents box.

**Save as type**

The following file types save **Instrument states and Calibration data**. You can save, and later recall, instrument settings and calibration data for **all channels** currently in use on the PNA. These file types are only recognized by Agilent PNA Series analyzers.

Learn more about these file types.

- ***.csa** - save Instrument state and actual Cal Set data (cal/state archive) **Default selection**.
- ***.cst** - save Instrument state and a link to the Cal Set data.
- ***.sta** - save Instrument state ONLY (no calibration data)
- ***.cal** - save actual Calibration data ONLY (no Instrument state)

**Note:** To save the PNA screen as .bmp, .jpg, or .png graphics file types, click File / Print to File. Learn more.

**Save** Saves the file to the specified file name and directory.

---

How to Save Measurement Data

Use one of the following methods:

**Using front-panel HARDKEY [softkey] buttons**

1. Press **SAVE**
2. then **[Save Data As]**

**Using a mouse with PNA Menus**

1. Click **File**
2. then **Save Data As**

**Save Data As** Saves the current trace(s) to the specified type of file.

**Note:** This dialog now contains the settings previously selected from the old Define Data Save dialog.

[Programming Commands]
Save Data As dialog box help

**Note:** Before saving measurement data, always trigger a single measurement, and then allow the PNA channel to go into Hold. This ensures that the entire measurement trace is saved.

**Note:** Memory traces can NOT be saved to any file type (PRN, SNP, CTI, CSV, MDF).

**Save** Allows you to navigate to the directory where you want to save the file.

**File name** Displays the filename that you either typed in or clicked on in the directory contents box.

**Save as type** Choose from: (click each to learn more about each file type): `.prn`, `.SNP`, `.SNPX`, `.cti` (citifile), `.csv`, `.mdf`.

- FCA, GCA, Swept IMD, and Swept IMDx data can be saved to a special csv format. Learn how ([FCA](#), [GCA](#) - Swept IMD)
- To save the PNA screen as .bmp, .jpg, or .png graphics file types, click **File / Print to File**. [Learn more](#).

**Data Scope**
Determines what traces are saved to a file. Available ONLY with `.cti`, `.csv`, and `.mdf`.

- **Auto**
  - When correction is OFF, saves the specified trace.
  - When correction is ON, saves all corrected parameters associated with the calibrated ports in the Cal Set.
    - For GCA and Swept IMD channels, saves the active trace only.
- **Single Trace** - Saves the active trace.
- **Displayed Traces** - Saves all displayed traces for all channels.

**Format**
Determines the format of the data. Available with (CTI Formatted, CSV, SNP, MDIF)

- **Auto** - Data is saved in LogMag or LinMag if one of these is the currently selected display format. If format is other than these, then data is saved in Real/Imag.
- **Displayed Format** (CSV and MDIF only) - Data is saved in the format of the displayed trace.
- **LogMag, LinMag, Real/Imag** - Select output format.
• The imaginary portion for all LogMag and LinMag data is saved in degrees.
• Real/Imag data is never smoothed.

Note: .prn files can only save the active trace in the displayed format.
Save Saves the file to the specified file name and directory.

How to Recall (open) a file
Select a file from the 'most recently used' list. The list is saved when the PNA application exits.
Use one of the following methods:

Using front-panel HARDKEY [softkey] buttons
1. Press Recall
2. then [Recall]

Using a mouse with PNA Menus
1. Click File
2. then Recall
Recall dialog box help

**Look in**  Allows you to select the directory that contains the file that you want to recall.

**Filename**  Displays the filename that you either typed in or clicked on in the directory contents box.

**Files of type**  Allows you view and select files that are listed in categories of a file type. The following types of files can be recalled into the PNA: All State files, Citi files, SNP files.

**Recalling PNA State files**

When an Instrument State file is recalled into the PNA, the current state of the instrument is overwritten with the recalled state. A *.cal file does not contain an instrument state, but only calibration data. Learn more about Instrument States.

See also Power ON and OFF during Save / Recall, User Preset, and Preset.

**Recalling Data files into the PNA**

Citi files and SNP files can be recalled and viewed in the PNA.

1. Click **File** then **Recall**.
2. Select **Citifile Data** or **Snp**.
3. Select the file to recall
4. Click **Recall**.

**Note:**  Citi files that were saved in **CW Time sweep** can NOT be recalled into the PNA

Recalled data is ALWAYS displayed on the PNA using **LogMag format**, regardless of how the file was stored. The channel is placed in Trigger Hold. If triggering is resumed, the data will be overwritten.

**SNP files**  are recalled as traces into a single window and channel, beginning at the highest available channel number allowed on the PNA. For multi-port SNP files (greater than 4 ports), if the number of S parameters in the file is beyond the maximum number of traces in a window, then new windows will be created.

**Citi files**  are recalled into the same window and channel configuration as when they were saved. However, the new recalled channel numbers begin with the highest channel number allowed on the PNA and decrement for each additional channel.

For example, when a citi file is saved, two traces are in window 1, channel 1 and two additional traces are in window 2, channel 2. When recalled into a factory preset condition (1 trace in window 1, channel 1), the first two recalled traces appear in window 2, highest channel number, and the second two traces appear in window 3, (highest channel number -1).

See also Traces, Channels, and Windows on the PNA

**Recall**  Recalls the file displayed in the file name box.

**Instrument State / Calibration Files**

You can save, and later recall, instrument settings and calibration data **for all channels** currently in use on the
PNA.

An **Instrument State** contains almost every PNA setting. The following PNA settings are NOT saved and recalled with Instrument State:

- **GPIB address**
- **RF power ON/OFF** (depends on current setting)
- **Test set I/O settings**

The following file types are used to save and recall instrument states and Cal Set information:

<table>
<thead>
<tr>
<th>File Types</th>
<th>Information that is stored for each channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>.csta</td>
<td><strong>Instrument State Information</strong></td>
</tr>
<tr>
<td></td>
<td>- Channels/Traces</td>
</tr>
<tr>
<td></td>
<td>- Windows</td>
</tr>
<tr>
<td></td>
<td>- Triggering</td>
</tr>
<tr>
<td></td>
<td>- Format</td>
</tr>
<tr>
<td></td>
<td>- Scale</td>
</tr>
<tr>
<td></td>
<td>- Stimulus Information:</td>
</tr>
<tr>
<td></td>
<td>- Frequency range</td>
</tr>
<tr>
<td></td>
<td>- Number of points</td>
</tr>
<tr>
<td></td>
<td>- IF bandwidth</td>
</tr>
<tr>
<td></td>
<td>- Sweep type</td>
</tr>
<tr>
<td></td>
<td>- Sweep mode</td>
</tr>
<tr>
<td>.sta</td>
<td><em>Cal Set Information</em></td>
</tr>
<tr>
<td></td>
<td>- GUID (Globally Unique Identifier)</td>
</tr>
<tr>
<td></td>
<td>- Name, Description, Modify date</td>
</tr>
<tr>
<td></td>
<td>- Stimulus Information:</td>
</tr>
<tr>
<td></td>
<td>- Frequency range</td>
</tr>
<tr>
<td></td>
<td>- Number of points</td>
</tr>
<tr>
<td></td>
<td>- IF bandwidth</td>
</tr>
<tr>
<td></td>
<td>- Sweep type</td>
</tr>
<tr>
<td></td>
<td>- Sweep mode</td>
</tr>
<tr>
<td>.csa</td>
<td>Error Terms: Directivity, Crosstalk, Source match, Load match, Reflection tracking, Transmission tracking</td>
</tr>
</tbody>
</table>

**File Type Descriptions and Recall**

The following describes each file type, and what occurs when the file type is recalled.

***.sta files**

- Contain ONLY instrument state information - NOT Cal data.
- When recalled, they always replace the current instrument state immediately.
*.cst files

- Contain BOTH instrument state and a LINK to the Cal Sets. Learn more about Cal Sets.

- The quickest and most flexible method of saving and recalling a calibrated instrument state.

- Channels need not have cal data to save as .cst file.

- When recalled, the state information is loaded first. Then the PNA tries to apply a Cal Set as you would do manually. If the stimulus settings are different between the instrument state and the linked Cal Set, the usual choice is presented (see Cal Sets). If the linked Cal Set has been deleted, a message is displayed, but the state information remains in place.

- Because only a link to the Cal Set is saved, the Cal Set can be shared with other measurements.

Note: Before saving a .cst file, be sure that a User Cal Set (NOT a Cal Register) is being used for the calibration. Cal Registers are overwritten with new data whenever a calibration is performed, and may not be accurate cal data when the .cst file is recalled. Learn more about Cal Sets.

*.cal files

- Contain ONLY Cal Set information.

- When recalled, the Cal Set is NOT automatically applied. Apply the calibration data to a channel as you would apply any Cal Set.

- Learn about Recalling

*.csa files

- Contain ALL instrument state and the actual Cal Set; not a link to the Cal Set.

- The safest method of saving and recalling a calibrated instrument state. However, the file size is larger than a *.cst file, and the save and recall times are longer. In addition, because the actual Cal Set is saved, it is very difficult to share the cal data with other measurements.

- Channels need not be calibrated to save as .cst file.

- The Cal Set that is saved could be a Cal Register or a User Cal Set.

- Learn about Recalling
Note: *.pcs files are the internal file format the PNA uses for storing cal sets. There is no reason for users to access or copy these files.

**Recalling Cal Sets**

Both .cal and .csa file types contain whole Cal Sets. When these file types are recalled, the PNA checks to see if the incoming Cal Set GUID matches an existing PNA Cal Set GUID. If it does, and if the rest of the Cal Set contents are different in any way, then both of these Cal Sets can NOT coexist in the PNA and you are offered the following choices.

Because all PNA channels are saved, there could be more than one Cal Set in either of these file types.

- **Overwrite** The incoming Cal Set will replace the existing Cal Set.
- **Duplicate** (Only available with .cal recalls.) Because the Cal Set is not automatically applied, you can choose to apply either the original or duplicate Cal Set. The original Cal Set remains in the .cal file.
- **Cancel** Abandon the recall operation.

The PNA will offer a choice as described in each file type below. Learn more about Cal Sets.

**Measurement Data Files**

Measurement data is saved as ASCII file types for use in a spreadsheet or CAE programs.

**Note:** Before saving measurement data, always trigger a single measurement, and then allow the PNA channel to go into Hold. This ensures that the entire measurement trace is saved.

**Note:** Memory traces can NOT be saved to any file type (PRN, SNP, CTI, CSV, MDF).

The following file types can be saved by the PNA.
*.prn files
*.SNP (Touchstone)
*.cti (Citifile)
*.csv
*.mdf (MDIF)

*.prn Files

Prn files have the following attributes:

- Comma-separated data which can be read into rows and columns by spreadsheet software, such as Microsoft® Excel. To avoid the "delimiting" dialog boxes, change the filename extension from .prn to .csv. Then open directly into Microsoft Excel.
- Contain formatted and corrected stimulus and response data for the current active trace ONLY.
- Are Output only - they cannot be read by the PNA.
- Beginning with PNA Rev 6.2, FCA and Cal Set Viewer data can be saved to *.prn files

Example:

"S11 Log Mag"

"Frequency (Hz)" , "dB"
3.000000e+005 , -3.528682e+001 ,
4.529850e+007 , -2.817913e+001 ,
9.029700e+007 , -3.216808e+001 ,
1.352955e+008 , -3.101017e+001 ,

.SNP Format (*.s1p, *.s2p, *.s3p, *.s4p, and so forth)

Note:
This file format is used by CAE programs such as Agilent's Microwave Design System (MDS) and Advanced Design System (ADS).
Notes
Beginning with PNA Rev. A.09.20...

- *.S2P data can be saved using the File, Save Data As dialog.
- FCA data can be saved to an *.S2P file using the File, Save Data As dialog.
- Balanced parameters can be saved to *.SNP files. See the "Choose Ports " dialog.

Beginning with PNA Rev. A.09.33...

- *.S2PX files are used to save and recall Segmented FCA data. Learn more.
- *.SNP data can be recalled and viewed on the PNA, or read by the PNA embed/de-embed functions.
- The amount of data that is saved depends on the file type that you specify and the amount of data that is available:
  - To save SNP data with an external test set enabled, at the File, Save As dialog, select SNP File(*.s*p), then complete the "Choose Ports " dialog.
  - When Fixturing is enabled, all of the enabled data transforms (De-embedding, Port Z Conversion, and so forth) are applied to saved SNP files.

<table>
<thead>
<tr>
<th>File Type</th>
<th># of Ports</th>
<th># of S-parameters saved</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.s1p</td>
<td>1</td>
<td>1 S-parameter</td>
</tr>
<tr>
<td>*.s2p</td>
<td>2</td>
<td>4 S-parameters</td>
</tr>
<tr>
<td>*.s3p</td>
<td>3</td>
<td>9 S-parameters</td>
</tr>
<tr>
<td>*.s4p</td>
<td>4</td>
<td>16 S-parameters</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>*.SNP</td>
<td>N</td>
<td>N^2 S-parameters</td>
</tr>
</tbody>
</table>

.SNP data is generally used to gather all S-parameters for a fully corrected measurement. The PNA uses the data that is available on the channel of the active measurement.

- If correction is applied, then valid data is returned for all corrected s-parameters.
- If requesting less data than is available, the Choose ports for SNP data dialog appears. Previous to PNA release 6.2, data was returned beginning with the first calibrated ports until your request if fulfilled.
- If correction is NOT applied, the PNA returns as much applicable raw data as possible using S-parameter measurements on the selected channel. Data that is not available is zero-filled. For example, if correction is...
NOT applied and the active measurement is S11, and an S21 measurement also exists on the channel, then
data is returned for the S11 and S21 measurements. Data for S12 and S22 is not available and therefore
returned as zeros in Real/Imaginary format. In Log Mag/Phase format, this appears as -200 dB and 45
degrees.

- **IMPORTANT** - ALL valid data is saved using the same format and settings (trace math, offset, delay, and so forth) as the active measurement. This can cause the data that is saved for the non-active measurements to be dramatically different from the data that is displayed. For example, when saving an S2P file, if the active S11 measurement is set to Data/Mem (data divided by memory), then ALL 4 S-parameters are saved using Data/Mem. The memory trace that is used in the Data/Mem operation is the same as that used in the active (S11) measurement.

- Before saving measurement data, always **trigger a single** measurement, and then allow the PNA channel to go into Hold. This ensures that the entire measurement trace is saved.

- Learn about [FCA parameters that are saved to an S2P file](#).

### .SNP Data Output

.SNP files contain header information, stimulus data, a response data pair for EACH S-parameter measurement. The only difference between .s1p, s2p, and so forth, is the number of S-parameters that are saved.

The following is a sample of **Header information**:

```
!Agilent Technologies,E8362B,US42340026,Q.03.54
!Agilent E8362B: Q.03.54
!Date: Friday, April 25, 2003 13:46:41
!Correction: S11(Full 2 Port SOLT,1,2) S21(Full 2 Port SOLT,1,2) S12(Full 2 Port SOLT,1,2) S22(Full 2 Port SOLT,1,2)
!S2P File: Measurements:S11,S21,S12,S22:
# Hz  S RI R 50
```

**Note:** Although the following shows Real / Imag pairs, the format could also be LogMag / Phase or LinMag / Phase

* .s1p Files
  Each record contains 1 stimulus value and 1 S-parameter (total of 3 values)
  Stim Real(Sxx) Imag(Sxx)

* .s2p Files
  Each record contains 1 stimulus value and 4 S-parameters (total of 9 values)
  Stim Real(S11) Imag(S11) Real(S21) Imag(S21) Real(S12) Imag(S12) Real(S13) Imag(S13)

* .s3p Files
  Each record contains 1 stimulus value and 9 S-parameters (total of 19 values)
  Stim Real(S11) Imag(S11) Real(S12) Imag(S12) Real(S13) Imag(S13) Real(S21) Imag(S21) Real(S22) Imag(S22)

* .s4p Files (and so forth...)
  Each record contains 1 stimulus value and 16 S-parameters (total of 33 values)
  Stim Real(S11) Imag(S11) Real(S12) Imag(S12) Real(S13) Imag(S13) Real(S14) Imag(S14)
.S2PX Data Output

*.S2PX files are used for Segmented Mixer Data. Learn more.

The following ADDITIONAL columns precede parameter data:

SegIndex, InputFreq, OutputFreq, LO1Freq, InputPower, LO1Power, <parameter data>

Choose ports for SNP File dialog box help

This dialog allows you to choose which S-parameter data to save when selecting File, Save As, Trace SNP and any of the following conditions exist:

- you request less data than is available
- you want data for more than 4 ports
- a balanced measurement is active

**Number of ports** Select the number of ports for which data will be saved.

The following buttons appear when a Balanced measurement is displayed.

**Normal** Click to save normal (single-ended) port data.

**Mixed Mode** Click to save balanced (logical) port data. Choices are based on the topology selection for current active parameter:

- **SE-Bal**: Choose from S1, D2, C2 (Single-ended port 1, Differential port 2, Common port 2)
- **SE, SE, Bal**: Choose from S1, S2, D3, C3 (Single-ended port 1, Single-ended port 2, Differential port 3, Common port 3)
Bal-Bal: Choose from D1, C1, D2, C2 (Differential port 1, Common port 1, Differential port 2, Common port 2)

For example, with SE-Bal topology, choose 2 ports, S1 for first, and D2 for second. The following 4 parameters are saved: Sss11, Ssd12, Sds21, Sdd22.

Arrow buttons Click to Add and Remove ports for the following columns:

Available Ports The PNA / External test set ports. There may NOT be valid data available for all of these ports. Learn more.

Chosen Ports When OK is clicked, SNP data is saved for these ports.

OK Becomes available when the number of Chosen ports = the Number of ports to save. Click to save to SNP file.

With Number of ports = 2, .s2p data is saved; with Number of ports = 3, .s3p data is saved, and so forth.

Learn more about SNP files

.cti CitiFiles

CitiFile format is compatible with the Agilent 8510 Network Analyzer and Agilent's Microwave Design System (MDS).

You can do the following using citifiles:

- save the active trace, or all traces.
- save formatted or unformatted citifile data

To save Formatted *.cti data

1. Click File then Save Data As
2. Select Citifile Formatted Data (*.cti)
The above image is a Citifile opened in Notepad. There are two traces in separate channels - one is an FCA trace. Each trace has 3 data points. The save settings = **Displayed Traces** Content, and **Auto** Format.

Format is identified by DBANGLE (log mag), MAGANGLE (Lin Mag), or RI (real, imaginary - NOT shown)

On the data access map, Formatted data is taken from location 2 or 4.

**To save Unformatted *.cti data**

1. Click **File**, then **Save Data As**

2. Select **Citifile Data Data (*.cti)**

On the data access map, Unformatted data is taken from the block just before Format. Citifiles can be recalled and viewed in the PNA. Learn more.

**.csv Files**

**Note:** 2D Gain Compression data is saved as *.csv files using a different format than shown here. Learn more.

CSV files are read by spreadsheet programs such as Microsoft Excel.
To save *.csv files:

1. Click **File**, then **Save Data As**

2. Under **Save as type**, select **CSV Formatted Data**

*.csv files contain: header information and the following **Comma-Separated Values**.

- Stimulus data
- Data pairs for EACH S-parameter

```
!CSV A.01.01
!Agilent Technologies, N5242A, US473000004, A.08.20.01
!Format: LogMag/Phase
!Date: Thursday, October 23, 2008 11:14:01

BEGIN CH1_DATA
!Freq(Hz), S11(DB), S11(DEG), S12(DB), S12(DEG), S21(DB), S21(DEG), S22(DB), S22(DEG)
1e+009, 0.042540751, -0.024109257, 0.56931657, 0.042540751, -0.024109257, 0.56931657...
...
END

BEGIN CH2_DATA
!Freq(Hz), S11(DB), S11(DEG), S12(DB), S12(DEG), S21(DB), S21(DEG), S22(DB), S22(DEG)
1e+009, 0.042540751, -0.024109257, 0.56931657, 0.042540751, -0.024109257, 0.56931657...
...
END
```

**.mdf Files**

MDIF files are compatible with Agilent ADS (Advanced Design System). [Learn more at the Agilent website](https://www.thermal.com).  

To save *.mdf files:

1. Click **File**, then **Save Data As**

2. Under **Save as type**, select **MDIF Data**

*.mdf files contain: header information and space-separated data:

- Stimulus data
- Real and Imaginary data pair for EACH S-parameter measurement
Define Data Saves

Note: Although these settings are still supported, they are no longer necessary to save data files. The *Save Data As* dialog box contains these settings.

### How to select Define Data Saves

**Using front-panel HARDKEY [softkey] buttons**

1. Press `SYSTEM`
2. then [Configure]
3. then [More]
4. then [Preferences]
5. then **Data Saves**

**Using a mouse with PNA Menus**

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **Preferences**
5. then **Data Saves**
Define Data Saves dialog box help

Note: Although these settings are still supported, they are no longer necessary to save data files. The Save Data As dialog box contains these settings.

The following settings survive an Instrument Preset and PNA Shutdown.

**CitiFile, CSV, and MDIF Contents**

Determines what is saved to a .cti file.

- **Auto** - Saves the active trace. Additional traces are saved if correction is ON. For Full 2-port calibration, 4 traces are saved; for Full 3-port calibration, 9 traces are saved, and so forth.
- **Single Trace** - Saves the active trace.
- **Displayed Traces** - Saves all displayed traces for all channels.

**CitiFile and CSV Format**

- **Auto** - Data is saved in LogMag or LinMag if one of these is the currently selected display format. If format is other than these, then data is saved in Real/Imag.
- **LogMag, LinMag, Real/Imag** - Select output format.
  - The imaginary portion for all LogMag and LinMag data is saved in degrees.
  - Real/ Imag data is never smoothed.

**SnP Format (.s1p, .s2p, .s3p)**

- **Auto** - Data is saved in LogMag or LinMag if one of these is the currently selected format. If format is other than these, then data is saved in Real/Imag.
- **LogMag, LinMag, Real/Imag** - Select output format. The imaginary portion for all LogMag and LinMag data is output is in degrees.

Manage Files without a Mouse

The Manage Files dialog box is designed to be used from the front panel. It performs the same function as Windows Explorer, but can be used without the use of a mouse or keyboard.
Manage Files dialog box help

**Recall**  Opens a Network Analyzer file already stored in memory.
**Rename**  Renames a file that is selected in the open folder.
**Delete**  Removes a selected file from the open folder.
**Delete All**  Removes all files of the file type selected that appear in the open folder.
**New folder**  Create a new folder and give it a name.

Last modified:

1-Jun-2011  Added S2PX
20-Sep-2010  Added memory trace notes
11-Nov-2009  Added snp fixturing note
3-Sep-2008  Removed legacy content
22-Aug-2008  Clarified invalid SNP data as -200 dB.
9-Jun-2008  Added no smoothing for real/imag cti files.
12-May-2008  Removed csv detail
17-Apr-2008  Added clarification and image to define data saves.
17-Oct-2007  Added note for MM
10/23/06  Added pcs note

510
9/18/06  MQ Added choose ports for snp

9/12/06  Added link to programming commands
**Drive Mapping**

Drive mapping allows you to share disk drives between the PNA and an external computer. You can either map from the PNA, or from your PC, to the other.

- **From the PNA, map to a drive on an External PC**
- **From an External PC, map to a drive on the PNA**

**To prepare for Drive Mapping:**

1. Both the PC and PNA must be connected to a shared computer network.
2. You must know the full computer name of the PC (or analyzer) you are mapping TO. [Tell me how]
3. Your logon and password on the analyzer must be the same as that on the external PC. You can add your PC logon to the analyzer. [Tell me how]

**Note:** These procedures require a mouse and keyboard. Also, the external PC must have Windows NT 4.0 (or later).

**From the Analyzer, map to a drive on the External PC**

1. On the external computer desktop, go to **Windows Explorer**. In the listing of drives, right click on the drive you want to share. Click **Sharing**.
2. In the dialog box, select **Shared As**. In the **Share Name** box, use the arrow key or type in a share name for the drive. For example: *C*$. Click **OK**.
3. On the analyzer desktop, click **Windows Explorer**. From the **Tools** menu, click **Map Network Drive**. (To get to the analyzer desktop, click **View**, then click **Title Bars**)
4. If you would like to connect to your external PC using a different logon, click **Connect using a different Logon**. This logon must be registered on the analyzer and you must be currently logged on the external PC using this logon.
   1. In the **Connect as** box, type your logon name. The logon name and password must be exactly the same on both the external PC and the analyzer.
   2. In the **Password** box, type the logon password that you use on the external computer. Click **OK**. The logon name and password must be exactly the same on both the external PC and the analyzer.
5. In the **Folder** box, type `//(full computer name of analyzer)/share name (from step 2).` (For example: `/SLT1234/C$`)
6. Click **Finish**.
From an External PC, map to a drive on the Analyzer

1. On the analyzer desktop, click **Windows Explorer**. Right click on the drive you want to share. Click on **Sharing**.

2. In the dialog box, select **Shared this folder**. In the **Share Name** box, type in a share name for the drive. For example: **C$**. Click **OK**.

3. On the external PC desktop, click **Windows Explorer**. From the **Tools** menu, click **Map Network Drive**.

4. If the current logon on your PC is different from the current logon on the analyzer, click **Connect using a different Logon** to connect to using the current analyzer logon. This logon must be registered on the external PC. To see the current logon on either the PC or analyzer, hold **Ctrl - Alt** and press **Delete**.

   1. In the **Connect as** box, type the logon currently being used by the analyzer.

   2. In the **Password** box, type the logon password that you use on the external computer. Click **OK**

5. In the **Folder** box, type ///computername (prep1)/share name (from step 2). (For example: //SLT1234/C$)

6. Click **Finish**.
Print a Displayed Measurement

The analyzer allows you to print a displayed measurement to a printer or to a file. The printer can be either networked or local.

- Connecting a Printer
- Printing

Other Outputting Data topics

Connecting a Printer

You can connect a printer to one of the PNA USB ports or to the LAN connector.

**CAUTION:** Do NOT connect your printer to the 25-pin female port labeled *Ext. Test Set Interface*. Voltage levels of signal lines may damage the printer's I/O.

To Add a Printer

**Note:** If you try to print from the PNA application and the Add Printer Wizard appears, click **Cancel** and add the printer using the following procedure.

1. From the PNA application, click **View** then click **Minimize Application**
2. On the Windows taskbar, click **Start**, point to **Settings**, then click **Printers**.
3. Double-click **Add Printer**.
4. Follow the instructions in the **Add Printer** Wizard.

For more information, refer to Microsoft Windows Help or your printer documentation.

Printing

- **Print a Hardcopy**
- **Page Setup**
- **Print to File**

The measurement information on the screen can be printed to any local or networked printer that is connected to the PNA. The graphic below shows an example of how a screen-capture image appears when printed. The Page Setup settings allows you to customize the printed form of the measurement information.
How to Print a Hardcopy

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press PRINT</td>
<td>1. Click File</td>
</tr>
<tr>
<td>2. then [Print]</td>
<td>2. then Print</td>
</tr>
<tr>
<td></td>
<td>3. then Print</td>
</tr>
</tbody>
</table>

No programming commands are available for this feature.
Page Setup

The Page Setup dialog allows flexibility in the appearance that measurement data is printed. After setting up the page, click **File**, then **Print**... to obtain a hard-copy.

### How to select Page Setup

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press PRINT</td>
<td>1. Click File</td>
</tr>
<tr>
<td>2. then [Page Setup]</td>
<td>2. then Print</td>
</tr>
<tr>
<td></td>
<td>3. then Print Options</td>
</tr>
</tbody>
</table>

**Note:** For information on the choices in the Print dialog box, see Windows Help.
Page Setup dialog box help

Paper, Orientation, and Margins
These settings do NOT survive a PNA shutdown.
See Windows Help for information on these settings.

Windows
The following PNA-specific settings DO survive a PNA shutdown:

Minimum vertical size  Adjust to change the amount of a page that the measurement window fills. The adjustment range is from 40 to 100%.

One window per page  Check to print one window per page. Clear to print all selected windows without a forced page break.

Only active window  Check to print only the active window. Clear to print all windows.

Agilent logo  Check to print the Agilent logo to the header.

Data and Time  Check to add the current date and time to the header.

Global Pass/Fail  Check to add the Global Pass/Fail status to the header.

Page Numbers  Check to add page numbers (1 of n) to the header.

Channel Settings Table

Print  Check to print the channel settings table.

Segment data can no longer be printed.

Trace Attributes Table

Print  Check to print the Trace Attributes Table. The Trace Attributes are measurement type, correction factors ON or OFF, smoothing, options, and marker details. The Trace Attributes are listed by Trace ID# for each window.
Each Trace ID# can have multiple entries depending on the number of markers associated with the trace. The marker details are marker number, position and response. If there are multiple markers on a trace, the trace attributes are only shown for the first marker. However, the trace attributes for the first marker apply to all other markers on that trace.

The options column can have one or more options. D for Delay, M for Marker, G for Gating. Multiple options selected would appear as follows: DMG.

Print marker data Check to print all marker data. The amount of data depends on how many markers are created.

Print to a File
The analyzer can save a screen-capture image in any of the following formats:

- .png (preferred format)
- .bmp (bitmap)
- .jpg

The analyzer automatically saves the file to the current path. If not previously defined, the analyzer automatically selects the default path C:/Program Files/Agilent/Network Analyzer/Documents/

A .bmp file, like a .prn file, can be imported into software applications such as Microsoft Excel, Word, or Paint to display a screen-capture image.

See Save and Recall files for more information.

How to Print to a File

Using front-panel HARDKEY [softkey] buttons

1. Press PRINT
2. then [Print to File]

Using a mouse with PNA Menus

1. Click File
2. then Print
3. then Print to File

Last modified:
13-Apr-2011  Removed segment data checkbox
3-Sep-2008  Removed legacy content
23-Apr-2008  Added preference survive note.
10/19/06  Modified for new print dialog
Two ways to find programming commands:

1. From a simulated PNA User Interface:

<table>
<thead>
<tr>
<th>File</th>
<th>Trace/Chan</th>
<th>Response</th>
<th>Cal</th>
<th>Marker/Analy</th>
<th>Stimulus</th>
<th>Utility</th>
</tr>
</thead>
</table>

2. Command Tree

<table>
<thead>
<tr>
<th>See Also</th>
<th>GPIB / SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Example Programs</td>
<td>Command Tree</td>
<td>COM Object Model</td>
</tr>
<tr>
<td>Learning about GPIB</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- New Programming Commands
- Remotely Specifying a Source Port
- Shut Down or Restart the PNA Remotely
- LXI and VXI-11.3 Compliance
- VEE Examples with runtime installed.
- Using Macros
- Superseded / Replacement Commands
- Data Access Map

See more PNA programming information and examples at: [http://na.tm.agilent.com/pna/programming/](http://na.tm.agilent.com/pna/programming/)
## PNA Object Model

### Application

<table>
<thead>
<tr>
<th>Channels</th>
<th>Measurements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Channel</td>
<td>Measurement</td>
</tr>
<tr>
<td>AuxiliaryTrigger</td>
<td>IMixer</td>
</tr>
<tr>
<td>BalancedTopology</td>
<td>EmbeddedLO</td>
</tr>
<tr>
<td>Calibrator</td>
<td>ELODiag</td>
</tr>
<tr>
<td>Converter</td>
<td>BalancedMeasurement</td>
</tr>
<tr>
<td>ConvEmbedLO</td>
<td>BalancedStimulus</td>
</tr>
<tr>
<td>CorrectionMethods</td>
<td>Equation</td>
</tr>
<tr>
<td>Fixturing</td>
<td>GainCompressionMeas</td>
</tr>
<tr>
<td>FOM</td>
<td>Gating</td>
</tr>
<tr>
<td>FOMRange</td>
<td>GroupDelayAperture</td>
</tr>
<tr>
<td>IFConfiguration</td>
<td>Marker</td>
</tr>
<tr>
<td>PathConfiguration</td>
<td>LimitTest</td>
</tr>
<tr>
<td>PathElement</td>
<td>LimitSegment</td>
</tr>
<tr>
<td>PhaseControl</td>
<td>PNOP</td>
</tr>
<tr>
<td>PulseGenerator</td>
<td>PSaturation</td>
</tr>
<tr>
<td>PulseMeasControl</td>
<td>Transform</td>
</tr>
<tr>
<td>RxLeveling</td>
<td></td>
</tr>
<tr>
<td>Segments</td>
<td>Capabilities</td>
</tr>
<tr>
<td>Segment</td>
<td>MeasurementClassProperties</td>
</tr>
<tr>
<td>SignalProcessing</td>
<td></td>
</tr>
<tr>
<td>GainCompression</td>
<td></td>
</tr>
<tr>
<td>NoiseFigure</td>
<td></td>
</tr>
<tr>
<td>SweptIMD</td>
<td></td>
</tr>
<tr>
<td>IMSpectrum</td>
<td></td>
</tr>
<tr>
<td>NaWindows</td>
<td></td>
</tr>
<tr>
<td>NaWindow</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Measurements

<table>
<thead>
<tr>
<th>Measurement</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMixer</td>
</tr>
<tr>
<td>EmbeddedLO</td>
</tr>
<tr>
<td>ELODiag</td>
</tr>
<tr>
<td>BalancedMeasurement</td>
</tr>
<tr>
<td>BalancedStimulus</td>
</tr>
<tr>
<td>Equation</td>
</tr>
<tr>
<td>GainCompressionMeas</td>
</tr>
<tr>
<td>Gating</td>
</tr>
<tr>
<td>GroupDelayAperture</td>
</tr>
<tr>
<td>Marker</td>
</tr>
<tr>
<td>LimitTest</td>
</tr>
<tr>
<td>LimitSegment</td>
</tr>
<tr>
<td>PNOP</td>
</tr>
<tr>
<td>PSaturation</td>
</tr>
<tr>
<td>Transform</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

### Capabilities

<table>
<thead>
<tr>
<th>Capabilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeasurementClassProperties</td>
</tr>
<tr>
<td>E5091ATestSets</td>
</tr>
<tr>
<td>E5091ATestSet</td>
</tr>
<tr>
<td>ENRFile</td>
</tr>
<tr>
<td>ExternalDevices</td>
</tr>
<tr>
<td>ExternalDevice</td>
</tr>
<tr>
<td>ExternalSource</td>
</tr>
<tr>
<td>PowerSensorAsReceiver</td>
</tr>
<tr>
<td>Traces</td>
</tr>
<tr>
<td>------------------------</td>
</tr>
<tr>
<td>Trace</td>
</tr>
<tr>
<td>CalKit</td>
</tr>
<tr>
<td>CalStandard</td>
</tr>
<tr>
<td>CalManager</td>
</tr>
<tr>
<td>CalSets</td>
</tr>
<tr>
<td>CalSet</td>
</tr>
<tr>
<td>GuidedCalibration</td>
</tr>
<tr>
<td>GuidedCalibration</td>
</tr>
<tr>
<td>PowerSensors</td>
</tr>
<tr>
<td>GuidedCalibration</td>
</tr>
<tr>
<td>PowerSensor</td>
</tr>
<tr>
<td>ECalUserCharacterizer</td>
</tr>
<tr>
<td>ECalModules</td>
</tr>
<tr>
<td>ECalModule</td>
</tr>
<tr>
<td>SMCType</td>
</tr>
<tr>
<td>VMCTYPE</td>
</tr>
<tr>
<td>NoiseCal</td>
</tr>
<tr>
<td>GainCompressionCal</td>
</tr>
<tr>
<td>SweptIMDCal</td>
</tr>
<tr>
<td>SourcePowerCal</td>
</tr>
<tr>
<td>PowerLossSegments</td>
</tr>
<tr>
<td>PowerLossSegment</td>
</tr>
<tr>
<td>PowerMeterInterfaces</td>
</tr>
<tr>
<td>PowerMeterInterface</td>
</tr>
<tr>
<td>PowerSensors</td>
</tr>
<tr>
<td>PowerSensor</td>
</tr>
<tr>
<td>CalFactorSegs</td>
</tr>
<tr>
<td>PowerSensorCal</td>
</tr>
<tr>
<td>FactorSegment</td>
</tr>
<tr>
<td>ExternalTestSets</td>
</tr>
<tr>
<td>TestSetControl</td>
</tr>
<tr>
<td>FIFO</td>
</tr>
<tr>
<td>GlobalPowerLimit</td>
</tr>
<tr>
<td>HWAUXIO</td>
</tr>
<tr>
<td>HWExternalTestSet</td>
</tr>
<tr>
<td>HWMaterialHandlerIO</td>
</tr>
<tr>
<td>InterfaceControl</td>
</tr>
<tr>
<td>PathConfigurationMgr</td>
</tr>
<tr>
<td>PortExtension</td>
</tr>
<tr>
<td>Preferences</td>
</tr>
<tr>
<td>Display/PrintColors</td>
</tr>
<tr>
<td>Trace(1-8)</td>
</tr>
<tr>
<td>SCPIStringParser</td>
</tr>
<tr>
<td>TriggerSetup</td>
</tr>
</tbody>
</table>
Last Modified:

23-May-2011  Added MeasurementClassProperties
1-Dec-2010   Added GuidedPowerSensor(s) and PhaseControl
10-Sep-2010  Added CorrectionMethods
29-Jun-2010  Fixed PathElement object
19-Feb-2010  Added PSAT, PNOP, GDAperture (9.2)
27-Aug-2009  Added External Devices
7-Apr-2009   Added ECalModules/Module
13-Nov-2007  Replaced image with text
              Added NFA, ENRFile, and GCA
Application Object

Description
The Application object is the highest object in the PNA object model. This object presents methods and properties that affect the entire analyzer, rather than a specific channel or measurement. For example, the application object provides the GetIDString method. There’s only one ID string for the instrument, unrelated to the channel or parameter being measured. Likewise, the TriggerSignal Property is global to the instrument. You can elect to use an internally generated (free run) trigger or a manual trigger. Either way, that type of trigger generation will be used on all measurements, on all channels. Therefore, it is under the Application object.

Accessing the Application object
This object is unique in that you must create this object rather than just get a handle to it.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
```

Replace `<analyzerName>` with the full computer name of your PNA. For example, "My PNA". See Change Computer Name.

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Getting a Handle to an Object.
- Example Programs
- Superseded commands

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActivateWindow</td>
<td>IApplication</td>
<td>Makes a window object the Active Window.</td>
</tr>
<tr>
<td>AllowAllEvents</td>
<td>IApplication</td>
<td>Monitors all events</td>
</tr>
<tr>
<td>AllowEventCategory</td>
<td>IApplication</td>
<td>Monitors an event category</td>
</tr>
<tr>
<td>AllowEventMessage</td>
<td>IApplication</td>
<td>Monitors an event</td>
</tr>
<tr>
<td>AllowEventSeverity</td>
<td>IApplication</td>
<td>Monitors an event severity level</td>
</tr>
<tr>
<td>BuildHybridKit</td>
<td>IApplication</td>
<td>Defines the user kit as port1kit + port2kit.</td>
</tr>
<tr>
<td>Channel</td>
<td>IApplication</td>
<td>Returns a handle to the channel object for the supplied channel number.</td>
</tr>
</tbody>
</table>

See History
<table>
<thead>
<tr>
<th>Method</th>
<th>Application</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configure</td>
<td>IApplication9</td>
<td>Restarts as an &quot;N-port&quot; PNA using the specified multiport test set.</td>
</tr>
<tr>
<td>CreateCustomMeasurement</td>
<td>IApplication3</td>
<td>Creates a new custom measurement with initialization.</td>
</tr>
<tr>
<td>CreateCustomMeasurementEx</td>
<td>IApplication</td>
<td>Superseded with CreateCustomMeasurementEx Method</td>
</tr>
<tr>
<td>CreateMeasurement</td>
<td>IApplication</td>
<td>Creates a new measurement.</td>
</tr>
<tr>
<td>CreateSParameter</td>
<td>IApplication</td>
<td>Creates a new S-Parameter measurement.</td>
</tr>
<tr>
<td>CreateSParameterEx</td>
<td>IApplication</td>
<td>Superseded with Create SParameter Method</td>
</tr>
<tr>
<td>DeleteShortcut</td>
<td>IApplication</td>
<td>Removes a macro (shortcut) from the list of macros</td>
</tr>
<tr>
<td>DisallowAllEvents</td>
<td>IApplication</td>
<td>Monitors NO events</td>
</tr>
<tr>
<td>DoPrint</td>
<td>IApplication</td>
<td>Prints the screen to the active Printer.</td>
</tr>
<tr>
<td>ExecuteShortcut</td>
<td>IApplication</td>
<td>Executes a macro (shortcut) stored in the analyzer.</td>
</tr>
<tr>
<td>GetAuxIO</td>
<td>IApplication</td>
<td>Returns a handle to the AuxIO interface</td>
</tr>
<tr>
<td>GetCalManager</td>
<td>IApplication</td>
<td>Returns a handle to the CalManager interface</td>
</tr>
<tr>
<td>GetExternalTestSetIO</td>
<td>IApplication</td>
<td>Returns a handle to the ExternalTestSet IO interface</td>
</tr>
<tr>
<td>GetIPConfigurationStruct</td>
<td>IApplication14</td>
<td>Returns an NA_IPConfiguration data structure which contains information about the current status of the PNA computer networking configuration.</td>
</tr>
<tr>
<td>GetMaterialHandlerIO</td>
<td>IApplication</td>
<td>Returns a handle to the MaterialHandlerIO interface</td>
</tr>
<tr>
<td>GetShortcut</td>
<td>IApplication</td>
<td>Returns the title and path of the specified macro (shortcut).</td>
</tr>
<tr>
<td>LANConfigurationInitialize</td>
<td>IApplication13</td>
<td>Resets the PNA LAN configuration.</td>
</tr>
<tr>
<td>LaunchCalWizard</td>
<td>IApplication</td>
<td>Launches the Cal Wizard</td>
</tr>
<tr>
<td>LaunchDialog</td>
<td>IApplication10</td>
<td>Launches the specified dialog box.</td>
</tr>
<tr>
<td>ManualTrigger</td>
<td>IApplication</td>
<td>Triggers the analyzer when TriggerSignal = naTriggerManual.</td>
</tr>
<tr>
<td>Preset</td>
<td>IApplication</td>
<td>Resets the analyzer to factory defined default settings.</td>
</tr>
<tr>
<td>PrintToFile</td>
<td>IApplication</td>
<td>Saves the screen data to bitmap (.bmp) file of the screen.</td>
</tr>
<tr>
<td>PutShortcut</td>
<td>IApplication</td>
<td>Puts a Macro (shortcut) file into the analyzer.</td>
</tr>
<tr>
<td>Quit</td>
<td>IApplication</td>
<td>Ends the Network Analyzer application.</td>
</tr>
<tr>
<td>Recall</td>
<td>IApplication</td>
<td>Recalls a measurement state, calibration state, or both from the hard drive into the analyzer.</td>
</tr>
</tbody>
</table>
RecallKits | IApplication | Recalls the calibration kits definitions that were stored with the SaveKits command.
---|---|---
Reset | IApplication | Removes all existing windows and measurements.
RestoreCalKitDefaults | IApplication | Restores the factory defaults for the specified kit.
RestoreCalKitDefaultsAll | IApplication | Restores the factory defaults for all kits.
Save | IApplication | Saves instrument state and calibration files to disk

| SaveCitiDataData | IApplication5 | Saves UNFORMATTED trace data to .cti file. Superseded with SaveData
---|---|---
| SaveCitiFormattedData | IApplication5 | Saves FORMATTED trace data to .cti file. Superseded with SaveData

SaveData | IApplication18 | Saves trace data to files on disk.
---|---|---
SaveKits | IApplication | Saves all cal kits to disk.
SetFailOnOverRange | IApplication | Causes over range values to return an error code
SetIPConfiguration | IApplication14 | Modifies settings of the PNA’s computer networking configuration.
ShowStatusBar | IApplication | Shows and Hides the Status Bar.
ShowStimulus | IApplication | Shows and Hides Stimulus information.
ShowTitleBars | IApplication | Shows and Hides the Title Bars.
ShowToolbar | IApplication | Shows and Hides the specified Toolbar.
UserPreset | IApplication7 | Performs a User Preset.
UserPresetLoadFile | IApplication7 | Loads an existing instrument state file (.sta or .cst) to be used for User Preset.
UserPresetSaveState | IApplication7 | Saves the current instrument settings as UserPreset.sta.

| Properties | Description |
---|---|---
ActiveCalKit | IApplication | Returns a pointer to the kit identified by kitNumber.
ActiveChannel | IApplication | Returns a handle to the Active Channel object.
ActiveMeasurement | IApplication | Returns a handle to the Active Measurement object.
ActiveNAWindow | IApplication | Returns a handle to the Active Window object.
<table>
<thead>
<tr>
<th>Function</th>
<th>Category</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ArrangeWindows</td>
<td>IApplication</td>
<td>Sets or returns the arrangement of all the windows.</td>
</tr>
<tr>
<td>AuxiliaryTriggerCount</td>
<td>IApplication11</td>
<td>Returns the number of Aux trigger input / output connector pairs in the instrument.</td>
</tr>
<tr>
<td>CalKitType</td>
<td>IApplication</td>
<td>Sets or returns the calibration kit type for to be used for calibration or for kit modification. Shared with the CalKit object.</td>
</tr>
<tr>
<td>Capabilities</td>
<td>IApplication4</td>
<td>Return capabilities of the remote PNA.</td>
</tr>
<tr>
<td>Channels</td>
<td>IApplication</td>
<td>Collection for iterating through the channels</td>
</tr>
<tr>
<td>CoupledMarkers</td>
<td>IApplication</td>
<td>Sets (or reads) coupled markers ON and OFF</td>
</tr>
<tr>
<td>DisplayAutomationErrors</td>
<td>IApplication2</td>
<td>Enables or disables automation error messages from being displayed on the screen.</td>
</tr>
<tr>
<td>DisplayGlobalPassFail</td>
<td>IApplication6</td>
<td>Shows or hides the dialog which displays global pass/fail results.</td>
</tr>
<tr>
<td>E5091Testsets</td>
<td>IApplication8</td>
<td>Collection to control the E5091A testset.</td>
</tr>
<tr>
<td>ENRFile</td>
<td>IApplication13</td>
<td>Manages Noise ENR files.</td>
</tr>
<tr>
<td>ExternalALC</td>
<td>IApplication</td>
<td>Sets or returns the source of the analyzer leveling control.</td>
</tr>
<tr>
<td>ExternalDevices</td>
<td>IApplication16</td>
<td>Collection to control External Devices</td>
</tr>
<tr>
<td>ExternalTestsets</td>
<td>IApplication9</td>
<td>Collection to control External Test sets.</td>
</tr>
<tr>
<td>FIFO</td>
<td>IApplication15</td>
<td>Controls FIFO settings</td>
</tr>
<tr>
<td>GlobalPowerLimit</td>
<td>IApplication17</td>
<td>Controls Global Power Limit settings</td>
</tr>
<tr>
<td>GPIBAddress</td>
<td>IApplication8</td>
<td>Sets and returns the PNA GPIB address.</td>
</tr>
<tr>
<td>GPIBMode</td>
<td>IApplication</td>
<td>Makes the analyzer the system controller or a talker/listener.</td>
</tr>
<tr>
<td>GridLineType</td>
<td>IApplication17</td>
<td>Set and return the line type of the window grid (solid</td>
</tr>
<tr>
<td>IDString</td>
<td>IApplication</td>
<td>Returns the model, serial number and software revision of the analyzer</td>
</tr>
<tr>
<td>InterfaceControl</td>
<td>IApplication8</td>
<td>Control the Interface control features.</td>
</tr>
<tr>
<td>LANConfiguration</td>
<td>IApplication13</td>
<td>Returns the current status of the PNA computer networking configuration.</td>
</tr>
<tr>
<td><strong>LXIDeviceIDState</strong></td>
<td>IApplication14</td>
<td>Displays the LAN Status dialog with LAN Status Indicator showing IDENTIFY.</td>
</tr>
<tr>
<td>----------------------</td>
<td>----------------</td>
<td>---------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>LocalLockoutState</strong></td>
<td>IApplication4</td>
<td>Prevents use of the mouse, keyboard, and front panel while your program is running.</td>
</tr>
<tr>
<td><strong>Measurement</strong></td>
<td>IApplication</td>
<td>Create and manage measurements</td>
</tr>
<tr>
<td><strong>Measurements</strong></td>
<td>IApplication</td>
<td>Collection for iterating through the Application measurements.</td>
</tr>
<tr>
<td><strong>MessageText</strong></td>
<td>IApplication</td>
<td>Returns text for the specified eventID</td>
</tr>
<tr>
<td><strong>NaWindows</strong></td>
<td>IApplication</td>
<td>Collection for iterating through the Application windows.</td>
</tr>
<tr>
<td><strong>NoiseSourceState</strong></td>
<td>IApplication13</td>
<td>Sets and Reads the ON</td>
</tr>
<tr>
<td><strong>NumberOfPorts</strong></td>
<td>IApplication</td>
<td>Returns the number of hardware source ports on the PNA</td>
</tr>
<tr>
<td><strong>Options</strong></td>
<td>IApplication</td>
<td>Returns the options on the analyzer</td>
</tr>
<tr>
<td><strong>PathConfigurationManager</strong></td>
<td>IApplication11</td>
<td>Provides access to hardware configuration.</td>
</tr>
<tr>
<td><strong>Port Extensions</strong></td>
<td>IApplication</td>
<td>Superseded with Fixturing Object</td>
</tr>
<tr>
<td><strong>Preferences</strong></td>
<td>IApplication5</td>
<td>Preferences for many PNA settings..</td>
</tr>
<tr>
<td><strong>ScpiStringParser</strong></td>
<td>IApplication</td>
<td>Provides the ability to send a SCPI command from within the COM command.</td>
</tr>
<tr>
<td><strong>SecurityLevel</strong></td>
<td>IApplication4</td>
<td>Turns ON or OFF the display of frequency information.</td>
</tr>
<tr>
<td><strong>SICL</strong></td>
<td>IApplication5</td>
<td>Allows control of the PNA via SICL</td>
</tr>
<tr>
<td><strong>SICLAddress</strong></td>
<td>IApplication8</td>
<td>Sets and returns the PNA SICL address</td>
</tr>
<tr>
<td><strong>SourcePowerCalibrator</strong></td>
<td>IApplication2</td>
<td>Allows capability for performing source power calibrations.</td>
</tr>
<tr>
<td><strong>SourcePowerState</strong></td>
<td>IApplication</td>
<td>Turns Source Power ON and OFF.</td>
</tr>
<tr>
<td><strong>SystemImpedanceZ0</strong></td>
<td>IApplication</td>
<td>Sets the analyzer impedance value.</td>
</tr>
<tr>
<td><strong>SystemName</strong></td>
<td>IApplication</td>
<td>Returns the full computer name of the PNA.</td>
</tr>
<tr>
<td><strong>Touchscreen</strong></td>
<td>IApplication12</td>
<td>Enables and disables touchscreen.</td>
</tr>
<tr>
<td><strong>TriggerDelay</strong></td>
<td>IApplication</td>
<td>Sets or returns the delay time for a trigger.</td>
</tr>
<tr>
<td><strong>TriggerSetup</strong></td>
<td>IApplication4</td>
<td>Controls triggering for the entire PNA application.</td>
</tr>
<tr>
<td>Event</td>
<td>Interface</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------</td>
<td>-------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>OnCalEvent</td>
<td>IApplication</td>
<td>Triggered by a calibration event.</td>
</tr>
<tr>
<td>OnChannelEvent</td>
<td>IApplication</td>
<td>Triggered by a channel event.</td>
</tr>
<tr>
<td>OnDisplayEvent</td>
<td>IApplication</td>
<td>Triggered by a display event.</td>
</tr>
<tr>
<td>OnHardwareEvent</td>
<td>IApplication</td>
<td>Triggered by a hardware event.</td>
</tr>
<tr>
<td>OnMeasurementEvent</td>
<td>IApplication</td>
<td>Triggered by a measurement event.</td>
</tr>
<tr>
<td>OnSCPIEvent</td>
<td>IApplication</td>
<td>Triggered by a SCPI event.</td>
</tr>
<tr>
<td>OnSystemEvent</td>
<td>IApplication</td>
<td>Triggered by a system event.</td>
</tr>
<tr>
<td>OnUserEvent</td>
<td>IApplication</td>
<td>For future use</td>
</tr>
</tbody>
</table>

**UserPresetEnable**

'Checks' and 'clears' the enable box on the User Preset dialog box.

**VelocityFactor**

Sets the velocity factor to be used with Electrical Delay, Port Extensions, and Time Domain marker distance calculations.

**Visible**

Makes the Network Analyzer application visible or not visible.

**WindowState**

Sets or returns the window setting of Maximized, Minimized, or Normal.

Shared with the NAWindow Object

**TriggerSignal**

IAplication Superseded with Source Property

**TriggerType**

IAplication Superseded with Scope Property

**IApplication History**
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IApplication</td>
<td>1.0</td>
</tr>
<tr>
<td>IApplication2</td>
<td>3.0</td>
</tr>
<tr>
<td>IApplication3</td>
<td>3.2</td>
</tr>
<tr>
<td>IApplication4</td>
<td>3.5</td>
</tr>
<tr>
<td>IApplication5</td>
<td>4.0</td>
</tr>
<tr>
<td>IApplication6</td>
<td>5.0</td>
</tr>
<tr>
<td>IApplication7</td>
<td>5.0</td>
</tr>
<tr>
<td>IApplication8</td>
<td>5.2</td>
</tr>
<tr>
<td>IApplication9</td>
<td>6.0</td>
</tr>
<tr>
<td>IApplication10</td>
<td>7.20</td>
</tr>
<tr>
<td>IApplication11</td>
<td>7.20</td>
</tr>
<tr>
<td>IApplication12</td>
<td>7.21</td>
</tr>
<tr>
<td>IApplication13</td>
<td>8.0</td>
</tr>
<tr>
<td>IApplication14</td>
<td>8.2</td>
</tr>
<tr>
<td>IApplication15</td>
<td>8.34</td>
</tr>
<tr>
<td>IApplication16</td>
<td>9.0</td>
</tr>
<tr>
<td>IApplication17</td>
<td>9.0</td>
</tr>
<tr>
<td>IApplication18</td>
<td>9.2</td>
</tr>
</tbody>
</table>

Last Modified:

17-Oct-2007   Updated IPathConfigMgr Prop
AuxiliaryTrigger Object

Description
These properties setup Auxiliary triggering on a channel.

Accessing the object
Use chan.AuxTrigger (n) to access the object.
where n= the connector pair to be used for Auxiliary Triggering.

- PNA-X models: Use 1 or 2
- All other PNA models: Use 1 - These models do NOT have an Aux Input; only an Output. Therefore, the following 'Input' commands will return an error when sent to PNA models other than the PNA-X.

Use app.AuxiliaryTriggerCount to determine the number of auxiliary trigger pairs on the rear panel of a PNA.

```
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim AuxTrig as AuxTrigger
AuxTrig = chan.AuxTrigger(2)
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Triggering in the PNA
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delay (Input)</td>
<td>IAuxTrigger</td>
</tr>
<tr>
<td>Enable</td>
<td>IAuxTrigger</td>
</tr>
</tbody>
</table>
HandshakeEnable (Input)  I AuxTrigger  Turns handshake ON / OFF.

Number  I AuxTrigger  Reads the number of the Aux I/O pair being used.

TriggerInPolarity (Input)  I AuxTrigger  Specifies the polarity of the trigger IN signal to which the PNA will respond.

TriggerInType (Input)  I AuxTrigger  Specifies the type of Aux trigger input being supplied to the PNA

TriggerOutDuration  I AuxTrigger  Specifies the width of the pulse or the time that the Aux trigger output will be asserted

TriggerOutInterval  I AuxTrigger  Specifies how often a trigger output signal is sent.

TriggerOutPolarity  I AuxTrigger  Specifies the polarity of the trigger output signal being supplied by the PNA.

TriggerOutPosition  I AuxTrigger  Specifies whether the Aux trigger out signal is sent Before or After the acquisition.

I AuxTrigger History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>I AuxTrigger</td>
<td>7.2</td>
</tr>
</tbody>
</table>

Last Modified:

6-Apr-2009  Replaced N5242A with PNA-X
13-Aug-2008  Fixed example
BalancedMeasurement Object

Description
These properties set the measurement type that is used with balanced topologies.

Use the BalancedTopology Object to set the topology and port mappings for the DUT,

Accessing the BalancedMeasurement object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim balMeas As BalancedMeasurement
Set balMeas = app.ActiveMeasurement.BalancedMeasurement

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- About Balanced Measurements
- Example Programs

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BalancedMode</td>
<td>IBalancedMeasurement</td>
<td>Sets and returns whether the balanced transform is ON or OFF.</td>
</tr>
<tr>
<td>BalancedTopology</td>
<td>IBalancedMeasurement</td>
<td>Sets and returns the topology of a balanced DUT.</td>
</tr>
<tr>
<td>BBalMeasurement</td>
<td>IBalancedMeasurement</td>
<td>Sets and returns the measurement for the Balanced - Balanced topology.</td>
</tr>
</tbody>
</table>
IBalancedMeasurement Sets and returns the measurement for the Single-Ended - Single-Ended - Balanced topology

IBalancedMeasurement History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IBalancedMeasurement</td>
<td>5.0</td>
</tr>
</tbody>
</table>
**BalancedStimulus Object**

**Description**
These properties set the values that are unique to iTMSA - Opt 460.
All other properties for iTMSA use the standard PNA commands.

**Accessing the BalancedStimulus object**

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim balStim As BalancedStimulus
```

**See Also:**
- PNA Automation Interfaces
- The PNA Object Model
- About iTMSA
- Example iTMSA Program

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BalPort1PhaseOffset</td>
<td>IBalancedStimulus</td>
<td>Sets balanced port 1 phase offset</td>
</tr>
<tr>
<td>BalPort1PowerOffset</td>
<td>IBalancedStimulus</td>
<td>Sets balanced port 1 power offset</td>
</tr>
<tr>
<td>BalPort1StartPhase</td>
<td>IBalancedStimulus2</td>
<td>Sets Phase start value for port 1</td>
</tr>
<tr>
<td>BalPort1StopPhase</td>
<td>IBalancedStimulus2</td>
<td>Sets Phase stop value for port 1</td>
</tr>
<tr>
<td>BalPort2PhaseOffset</td>
<td>IBalancedStimulus</td>
<td>Sets balanced port 2 phase offset</td>
</tr>
<tr>
<td>BalPort2PowerOffset</td>
<td>IBalancedStimulus</td>
<td>Sets balanced port 2 power offset</td>
</tr>
</tbody>
</table>
BalPort2StartPhase  IBalancedStimulus2  Sets Phase start value for port 2
BalPort2StopPhase  IBalancedStimulus2  Sets Phase start value for port 2
Mode  IBalancedStimulus  Sets Stimulus mode for balanced measurements
PhaseAsFixture  IBalancedStimulus  Sets the state of phase offset as a fixture
PhaseSwpAsFixture  IBalancedStimulus2  Enable Phase Sweep as fixture
PhaseSwpState  IBalancedStimulus2  Enable Phase Sweep
PowerAsFixture  IBalancedStimulus  Sets the state of power offset as a fixture

IBalancedStimulus History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IBalancedStimulus</td>
<td>8.2</td>
</tr>
<tr>
<td>IBalancedStimulus2</td>
<td>8.5</td>
</tr>
</tbody>
</table>

Last Modified:

27-Feb-2009  Added IBal2 - Phase Sweep
15-May-2008  MX New topic
BalancedTopology Object

Description
The DUTTopology property sets and returns the topology of a balanced DUT.
The following methods set the port mappings for the DUT.
The remaining properties return the port mappings for the DUT.
Use the BalancedMeasurement object to set the measurement type.

Accessing the BalancedTopology object

```vba
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel

Dim balTopology as BalancedTopology
Set balTopology = chan.BalancedTopology
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- About Balanced Measurements
- Example Programs

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SetBBPorts</td>
<td>IBalancedTopology</td>
<td>Sets the physical port mappings for the Balanced - Balanced DUT topology.</td>
</tr>
<tr>
<td>SetSBPorts</td>
<td>IBalancedTopology</td>
<td>Sets the physical port mappings for the Single-Ended - Balanced DUT topology.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BB_BalPort1Negative</td>
<td>IBalancedTopology</td>
<td>Returns the PNA port number that is connected to the Negative side of the DUT's logical Port 1.</td>
</tr>
</tbody>
</table>
BB_BalPortPositive IBalancedTopology Returns the first positive balanced port number in the Balanced - Balanced topology

BB_BalPort2Negative IBalancedTopology Returns the second negative balanced port number in the Balanced - Balanced topology.

BB_BalPort2Positive IBalancedTopology Returns the second positive balanced port number in the Balanced - Balanced topology.

DUTTopology IBalancedTopology Sets and returns the device topology setting.

SB_BalPortNegative IBalancedTopology Returns the negative balanced port number in the Single-Ended - Balanced topology.

SB_BalPortPositive IBalancedTopology Returns the positive balanced port number in the Single-Ended - Balanced topology.

SB_SEPort IBalancedTopology Returns the single ended port number in the Single-Ended - Balanced topology.

SSB_BalPortNegative IBalancedTopology Returns the negative balanced port number in the Single-Ended - Single-Ended - Balanced topology.


BalancedTopology History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IBalancedTopology</td>
<td>5.0</td>
</tr>
</tbody>
</table>
CalFactorSegments Collection

Description
A collection object that provides a mechanism for iterating through the segments of a power sensor cal factor table. The Cal Factor table can contain up to 100 segments.

Accessing the CalFactorSegments collection

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim calFact As CalFactorSegments

See Also:
- PowerSensorCalFactorSegment Object
- About Source Power Cal
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a PowerSensorCalFactorSegment object to the collection</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to a PowerSensorCalFactorSegment object in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes an object from the collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of objects in the collection.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the Parent object (PowerSensor) of this collection.</td>
</tr>
</tbody>
</table>
**CalFactorSegmentsPMAR Collection**

**Description**
A collection object that provides a mechanism for iterating through the segments of a power sensor cal factor table. The Cal Factor table can contain up to 100 segments. This collection is used when the Power Meter is used as a Receiver.

**Accessing the CalFactorSegmentsPMAR collection**
Example: Create a PMAR Device and Measurement

**See Also:**
- **PowerSensorCalFactorSegmentPMAR Object**
- **About PMAR**
- **Collections in the Analyzer**
- **The PNA Object Model**

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Add</strong></td>
<td>Adds a PowerSensorCalFactorSegmentPMAR object to the collection</td>
</tr>
<tr>
<td><strong>Item</strong></td>
<td>Use to get a handle to a PowerSensorCalFactorSegmentPMAR object in the collection.</td>
</tr>
<tr>
<td><strong>Remove</strong></td>
<td>Removes an PowerSensorCalFactorSegmentPMAR object from the collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Count</strong></td>
<td>Returns the number of PowerSensorCalFactorSegmentPMAR objects in the collection.</td>
</tr>
<tr>
<td><strong>Parent</strong></td>
<td>Returns a handle to the Parent object of this collection.</td>
</tr>
</tbody>
</table>

Last Modified:
25-Aug-2009    MX New topic
Calibrator Object

See Also

- Example Programs
- Calibrator Methods and Properties
- ICalData Interface for putting and getting typed Calibration data.
- Superseded commands

Description

The Calibrator object, a child of the channel, is used to perform an **Unguided** calibration.

**Important!**

Do **NOT** use commands from the GuidedCalibration object when performing an Unguided calibration. Use ONLY the Calibrator object.

You can **NOT** perform a full 3 or 4-port using the Calibrator object. You must use the GuidedCalibration object.

There must be a measurement present for the calibrator to use or you will receive a "no measurement found" error. Therefore, to perform a 2-port cal, you must have any S-parameter measurement on the channel. For a 1-port measurement, you must have the measurement (S11 or S22) on the channel. The same is true for a response measurement.

There are a number of approaches to calibration with the calibrator object:

1. You can collect data yourself and download it to the ACQUISITION buffer. The acquisition buffer holds the actual measured data for each standard. See the PNA data map.

   1. Calibrator.SetCalInfo
   2. Connect a standard
   3. Trigger a sweep
   4. Retrieve the data for the standard
   5. Download the data - calibrator.putStandard
   6. Repeat for each standard
   7. Calibrator.CalculateErrorCoefficients

2. You can tell the calibrator to acquire a standard. In this case, the calibrator collects the data and places it in the ACQUISITION buffer.

   1. Calibrator.SetCalInfo
   2. Connect a standard
3. Calibrator.**AcquireCalStandard2**
4. Repeat for each standard
5. Calibrator.**CalculateErrorCoefficients**

- You can put previously-retrieved error terms in the error correction buffer.

1. **PutErrorTerm**
2. Repeat for each term
3. Measurement.**Caltype** = pick one

- You can also "piece together" a 2-port cal from two 1-port cals (S11 and S22) and four response (thru) cals. The system will detect that all the standards needed for a 2-port cal have been acquired even though they may not have gathered at the same time.

**Accessing the Calibrator object**

```vbs
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", < analyzerName >)

Dim cal As ICalibrator
Set cal = app.ActiveChannel.Calibrator
```

**See Also:**

- **PNA Automation Interfaces**
- **The PNA Object Model**
- Learn about reading and writing Calibration data.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcquireCalConfidenceCheckECAL</td>
<td>ICalibrator</td>
<td><strong>Superseded with</strong> AcquireCalConfidenceCheckECALEx</td>
</tr>
<tr>
<td>AcquireCalConfidenceCheckECALEx</td>
<td><strong>ICalibrator4</strong></td>
<td>Transfers ECAL confidence data into analyzer memory</td>
</tr>
<tr>
<td>AcquireCalStandard</td>
<td>ICalibrator</td>
<td><strong>Superseded with</strong> AcquireCalStandard2</td>
</tr>
<tr>
<td>AcquireCalStandard2</td>
<td>ICalibrator</td>
<td>Causes the analyzer to measure a calibration standard. Also provides for sliding load.</td>
</tr>
<tr>
<td>CalculateErrorCoefficients</td>
<td>ICalibrator</td>
<td>Generates Error Terms from standard and actual data in the error correction buffer.</td>
</tr>
<tr>
<td>Method</td>
<td>Interface</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>--------------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>DoECAL1Port</td>
<td>ICalibrator</td>
<td>Superseded with DoECAL1PortEx</td>
</tr>
<tr>
<td>DoECAL1PortEx</td>
<td>ICalibrator4</td>
<td>Completes a 1 port ECAL</td>
</tr>
<tr>
<td>DoECAL2Port</td>
<td>ICalibrator</td>
<td>Superseded with DoECAL2PortEx</td>
</tr>
<tr>
<td>DoECAL2PortEx</td>
<td>ICalibrator4</td>
<td>Completes a 2 port ECAL</td>
</tr>
<tr>
<td>DoneCalConfidenceCheckECAL</td>
<td>ICalibrator</td>
<td>Concludes an ECAL confidence check</td>
</tr>
<tr>
<td>DoReceiverPowerCal</td>
<td>ICalibrator5</td>
<td>Perform a receiver power cal.</td>
</tr>
<tr>
<td>DoResponseCal</td>
<td>ICalibrator9</td>
<td>Perform a response (normalization) cal.</td>
</tr>
<tr>
<td>GetCalKitTypeString</td>
<td>ICalibrator8</td>
<td>Returns information about the attached modules</td>
</tr>
<tr>
<td>GetECALModuleInfo</td>
<td>ICalibrator</td>
<td>Superseded with Get ECALModuleInfoEx</td>
</tr>
<tr>
<td>Get ECALModuleInfoEx</td>
<td>ICalibrator4</td>
<td>Returns information about the attached module</td>
</tr>
<tr>
<td>getErrorTerm</td>
<td>ICalibrator</td>
<td>Superseded with GetErrorTermByString</td>
</tr>
<tr>
<td>getStandard</td>
<td>ICalibrator</td>
<td>Superseded with GetStandardByString</td>
</tr>
<tr>
<td>putErrorTerm</td>
<td>ICalibrator</td>
<td>Superseded with PutErrorTermByString</td>
</tr>
<tr>
<td>putStandard</td>
<td>ICalibrator</td>
<td>Superseded with PutStandardByString</td>
</tr>
<tr>
<td>SaveCalSets</td>
<td>ICalibrator</td>
<td>Superseded with CalSet.Save</td>
</tr>
<tr>
<td>setCallInfo</td>
<td>ICalibrator</td>
<td>Specifies the type of calibration and prepares the internal state for the rest of the calibration.</td>
</tr>
</tbody>
</table>

**Properties**

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcquisitionDirection</td>
<td>ICalibrator</td>
<td>Specifies the direction in a 2-Port cal using one set of standards.</td>
</tr>
<tr>
<td>CalKitType</td>
<td>ICalibrator10</td>
<td>Sets and returns the name of the Cal Kit to use for unguided cal.</td>
</tr>
<tr>
<td>CalKitTypes</td>
<td>ICalibrator10</td>
<td>Returns the names of the first 50 mechanical cal kits in your PNA that can be used for unguided calibrations.</td>
</tr>
<tr>
<td>ECALCharacterization</td>
<td>ICalibrator2</td>
<td>Superseded with ECALCharacterizationEx</td>
</tr>
<tr>
<td>ECALCharacterizationEx</td>
<td>ICalibrator4</td>
<td>Specifies which set of characterization data within an ECal module will be used for ECal operations with that module.</td>
</tr>
</tbody>
</table>
**ICalibrator6** Returns a list of characterizations stored in the specified ECal module.

**ICalibrator** Specifies whether the acquisition of the ECal calibration should include isolation or not.

**ICalibrator6** Returns a list of index numbers to be used for referring to the ECal modules that are currently attached to the PNA.

**ICalibrator3** Superseded with ECALPortMapEx

**ICalibrator4** Specifies which ports of the ECal module are connected to which ports of the PNA.

**ICalibrator** Superseded with IsECALModuleFoundEx

**ICalibrator4** Superseded with ECALCharacterizationIndexList and ECALModuleNumberList

**ICalibrator7** Value to increase the channel's averaging factor.

**ICalibrator3** Specifies if the PNA should perform orientation of the ECal module during calibration.

**ICalibrator** Allows the use of 2 sets of standards at the same time.

**ICalibrator History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalibrator</td>
<td>1.0</td>
</tr>
<tr>
<td>ICalibrator2</td>
<td>3.1</td>
</tr>
<tr>
<td>ICalibrator3</td>
<td>3.1</td>
</tr>
<tr>
<td>ICalibrator4</td>
<td>3.5</td>
</tr>
<tr>
<td>ICalibrator5</td>
<td>5.0</td>
</tr>
<tr>
<td>ICalibrator6</td>
<td>5.26</td>
</tr>
<tr>
<td>ICalibrator7</td>
<td>7.21</td>
</tr>
<tr>
<td>ICalibrator8</td>
<td>8.1</td>
</tr>
<tr>
<td>ICalibrator9</td>
<td>9.1</td>
</tr>
<tr>
<td>ICalibrator10</td>
<td>9.2</td>
</tr>
</tbody>
</table>
**ICalData Interface**

**Description**
Contains methods for putting Calibration data in and getting Calibration data out of the analyzer using typed data. This interface transfers data more efficiently than variant data. However, this interfaces is only usable from VB6, C, & C++. All other programming languages must use the ICalSet interface.

There is also an ICalData Interface on the CalSet Object

Learn about reading and writing Calibration data.

---

**Methods**

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>getErrorTermComplex</td>
<td>Retrieves error term data</td>
</tr>
<tr>
<td>getStandardComplex</td>
<td>Retrieves calibration data from the acquisition data buffer (before error-terms are applied).</td>
</tr>
<tr>
<td>putErrorTermComplex</td>
<td>Puts error term data</td>
</tr>
<tr>
<td>putStandardComplex</td>
<td>Puts calibration data into the acquisition data buffer (before error-terms are applied).</td>
</tr>
</tbody>
</table>

**Properties**

None

**ICalData History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalData</td>
<td>1.0</td>
</tr>
</tbody>
</table>
**CalKit Object**

**Description**
The calkit object provides the properties and methods to access and modify a calibration kit.

**Accessing a CalKit object**
The **active** cal kit is the kit that is selected for use in **Unguided** calibrations. To get a handle to the active kit, use the `app.ActiveCalKit` property. To access the CalKit object for a specific cal kit, you must first make that kit the active kit using `app.CalKitType`.

The CalKit object behaves differently from other objects in that you can only have a handle to one cal kit -- the active cal kit. Therefore, when you change the CalKitType from either the Application object or the CalKit object, you may also be changing the object to which you may have other references.

For example, the following example specifies two CalKit type objects and in turn, assigns them to two different variables: ck1 and ck2.

```vba
Dim app As AgilentPNA835x.Application
Dim ck1 As calKit
Dim ck2 As calKit

Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
app.CalKitType = naCalKit_User1
Set ck1 = app.ActiveCalKit
ck1.Name = "My CalKit1"

app.CalKitType = naCalKit_User2
Set ck2 = app.ActiveCalKit
ck2.Name = "My CalKit2"

Print "ck1: " & ck1.Name
Print "ck2: " & ck2.Name
```

When the pointer to each of these kits is read (printed), they each have a pointer to the last kit to be assigned to the active cal kit:

- ck1: My CalKit2
- ck2: My CalKit2

**See Also:**
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

(Bold Methods or Properties provide access to a child object)
### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>getCalStandard</code></td>
<td>Returns a handle to a calibration standard for modifying its definitions.</td>
</tr>
<tr>
<td><code>GetStandardsForClass</code></td>
<td>Returns the calibration standard numbers for a specified calibration class.</td>
</tr>
<tr>
<td><code>SetStandardsForClass</code></td>
<td>Sets the calibration standard numbers for a specified calibration class</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalKitType</td>
<td>Sets or returns the calibration kit type to be used for calibration or for kit modification. Shared with the Application object.</td>
</tr>
<tr>
<td>Name</td>
<td>Sets and returns the name of the cal kit</td>
</tr>
<tr>
<td>PortLabel</td>
<td>Labels the ports for the kit; only affects the cal wizard annotation.</td>
</tr>
<tr>
<td>StandardForClass</td>
<td><strong>Superseded with</strong> Use <code>GetStandardForClass</code> and <code>SetStandardForClass</code>. Maps a standard device to a cal class.</td>
</tr>
</tbody>
</table>

### ICalKit History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalKit</td>
<td>1.0</td>
</tr>
</tbody>
</table>
CalManager Object

Description
Use this interface to list, save, and delete Cal Sets.

Accessing the CalManager object
Get a handle to a the CalManager with the app.GetCalManager Method.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim mgr as ICalManager
Set mgr = app.GetCalManager
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
- Superseded commands

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AllowChannelToSweepDuringCalAcquisition</td>
<td>ICalManager5</td>
<td>Specifies the channel to sweep during a Calibration.</td>
</tr>
<tr>
<td>CreateCalSet</td>
<td>ICalManager</td>
<td>Creates a new Cal Set</td>
</tr>
<tr>
<td>CreateCustomCal</td>
<td>ICalManager2</td>
<td>Creates an FCA cal object.</td>
</tr>
<tr>
<td>CreateCustomCalEx</td>
<td>ICalManager5</td>
<td>Creates a custom cal object.</td>
</tr>
<tr>
<td>Deembed</td>
<td>ICalManager8</td>
<td>De-embeds a fixture from an existing Cal Set based on an S2P file.</td>
</tr>
<tr>
<td>DeleteCalSet</td>
<td>ICalManager</td>
<td>Deletes a Cal Set</td>
</tr>
<tr>
<td>DisplayNAWindowDuringCalAcquisition</td>
<td>ICalManager5</td>
<td>Set the 'show' state of the window to be displayed during a calibration.</td>
</tr>
<tr>
<td>Method Name</td>
<td>ICalManager</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>-------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DisplayOnlyCalWindowDuringCalAcquisition</td>
<td>ICalManager5</td>
<td>Clears the flags for windows to be shown during calibrations.</td>
</tr>
<tr>
<td>Embed</td>
<td>ICalManager8</td>
<td>Embeds a fixture into an existing Cal Set based on an S2P file.</td>
</tr>
<tr>
<td>EnumerateCalSets</td>
<td>ICalManager4</td>
<td>Returns an array of Cal Set names being stored on the PNA.</td>
</tr>
<tr>
<td>GetCalSetByGUID</td>
<td>ICalManager</td>
<td>Get a handle to a Cal Set</td>
</tr>
<tr>
<td>GetCalSetCatalog</td>
<td>ICalManager</td>
<td><strong>Superseded with</strong> EnumerateCalSets</td>
</tr>
<tr>
<td>GetCalSetUsageInfo</td>
<td>ICalManager</td>
<td>Returns the Cal Set ID and Error Term ID currently in use</td>
</tr>
<tr>
<td>GetCalTypes</td>
<td>ICalManager2</td>
<td>Query for a list of available calibration types.</td>
</tr>
<tr>
<td>GetEcalUserCharacterizer</td>
<td>ICalManager6</td>
<td>Returns the ECalUserCharacterizer object.</td>
</tr>
<tr>
<td>GetRequiredEtermNames</td>
<td>ICalManager2</td>
<td>Returns an array of strings specifying the error terms required by the Cal Type correction algorithm.</td>
</tr>
<tr>
<td>SaveCalSets</td>
<td>ICalManager</td>
<td><strong>Superseded with</strong> CalSet.Save</td>
</tr>
<tr>
<td>SweepOnlyCalChannelDuringCalAcquisition</td>
<td>ICalManager5</td>
<td>Clears ALL flags for channels to sweep during calibration.</td>
</tr>
</tbody>
</table>

**Properties**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>ICalManager</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalSets Collection</td>
<td>ICalManager</td>
<td>Collection for iterating through all the Cal Sets in the analyzer.</td>
</tr>
<tr>
<td>ECalModules Collection</td>
<td>ICalManager7</td>
<td>Collection of ECal Modules that are connected to the PNA.</td>
</tr>
<tr>
<td>GuidedCalibration</td>
<td>ICalManager3</td>
<td>Used to perform a Guided Calibration.</td>
</tr>
</tbody>
</table>

**ICalManager History**
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICaCalManager</td>
<td>2.0</td>
</tr>
<tr>
<td>CalManager2</td>
<td>3.1</td>
</tr>
<tr>
<td>CalManager3</td>
<td>3.5</td>
</tr>
<tr>
<td>CalManager4</td>
<td>5.0</td>
</tr>
<tr>
<td>ICaCalManager5</td>
<td>8.0</td>
</tr>
<tr>
<td>ICaCalManager6</td>
<td>8.3</td>
</tr>
<tr>
<td>ICaCalManager7</td>
<td>8.5</td>
</tr>
<tr>
<td>ICaCalManager8</td>
<td>9.33</td>
</tr>
</tbody>
</table>
CalSet Object

See [ICalData Interface](#) for putting and getting typed Cal Set data.

**Description**

Use this interface to query and or change the contents of a Cal Set.

**Accessing the CalSet object**

Get a handle to a CalSet object by using the CalSets collection. This is done through the CalManager object with the app.GetCalManager Method.

```vbnet
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim calst As ICalSet
Set calst = app.GetCalManager.CalSets.Item(1)
' OR Get a handle by CalSet Name
Set calst = app.GetCalManager.CalSets.Item("MyCalSet")
```

See Also:

- [PNA Automation Interfaces](#)
- [The PNA Object Model](#)
- [Reading and Writing Calibration data](#)
- [Example Programs](#)
- [Superseded commands](#)

### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CloseCalSet</strong></td>
<td>ICalSet</td>
<td>Obsolete - No longer necessary.</td>
</tr>
<tr>
<td><strong>ComputeErrorTerms</strong></td>
<td>ICalSet</td>
<td>Computes error terms for the CalType specified by a preceding OpenCal Set call.</td>
</tr>
<tr>
<td><strong>Copy</strong></td>
<td>ICalSet</td>
<td>Creates a new Cal Set and copies the current Cal Set data into it.</td>
</tr>
<tr>
<td><strong>EnumerateItems</strong></td>
<td>ICalSet6</td>
<td>Returns a list of all name-value pairs (items) in the Cal Set.</td>
</tr>
<tr>
<td><strong>getErrorTerm</strong></td>
<td>ICalSet</td>
<td>Superseded with <strong>getErrorTermByString</strong></td>
</tr>
<tr>
<td><strong>getErrorTermByString</strong></td>
<td>ICalSet2</td>
<td>Returns variant error term data by specifying the string name of the error term.</td>
</tr>
<tr>
<td>Function</td>
<td>Type</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>getErrorTermList</td>
<td>ICalSet</td>
<td>Superseded with getErrorTermList2</td>
</tr>
<tr>
<td>getErrorTermList2</td>
<td>ICalSet2</td>
<td>Returns a list of error term names found in a calset.</td>
</tr>
<tr>
<td>GetErrorTermStimulus</td>
<td>ICalSet7</td>
<td>Returns the stimulus values over which the specific error term was acquired.</td>
</tr>
<tr>
<td>GetGUID</td>
<td>ICalSet</td>
<td>Returns the GUID identifying a Cal Set</td>
</tr>
<tr>
<td>getStandard</td>
<td>ICalSet</td>
<td>Superseded with getStandardByString</td>
</tr>
<tr>
<td>getStandardByString</td>
<td>ICalSet2</td>
<td>Returns variant standard acquisition data by specifying the string name of the standard.</td>
</tr>
<tr>
<td>getStandardsList</td>
<td>ICalSet</td>
<td>Superseded with getStandardList2</td>
</tr>
<tr>
<td>getStandardsList2</td>
<td>ICalSet2</td>
<td>Returns a list of standard names found in a Cal Set.</td>
</tr>
<tr>
<td>HasCalType</td>
<td>ICalSet</td>
<td>Verifies that the Cal Set object contains the error terms required to apply the specified CalType to an appropriate measurement.</td>
</tr>
<tr>
<td>OpenCalSet</td>
<td>ICalSet</td>
<td>Obsolete - No longer necessary.</td>
</tr>
<tr>
<td>putErrorTerm</td>
<td>ICalSet</td>
<td>Superseded with putErrorTermByString</td>
</tr>
<tr>
<td>putErrorTermByString</td>
<td>ICalSet2</td>
<td>Writes variant error term data by specifying the string name of the error term.</td>
</tr>
<tr>
<td>PutErrorTermStimulus</td>
<td>ICalSet7</td>
<td>Adds stimulus data to the specified buffer.</td>
</tr>
<tr>
<td>putStandard</td>
<td>ICalSet</td>
<td>Superseded with putStandardByString</td>
</tr>
<tr>
<td>putStandardByString</td>
<td>ICalSet2</td>
<td>Writes variant standard acquisition data by specifying the string name of the standard.</td>
</tr>
<tr>
<td>RemoveItem</td>
<td>ICalSet6</td>
<td>Removes a name-value pair from the Cal Set.</td>
</tr>
<tr>
<td>Save</td>
<td>ICalSet</td>
<td>Saves the current Cal Set to disk.</td>
</tr>
<tr>
<td>StringToNACalClass</td>
<td>ICalSet</td>
<td>Converts string values from GetStandardsList into enumeration data</td>
</tr>
<tr>
<td>StringToNAErrorTerm2</td>
<td>ICalSet</td>
<td>Converts string values from GetErrorTermList into enumeration data</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Description</td>
<td>ICAlSet</td>
</tr>
<tr>
<td>-------------</td>
<td>---------</td>
</tr>
<tr>
<td>AlternateSweep</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>Attenuator</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>AttenuatorMode</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>CouplePorts</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>CWFrequency</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>Description</td>
<td>ICAlSet</td>
</tr>
<tr>
<td>DwellTime</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>FrequencyOffsetCWOverride</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>FrequencyOffsetDivisor</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>FrequencyOffsetFrequency</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>FrequencyOffsetMultiplier</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>FrequencyOffsetState</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>IFBandwidth</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>Item</td>
<td>ICAlSet6</td>
</tr>
<tr>
<td>LastModified</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>Name</td>
<td>ICAlSet4</td>
</tr>
<tr>
<td>NumberOfPoints</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>OutputPorts</td>
<td>ICAlSet5</td>
</tr>
<tr>
<td>PowerSlope</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>ReceiverAttenuator</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>StartFrequency</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>StartPower</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>StimulusValues</td>
<td>ICAlSet3</td>
</tr>
<tr>
<td>StopFrequency</td>
<td>ICAlSet3</td>
</tr>
</tbody>
</table>
**StopPower** | ICalSet3 | Returns the stop power of the PNA when sweep type is set to Power Sweep.

**SweepGenerationMode** | ICalSet3 | Returns the method being used to generate a sweep: analog or stepped.

**SweepTime** | ICalSet3 | Returns the sweep time of the analyzer.

**SweepType** | ICalSet3 | Returns the type of X-axis sweep that is performed on a channel.

**TestPortPower** | ICalSet3 | Returns the RF power level for the channel.

**TestSetType** | ICalSet5 | Returns the Test Set type used for the Cal Set.

### ICalSet History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalSet</td>
<td>2.0</td>
</tr>
<tr>
<td>ICalSet2</td>
<td>3.0</td>
</tr>
<tr>
<td>ICalSet3</td>
<td>3.2</td>
</tr>
<tr>
<td>ICalSet4</td>
<td>6.0</td>
</tr>
<tr>
<td>ICalSet5</td>
<td>6.2</td>
</tr>
<tr>
<td>ICalSet6</td>
<td>9.30</td>
</tr>
<tr>
<td>ICalSet7</td>
<td>9.40</td>
</tr>
</tbody>
</table>

### ICalData Interface

**Description**

Use this interface as an alternative to the ICalSet Interface to avoid using variants when transmitting data to and from the Cal Set.

[Learn about reading and writing Calibration data.](#)
<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>get ErrorTermComplex</td>
<td>ICalData2</td>
<td>Superseded with getErrorTermComplexByString</td>
</tr>
<tr>
<td>getErrorTermComplexByString</td>
<td>ICalData3</td>
<td>Returns typed error term data by specifying the string name of the error term.</td>
</tr>
<tr>
<td>getStandardComplex</td>
<td>ICalData2</td>
<td>Superseded with getStandardComplexByString</td>
</tr>
<tr>
<td>getStandardComplexByString</td>
<td>ICalData3</td>
<td>Returns typed standard acquisition data by specifying the string name of the standard.</td>
</tr>
<tr>
<td>putErrorTermComplex</td>
<td>ICalData2</td>
<td>Superseded with putErrorTermComplexByString</td>
</tr>
<tr>
<td>putErrorTermComplexByString</td>
<td>ICalData3</td>
<td>Writes typed error term data by specifying the string name of the error term.</td>
</tr>
<tr>
<td>putStandardComplex</td>
<td>ICalData2</td>
<td>Superseded with putStandardComplexByString</td>
</tr>
<tr>
<td>putStandardComplexByString</td>
<td>ICalData3</td>
<td>Writes typed standard acquisition data by specifying the string name of the standard.</td>
</tr>
</tbody>
</table>

### Properties

None

### History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalData2</td>
<td>2.0</td>
</tr>
<tr>
<td>ICalData3</td>
<td>3.1</td>
</tr>
</tbody>
</table>

The original ICalData Interface was introduced with PNA 1.0 on the Calibrator Object.

Last modified:
2-May-2011  Added Get/Put ErrorTermStimulus
1-Nov-2006  New start and stop freq commands added
CalSets Collection

Description
A collection object that provides a mechanism for iterating through all the Cal Sets in the analyzer. There is no ordering to the items in the collection. Therefore make no assumptions about the formatting of the collection.

For the Item and Remove methods, you can specify either the Cal Set string name, or the integer item of the Cal Set in the collection.

Accessing the CalSets collection
Get a handle to the CalSets collection through the CalManager object with the app.GetCalManager Method.

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim calsts As CalSets
Set calsts = app.GetCalManager.CalSets

See Also:

- CalSet Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exists</td>
<td>ICalSets2</td>
<td>Returns whether the specified Cal Set exists</td>
</tr>
<tr>
<td>Item</td>
<td>ICalSets</td>
<td>Returns a handle to a CalSet object in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>ICalSets</td>
<td>Deletes the Cal Set residing at position index in the collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of Cal Sets in the collection.</td>
</tr>
</tbody>
</table>

CalSets History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalSets</td>
<td>1.0</td>
</tr>
<tr>
<td>ICalSets2</td>
<td>9.33</td>
</tr>
</tbody>
</table>
Last Modified:

28-Feb-2011  Added Exists (9.33)

30-Oct-2007  added item and remove note.
CalStandard Object

Description
Contains all of the settings that are required to modify a calibration standard.
For more information, read Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers (Application Note 1287-11)

Accessing the CalStandard object
Get a handle to a standard with the calkit.GetCalStandard Method.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim std As ICalStandard
Set std = app.ActiveCalKit.GetCalStandard(1)
std.Delay = 0.00000003
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Reading and Writing Calibration data
- Example Programs

Methods
None

Properties Interface Description

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>C0</td>
<td>ICalStandard</td>
<td>Sets and Returns the C0 (C-zero) value (the first capacitance value) for the calibration standard, when the Type is set to &quot;naOpen&quot;.</td>
</tr>
<tr>
<td>C1</td>
<td>ICalStandard</td>
<td>Sets and Returns the C1 value (the second capacitance value) for the calibration standard, when the Type is set to &quot;naOpen&quot;.</td>
</tr>
<tr>
<td>C2</td>
<td>ICalStandard</td>
<td>Sets and Returns the C2 value (the third capacitance value) for the calibration standard, when the Type is set to &quot;naOpen&quot;.</td>
</tr>
<tr>
<td>C3</td>
<td>ICalStandard</td>
<td>Sets and Returns the C3 value (the fourth capacitance value) for the calibration standard, when the Type is set to &quot;naOpen&quot;.</td>
</tr>
<tr>
<td>Delay</td>
<td>ICalStandard</td>
<td>Sets and Returns the electrical delay value for the calibration standard.</td>
</tr>
</tbody>
</table>
**L0** ICalStandard  
Sets and Returns the L0 (L-zero) value (the first inductance value) for the calibration standard, when the Type is set to "naShort".

**L1** ICalStandard  
Sets and Returns the L1 value (the second inductance value) for the calibration standard, when the Type is set to "naShort".

**L2** ICalStandard  
Sets and Returns the L2 value (the third inductance value) for the calibration standard, when the Type is set to "naShort".

**L3** ICalStandard  
Sets and Returns the L3 value (the third inductance value) for the calibration standard, when the Type is set to "naShort".

**Label** ICalStandard  
Sets and Returns the label for the calibration standard.

**loss** ICalStandard  
Sets and Returns the insertion loss for the calibration standard.

**Maximum Frequency** ICalStandard  
Sets and Returns the maximum frequency for the calibration standard.

**Medium** ICalStandard  
Sets and Returns the media type of the calibration standard.

**Minimum Frequency** ICalStandard  
Sets and Returns the minimum frequency for the calibration standard.

**Type** ICalStandard  
Sets and Returns the type of calibration standard. Selections are: naOpen, naShort, naLoad, naThru, naArbitraryImpedance and naSliding.

**TZReal** ICalStandard2  
Sets and Returns the TZReal value (the Real Terminal Impedance value) for the calibration standard, when the Type is set to "naArbitraryImpedance".

**TZImag** ICalStandard2  
Sets and Returns the TZImag value (the Imaginary Terminal Impedance value) for the calibration standard, when the Type is set to "naArbitraryImpedance".

**Z0** ICalStandard  
Sets and Returns the characteristic impedance for the calibration standard.

**ICalStandard History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalStandard</td>
<td>1.0</td>
</tr>
<tr>
<td>CalStandard2</td>
<td>3.0</td>
</tr>
</tbody>
</table>
Capabilities Object

Description
These properties return capabilities of the remote PNA.

Accessing the Capabilities object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim cap As Capabilities
Set cap = app.Capabilities

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- ICapabilities History
- Example Programs

Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GetPortNumber Method</td>
<td>ICapabilities4</td>
<td>Returns the port number for the specified string port name.</td>
</tr>
</tbody>
</table>

Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AvailableMeasurementClasses</td>
<td>ICapabilities7</td>
<td>Returns the measurement classes on the PNA</td>
</tr>
<tr>
<td>CpuRevision</td>
<td>ICapabilities6</td>
<td>Returns the CPU speed of the PNA</td>
</tr>
<tr>
<td>DspRevision</td>
<td>ICapabilities6</td>
<td>Returns the DSP Revision number</td>
</tr>
<tr>
<td>DspFpgaRevision</td>
<td>ICapabilities6</td>
<td>Returns the DSP FPGA Revision number</td>
</tr>
<tr>
<td>FirmwareMajorRevision</td>
<td>ICapabilities</td>
<td>Returns integer portion of firmware revision number.</td>
</tr>
<tr>
<td>FirmwareMinorRevision</td>
<td>ICapabilities</td>
<td>Return decimal portion of firmware revision number.</td>
</tr>
<tr>
<td>FirmwareSeries</td>
<td>ICapabilities</td>
<td>Returns the Alpha portion of the firmware revision number.</td>
</tr>
<tr>
<td>GPIBPortCount</td>
<td>ICapabilities3</td>
<td>Returns the number of GPIB ports (1 or 2)</td>
</tr>
<tr>
<td>Property</td>
<td>Context</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------------------------------</td>
<td>---------------</td>
<td>-------------------------------------------------------</td>
</tr>
<tr>
<td><code>InternalTestsetPortCount</code></td>
<td>ICapabilities</td>
<td>Returns the number of PNA test ports.</td>
</tr>
<tr>
<td><code>IsFrequencyOffsetPresent</code></td>
<td>ICapabilities</td>
<td>Returns the presence of Frequency Offset Option 080 (True or False).</td>
</tr>
<tr>
<td><code>IsReceiverStepAttenuatorPresent</code></td>
<td>ICapabilities</td>
<td>Returns the presence of receiver step attenuators (True or False).</td>
</tr>
<tr>
<td><code>IsReferenceBypassSwitchPresent</code></td>
<td>ICapabilities</td>
<td>Returns the presence of the reference switch (True or False).</td>
</tr>
<tr>
<td><code>MaximumFrequency</code></td>
<td>ICapabilities</td>
<td>Returns the maximum frequency of the PNA.</td>
</tr>
<tr>
<td><code>MaximumNumberOfChannels</code></td>
<td>ICapabilities2</td>
<td>Returns the maximum possible number of Channels</td>
</tr>
<tr>
<td><code>MaximumNumberOfPoints</code></td>
<td>ICapabilities</td>
<td>Returns the maximum possible number of data points.</td>
</tr>
<tr>
<td><code>MaximumNumberOfTracesPerWindow</code></td>
<td>ICapabilities2</td>
<td>Returns the maximum possible number of traces per window</td>
</tr>
<tr>
<td><code>MaximumNumberOfWindows</code></td>
<td>ICapabilities2</td>
<td>Returns the maximum possible number of windows</td>
</tr>
<tr>
<td><code>MaximumReceiverStepAttenuator</code></td>
<td>ICapabilities</td>
<td>Returns the maximum amount of receiver attenuation.</td>
</tr>
<tr>
<td><code>MaximumSourceALCPower</code></td>
<td>ICapabilities</td>
<td>Returns the maximum amount of source ALC power.</td>
</tr>
<tr>
<td><code>MaximumSourceStepAttenuator</code></td>
<td>ICapabilities</td>
<td>Returns the maximum amount of source attenuation.</td>
</tr>
<tr>
<td><code>MeasurementClassProperties</code></td>
<td>ICapabilities8</td>
<td>Returns a handle to the MeasurementClassProperties Object</td>
</tr>
<tr>
<td><code>MinimumFrequency</code></td>
<td>ICapabilities</td>
<td>Returns the minimum frequency of the PNA.</td>
</tr>
<tr>
<td><code>MinimumNumberOfPoints</code></td>
<td>ICapabilities</td>
<td>Returns the minimum possible number of data points.</td>
</tr>
<tr>
<td><code>MinimumReceiverStepAttenuator</code></td>
<td>ICapabilities</td>
<td>Returns the minimum amount of receiver attenuation.</td>
</tr>
<tr>
<td><code>MinimumSourceALCPower</code></td>
<td>ICapabilities</td>
<td>Returns the minimum amount of source ALC power.</td>
</tr>
<tr>
<td><code>ReceiverCount</code></td>
<td>ICapabilities</td>
<td>Returns the number of receivers in the PNA.</td>
</tr>
<tr>
<td><code>ReceiverStepAttenuatorStepSize</code></td>
<td>ICapabilities</td>
<td>Returns the step size of the attenuator.</td>
</tr>
<tr>
<td><code>SourceStepAttenuatorStepSize</code></td>
<td>ICapabilities5</td>
<td>Returns a value indicating the step size of the source attenuator.</td>
</tr>
<tr>
<td><code>SourceCount</code></td>
<td>ICapabilities</td>
<td>Returns the number of sources.</td>
</tr>
<tr>
<td><code>SourcePortCount</code></td>
<td>ICapabilities4</td>
<td>Returns the number of source ports.</td>
</tr>
<tr>
<td><code>SourcePortNames</code></td>
<td>ICapabilities4</td>
<td>Returns the string names of source ports.</td>
</tr>
</tbody>
</table>
ICapabilities5  Returns a value indicating the step size of the source attenuator.

**ICapabilities History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICapabilities</td>
<td>3.5</td>
</tr>
<tr>
<td>ICapabilities2</td>
<td>5.23</td>
</tr>
<tr>
<td>ICapabilities3</td>
<td>6.0</td>
</tr>
<tr>
<td>ICapabilities4</td>
<td>7.20</td>
</tr>
<tr>
<td>ICapabilities5</td>
<td>8.04</td>
</tr>
<tr>
<td>ICapabilities6</td>
<td>9.10</td>
</tr>
<tr>
<td>ICapabilities7</td>
<td>9.33</td>
</tr>
<tr>
<td>ICapabilities8</td>
<td>9.40</td>
</tr>
</tbody>
</table>
Channel Object

See SourcePowerCalData Interface for putting and getting typed source power calibration data.

Description

The channel object is like the engine that produces data. Channel settings consist of stimulus values like frequency, power, IF bandwidth, and number of points.

Accessing the Channel object

You can get a handle to a channel in a number of ways. But first you have to make sure that the channel exists. When you first startup the analyzer, there is one S11 measurement on channel 1. Thus there is only one channel in existence. You can do the following:

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim chan As IChannel
Set chan = app.ActiveChannel
```
or

```vba
Set chan = app.Channels(2)
```

The first method returns the channel object that is driving the active measurement. If there is no measurement, there may not be a channel. Once a channel is created, it does not go away. So if there once was a measurement (hence a channel), the channel will still be available.

If there is no channel you can create one in a couple ways. You can do the following:

```vba
Pna.CreateMeasurement( ch1, "S11", port1, window2)
```
or

```vba
Pna.Channels.Add(2)
```

The latter will have no visible effect on the analyzer. It will simply create channel 2 if it does not already exist.

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
- Superseded commands

(Bold Methods or Properties provide access to a child object)
<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ApplySourcePowerCorrectionTo</td>
<td>IChannel11</td>
<td>Copies an existing Source Power Calibration to another channel.</td>
</tr>
<tr>
<td>AveragingRestart</td>
<td>IChannel</td>
<td>Clears and restarts averaging of the measurement data.</td>
</tr>
<tr>
<td>Continuous</td>
<td>IChannel</td>
<td>The channel continuously responds to trigger signals.</td>
</tr>
<tr>
<td>CopyToChannel</td>
<td>IChannel2</td>
<td>Sets up another channel as a copy of this objects channel.</td>
</tr>
<tr>
<td>GetConverter</td>
<td>IChannel7</td>
<td>Returns a handle to a Converter object.</td>
</tr>
<tr>
<td>GetErrorCorrection</td>
<td>IChannel8</td>
<td>Returns the channel error correction state.</td>
</tr>
<tr>
<td>GetNumberOfGroups</td>
<td>IChannel3</td>
<td>Returns the number of groups a channel has yet to acquire.</td>
</tr>
<tr>
<td>GetPortNumber</td>
<td>IChannel13</td>
<td>Returns the port number for the specified string port name.</td>
</tr>
<tr>
<td>GetRxLevelingConfiguration</td>
<td>IChannel17</td>
<td>Returns a handle to a RxLevelingConfiguration object.</td>
</tr>
<tr>
<td>getSourcePowerCalData</td>
<td>IChannel</td>
<td><strong>Superseded with</strong> Get SourcePowerCalDataEx</td>
</tr>
<tr>
<td>getSourcePowerCalDataEx</td>
<td>IChannel4</td>
<td>Returns requested source power calibration data, if it exists.</td>
</tr>
<tr>
<td>GetSupportedALCModes</td>
<td>IChannel10</td>
<td>Returns a list of supported ALC modes</td>
</tr>
<tr>
<td>GetXAxisValues</td>
<td>IChannel</td>
<td>Returns the channel's X-axis values into a dimensioned Variant array.</td>
</tr>
<tr>
<td>GetXAxisValues2</td>
<td>IChannel</td>
<td>Returns the channel's X-axis values into a dimensioned NON-Variant array.</td>
</tr>
<tr>
<td>Hold</td>
<td>IChannel</td>
<td>Puts the Channel in Hold - not sweeping.</td>
</tr>
<tr>
<td>Next_IFBandwidth</td>
<td>IChannel</td>
<td>A function that returns the Next higher IF Bandwidth value.</td>
</tr>
<tr>
<td>NumberOfGroups</td>
<td>IChannel</td>
<td>Sets the Number of trigger signals the channel will receive.</td>
</tr>
<tr>
<td>Preset</td>
<td>IChannel</td>
<td>Resets the channel to factory defined settings.</td>
</tr>
<tr>
<td>PreviousIFBandwidth</td>
<td>IChannel</td>
<td>Returns the previous IF Bandwidth value.</td>
</tr>
<tr>
<td>putSourcePowerCalData</td>
<td>IChannel</td>
<td><strong>Superseded with</strong> Put SourcePowerCalDataEx Method</td>
</tr>
<tr>
<td>putSourcePowerCalDataEx</td>
<td>IChannel4</td>
<td>Inputs source power calibration data to this channel for a specific source port.</td>
</tr>
<tr>
<td>SelectCalSet</td>
<td>IChannel</td>
<td>Specifies the Cal Set to use for the Channel</td>
</tr>
<tr>
<td>Single</td>
<td>IChannel</td>
<td>Channel responds to one trigger signal from any source (internal, external, or manual). Then channel switches to Hold.</td>
</tr>
<tr>
<td>Method</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>ALCLevelingMode</td>
<td>Set or return the ALC leveling mode.</td>
<td></td>
</tr>
<tr>
<td>AlternateSweep</td>
<td>Sets sweeps to either alternate or chopped.</td>
<td></td>
</tr>
<tr>
<td>Attenuator</td>
<td>Sets or returns the value of the attenuator control for the specified port number.</td>
<td></td>
</tr>
<tr>
<td>AttenuatorMode</td>
<td>Sets or returns the mode of operation of the attenuator control for the specified port number.</td>
<td></td>
</tr>
<tr>
<td>AuxiliaryTrigger</td>
<td>Provides access to Auxiliary Triggering.</td>
<td></td>
</tr>
<tr>
<td>Averaging</td>
<td>Turns trace averaging ON or OFF for all measurements on the channel.</td>
<td></td>
</tr>
<tr>
<td>AveragingCount</td>
<td>Returns the number of sweeps that have been averaged into the measurements.</td>
<td></td>
</tr>
<tr>
<td>AveragingFactor</td>
<td>Specifies the number of measurement sweeps to combine for an average.</td>
<td></td>
</tr>
<tr>
<td>AverageMode</td>
<td>Sets Point or Sweep averaging.</td>
<td></td>
</tr>
<tr>
<td>BalancedTopology</td>
<td>Provides access to the topology of a balanced DUT.</td>
<td></td>
</tr>
<tr>
<td>Calibrator</td>
<td>Provides access to Unguided calibration.</td>
<td></td>
</tr>
<tr>
<td>CalSet</td>
<td>Provides access to the contents of a Cal Set</td>
<td></td>
</tr>
<tr>
<td>centerFrequency</td>
<td>Sets or returns the center frequency of the channel.</td>
<td></td>
</tr>
<tr>
<td>channelNumber</td>
<td>Returns the Channel number.</td>
<td></td>
</tr>
<tr>
<td>Converter</td>
<td>Provides access to a mixer/converter object.</td>
<td></td>
</tr>
<tr>
<td>CorrectionMethods</td>
<td>Provides access to channel correction properties.</td>
<td></td>
</tr>
<tr>
<td>CoupleChannelParams</td>
<td>Turns ON and OFF Time Domain Trace Coupling.</td>
<td></td>
</tr>
<tr>
<td>CouplePorts</td>
<td>Turns ON and OFF port power coupling.</td>
<td></td>
</tr>
<tr>
<td>CustomChannelConfiguration</td>
<td>Provides access to custom application objects.</td>
<td></td>
</tr>
<tr>
<td>CWFrequency</td>
<td>Set the Continuous Wave (CW) frequency.</td>
<td></td>
</tr>
<tr>
<td>DefinedRoles</td>
<td>Returns the roles for which sources can be used for the channel.</td>
<td></td>
</tr>
</tbody>
</table>
**DwellTime**  
IChannel  
Sets or returns the dwell time for the channel.  
Shared with the Segment Object

**ErrorCorrection**  
IChannel7  
Attempts to set error correction ON or OFF for all of the measurements on the channel.

**ExternalTriggerDelay**  
IChannel6  
Sets or returns the external trigger delay value for the channel.

**FastCWPointCount**  
IChannel16  
Enables Fast CW sweep and sets the number of data points for the channel.

**Fixturing**  
IChannel6  
Provides access to Port Ext, Embedding, and De-embedding functions.

**FOM Collection**  
IChannel9  
Provides access to Frequency Offset Measurements

**FrequencyOffsetDivisor**  
IChannel2

**FrequencyOffsetFrequency**  
IChannel2

**FrequencyOffsetMultiplier**  
IChannel2  
Superseded with FOM and FOMRange

**FrequencyOffsetCWOverride**  
IChannel2

**FrequencyOffsetState**  
IChannel2

**FrequencySpan**  
IChannel  
Sets or returns the frequency span of the channel.  
Shared with the Segment Object.

**IFBandwidth**  
IChannel  
Sets or returns the IF Bandwidth of the channel.  
Shared with the Segment Object.

**IFConfiguration**  
IChannel4  
Provides access to the IF gain and source path settings for the H11 Option.

**IsContinuous**  
IChannel3  
Returns whether or not a channel is in continuous mode.

**IsHold**  
IChannel3  
Returns whether or not a channel is in hold mode.

**MeasurementClass**  
IChannel15  
Returns the measurement class name.

**NumberOfPoints**  
IChannel  
Sets or returns the Number of Points of the channel.  
Shared with the Segment Object.

**Parent**  
IChannel  
Returns a handle to the parent object of the channel.

**PathConfiguration**  
IChannel10  
Provides access to path configuration switches and setting.
<table>
<thead>
<tr>
<th>Class</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PathConfigurationManager</td>
<td>IChannel10</td>
<td>Provides access to path configuration file management.</td>
</tr>
<tr>
<td>PointSweepState</td>
<td>IChannel16</td>
<td>Turns point sweep ON or OFF for all measurements on the channel.</td>
</tr>
<tr>
<td>PowerSlope</td>
<td>IChannel</td>
<td>Sets or returns the Power Slope value.</td>
</tr>
<tr>
<td>PowerSlopeState</td>
<td>IChannel18</td>
<td>Turns power slope ON or OFF.</td>
</tr>
<tr>
<td>PulseGenerator</td>
<td>IChannel10</td>
<td>Provides access to pulse generator configuration.</td>
</tr>
<tr>
<td>PulseMeasurementControl</td>
<td>IChannel20</td>
<td>Provides access to pulse measurement settings.</td>
</tr>
<tr>
<td>R1InputPath</td>
<td>IChannel2</td>
<td>Throws internal reference switch (option 081).</td>
</tr>
<tr>
<td>ReceiverAttenuator</td>
<td>IChannel</td>
<td>Sets or returns the value of the specified receiver attenuator control.</td>
</tr>
<tr>
<td>ReduceIFBandwidth</td>
<td>IChannel5</td>
<td>Sets or returns the state of the Reduced IF Bandwidth at Low Frequencies setting.</td>
</tr>
<tr>
<td>RoleDevice</td>
<td>IChannel22</td>
<td>Sets and returns the source to be used in the specified role.</td>
</tr>
<tr>
<td>RXLevelingConfiguration</td>
<td>IChannel21</td>
<td>Provides access to the ReceiverLeveling Object</td>
</tr>
<tr>
<td>Segments</td>
<td>IChannel</td>
<td>Provides access to the Collection for iterating through the sweep segments of a channel.</td>
</tr>
<tr>
<td>SourcePortCount</td>
<td>IChannel13</td>
<td>Returns the number of source ports.</td>
</tr>
<tr>
<td>SourcePortMode</td>
<td>IChannel9</td>
<td>Sets the state of the PNA sources. (AUTO</td>
</tr>
<tr>
<td>SourcePortNames</td>
<td>IChannel13</td>
<td>Returns the string names of source ports.</td>
</tr>
<tr>
<td>SourcePowerCalPowerOffset</td>
<td>IChannel4</td>
<td>Sets or returns a power level offset from the PNA test port power.</td>
</tr>
<tr>
<td>SourcePowerCorrection</td>
<td>IChannel</td>
<td>Turns source power correction ON or OFF for a specific source port.</td>
</tr>
<tr>
<td>StartFrequency</td>
<td>IChannel</td>
<td>Sets or returns the start frequency of the channel.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the Segment Object</td>
</tr>
<tr>
<td>StartPower</td>
<td>IChannel</td>
<td>Sets the start power of the analyzer when sweep type is set to Power Sweep.</td>
</tr>
<tr>
<td>StartPowerEx</td>
<td>IChannel13</td>
<td>Sets and reads the power sweep start power value for a specific port.</td>
</tr>
</tbody>
</table>
**StopFrequency**  
*IChannel*  
Sets or returns the stop frequency of the channel.  
Shared with the Segment Object

**StopPower**  
*IChannel*  
Sets the Stop Power of the analyzer when sweep type is set to Power Sweep.

**StopPowerEx**  
*IChannel13*  
Sets and reads the power sweep stop power value for a specific port.

**SweepDelay**  
*IChannel19*  
Sets the time to wait just before acquisition begins for each sweep.

**SweepGenerationMode**  
*IChannel*  
Sets the method used to generate a sweep: continuous ramp (analog) or discrete steps (stepped).

**SweepSpeedMode**  
*IChannel14*  
Set or returns the sweep speed mode.

**SweepTime**  
*IChannel*  
Sets the Sweep time of the analyzer.

**SweepType**  
*IChannel*  
Sets the type of X-axis sweep that is performed on a channel.

**TestPortPower**  
*IChannel*  
Sets or returns the RF power level for the channel.  
Shared with the Segment Object

**TriggerMode**  
*IChannel*  
Determines the measurement that occurs when a trigger signal is sent to the channel.

**UserRangeMax**  
*IChannel*  
*Superseded* - Use meas.**UserRangeMax**  
Sets the stimulus stop value for the specified User Range.

**UserRangeMin**  
*IChannel*  
*Superseded* - Use meas.**UserRangeMin**  
Sets the stimulus start value for the specified User Range.

**XAxisPointSpacing**  
*IChannel*  
Sets X-Axis point spacing for the active channel.

**IChannel History**
### ISourcePowerCalData Interface

**Description**
Contains methods for putting source power calibration data in and getting source power calibration data out of the analyzer using typed data. The methods in this interface transfer data more efficiently than methods that use...
variant data. However, this interfaces is only usable from VB6, C, & C++. All other programming languages must use the methods on the Channel Object.

**Note:** Interface **ISourcePowerCalData** is abbreviated as **ISPCD** in the following table.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>getSourcePowerCalDataScalar</td>
<td>ISPCD</td>
<td>Superseded - use PutSourcePowerCalDataScalarEx Method</td>
</tr>
<tr>
<td>getSourcePowerCalDataScalarEx</td>
<td>ISPCD2</td>
<td>Returns requested source power calibration data, if it exists.</td>
</tr>
<tr>
<td>putSourcePowerCalDataScalar</td>
<td>ISPCD</td>
<td>Superseded - use PutSourcePowerCalDataEx Method</td>
</tr>
<tr>
<td>putSourcePowerCalDataScalarEx</td>
<td>ISPCD2</td>
<td>Inputs source power calibration data to a channel, for a specific source port.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

**ISourcePowerCalData History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISourcePowerCalData</td>
<td>2.0</td>
</tr>
<tr>
<td>ISourcePowerCalData2</td>
<td>4.0</td>
</tr>
</tbody>
</table>
Channels Collection

Description
A collection object that provides a mechanism for iterating through the channels.
Collections are, by definition, unordered lists of like objects. You cannot assume that Channels.Item(1) is always Channel 1.

Accessing the Channels collection

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim chans As Channels
Set chans = app.Channels
```

See Also:
- Channel Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>IChannels</td>
<td>An alternate way to create a measurement.</td>
</tr>
<tr>
<td>Hold</td>
<td>IChannels</td>
<td>Places all channels in Hold trigger mode.</td>
</tr>
<tr>
<td>Item</td>
<td>IChannels</td>
<td>Use to get a handle to a channel in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>IChannels3</td>
<td>Delete a channel by specifying the index in the collection.</td>
</tr>
<tr>
<td>RemoveChannelNumber</td>
<td>IChannels3</td>
<td>Delete a channel by specifying the channel number.</td>
</tr>
<tr>
<td>Resume</td>
<td>IChannels2</td>
<td>Resumes the trigger mode of all channels that was in effect before sending the channels.Hold method.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of channels in the analyzer.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current Application.</td>
</tr>
</tbody>
</table>
**UnusedChannelNumbers**

**IChannels2**

Returns an array of channel numbers that are NOT in use.

**UsedChannelNumbers**

**IChannels2**

Returns an array of channel numbers that are in use.

**IChannels History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IChannels</td>
<td>1.0</td>
</tr>
<tr>
<td>IChannels2</td>
<td></td>
</tr>
<tr>
<td>IChannels3</td>
<td>9.30</td>
</tr>
</tbody>
</table>
ComColors Object

Description
Provides access to the methods and properties used to modify the PNA Display and Print colors.

Accessing the ComColors object

```vba
Dim app As AgilentPNA835X.Application
Set app = CreateObject("AgilentPNA835X.Application")
Set displayColors = app.Preferences.DisplayColors
' or
'Set printColors = app.Preferences.PrintColors
displayColors.ActiveLabels = 657930
```

See Also:
- ComTraceColors Object
- Modify Display Colors Example
- About PNA Display Colors
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LoadTheme</td>
<td>IComColors</td>
<td>Load a color theme from a disc file.</td>
</tr>
<tr>
<td>ResetTheme</td>
<td>IComColors</td>
<td>Resets the current theme to the default PNA colors.</td>
</tr>
<tr>
<td>StoreTheme</td>
<td>IComColors</td>
<td>Saves the current color theme to a disc file.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActiveLabels</td>
<td>Sets labels and grid frame colors in the active window.</td>
</tr>
<tr>
<td>ActiveBackground</td>
<td>Set and return the background color for the active window on the PNA display or hardcopy print.</td>
</tr>
<tr>
<td>Background</td>
<td>Set and return the background color for the inactive windows on the PNA display or hardcopy print.</td>
</tr>
</tbody>
</table>
FailedTraces  IComColors  Set and return the limit line color of failed traces.

Grid  IComColors  Set and return the inner lines of all grid in all windows.

InactiveLabels  IComColors  Set and return the Inactive (not selected) Window Labels.

Trace  IComColors  Provides access to the ComTraceColors Object for setting colors for the first 8 traces

IComColors History

<table>
<thead>
<tr>
<th>I Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IComColors</td>
<td>9.0</td>
</tr>
<tr>
<td>IComColors2</td>
<td>9.2</td>
</tr>
</tbody>
</table>

Last Modified:

7-Aug-2009      MX New topic
ComTraceColors Object

Description
Provides access to the methods and properties used to modify the PNA Display and Print colors.

Both the Display and Print ComColor objects contain 8 Trace objects (1 to 8).

'1st Trace' is NOT always Trace1 (Tr1). For example, the first trace in a window might be Tr2 which is drawn with the "1st Trace" pen.

The first 8 traces are drawn with the defined pen colors. The next eight traces reuse the same colors, and so forth. For example, if all traces are numbered sequentially, the 9th and 17th traces are drawn using the same color as the 1st trace.

Accessing the ComTraceColors object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835X.Application")
Set displayColors = app.Preferences.DisplayColors
'or
'Set printColors = app.Preferences.PrintColors
dim Trace1

Set Trace1 = displayColors.Trace(1)
Trace1.DataAndLimits = RGB(1,251,1)

See Also:
- Modify Display Colors Example
- About PNA Display Colors
- PNA Automation Interfaces
- The PNA Object Model
- IComColors History
- Example Programs
### Methods Interface

#### See History

None

### Properties Description

- **DataAndLimits** IComTraceColors Set and return the color of Data and Limit Lines for nth trace in a window.
- **Markers** IComTraceColors Set and return the color of data trace markers for nth trace in a window.
- **Memory** IComTraceColors Set and return the memory trace color for nth trace in a window.
- **MemoryMarkers** IComTraceColors Set and return the color of memory trace markers for nth trace in a window.

### IComTraceColors History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IComTraceColors</td>
<td>9.0</td>
</tr>
</tbody>
</table>
Converter Object

**Note:** The Converter Object replaces the IMixer Interface.

**Description**
Contains the methods and properties to setup a mixer for ALL PNA Mixer/Converter applications.

**Accessing the Converter Interface**

```vbscript
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim converter as Converter
Set converter = chan.Converter
```

**Scratch vs Applied Mixer Properties**
Each mixer configuration has two sets of properties:

1. **Scratch mixer** contains the properties that have been set, but NOT YET applied. Send the Apply Method to copy these properties to the Applied mixer.

2. **Applied mixer** contains the properties that makeup the current mixer configuration.

Power settings are immediately applied to both the Scratch and Applied mixer.
A successful Calculate also performs an Apply.

**Note:** Queries always return the Applied mixer properties. Therefore, first send Apply Method before querying new settings.

**See Also**
- PNA Automation Interfaces
- The PNA Object Model
- Mixer Setup UI topic

**Methods**

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AddSegment</td>
<td>Converter5</td>
<td>Add segments to the segment table (<strong>FCA Only</strong>)</td>
</tr>
<tr>
<td>Command</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>-------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td><strong>Apply</strong></td>
<td>Converter Applies mixer settings.</td>
<td></td>
</tr>
<tr>
<td><strong>AssignSourceToRole</strong></td>
<td>Converter <strong>Superseded</strong> - Use RoleDevice Property</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Assigns a configured source to the specified role.</td>
<td></td>
</tr>
<tr>
<td><strong>Calculate</strong></td>
<td>Converter Automatically calculate Input and Output frequencies for mixer setup.</td>
<td></td>
</tr>
<tr>
<td><strong>DeleteAllSegments</strong></td>
<td>Converter5 Remove all segments from the segment table. <strong>(FCA Only)</strong></td>
<td></td>
</tr>
<tr>
<td><strong>DeleteSegment</strong></td>
<td>Converter5 Remove segments from the segment table. <strong>(FCA Only)</strong></td>
<td></td>
</tr>
<tr>
<td><strong>DiscardChanges</strong></td>
<td>Converter Cancels changes that have been made to the Converter setup.</td>
<td></td>
</tr>
<tr>
<td><strong>GetSourceByRole</strong></td>
<td>Converter <strong>Superseded</strong> - Use DefinedRoles Property</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Returns the name of a source that is assigned to the specified role.</td>
<td></td>
</tr>
<tr>
<td><strong>GetSourceRoles</strong></td>
<td>Converter <strong>Superseded</strong> - Use DefinedRoles Property</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Returns the defined role names (&quot;RF2&quot;, &quot;LO1&quot;).</td>
<td></td>
</tr>
<tr>
<td><strong>LoadFile</strong></td>
<td>Converter Loads a previously-configured mixer attributes file <strong>.mxr</strong></td>
<td></td>
</tr>
<tr>
<td><strong>ReCalculate</strong></td>
<td>Converter5 Repeats the last calculation that was performed.</td>
<td></td>
</tr>
<tr>
<td><strong>SaveFile</strong></td>
<td>Converter Saves the settings for the mixer/converter test setup to a mixer attributes file.</td>
<td></td>
</tr>
<tr>
<td><strong>SegmentCalculate</strong></td>
<td>Converter5 Performs calculate on a specific segment. <strong>(FCA Only)</strong></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActiveXAxisRange</td>
<td>Converter Sets or returns the swept frequency range to display on the X-axis.</td>
</tr>
<tr>
<td>AvoidSpurs</td>
<td>Converter5 Sets and returns the state of the avoid spurs feature.</td>
</tr>
<tr>
<td><strong>ConverterEmbeddedLO</strong></td>
<td>Converter2 Provides access to the ConverterEmbeddedLO Object</td>
</tr>
<tr>
<td>DeviceInputPort</td>
<td>Converter6 Read the PNA port number which is connected to the DUT Input.</td>
</tr>
</tbody>
</table>
DeviceOutputPort  Converter6  Read the PNA port number which is connected to the DUT Output.

EnablePhase  Converter3  Sets and returns the state of SMC + Phase measurements and calibrations. *(SMC Only)*

IFDenominator  Converter4  Sets or returns the denominator value of the IF Fractional Multiplier.

IFNumerator  Converter4  Sets or returns the numerator value of the IF Fractional Multiplier.

IFSideband  Converter4  Select sum or difference for IF product.

IFStartFrequency  Converter4  Sets or returns the IF start frequency.

IFStopFrequency  Converter4  Sets or returns the IF stop frequency.

IncludeReverseSweep  Converter4  Sets whether to include SC12 sweeps during measurements.

InputDenominator  Converter  Sets or returns the denominator value of the Input Fractional Multiplier.

InputFixedFrequency  Converter  Sets or returns the mixer fixed Input frequency value.

InputNumerator  Converter  Sets or returns the numerator value of the Input Fractional Multiplier.

InputPower  Converter  Sets or returns the value of the Input Power.

InputRangeMode  Converter  Sets or returns the Input sweep mode.

InputStartFrequency  Converter  Sets or returns the start frequency of the mixer input.

InputStartPower  Converter4  Sets and returns the Start Power value of the mixer Input Power.

InputStopFrequency  Converter  Sets or returns the stop frequency of the mixer input.

InputStopPower  Converter4  Sets and returns the Stop Power value of the mixer Input Power.

IsInputGreaterThanLO  Converter  Specifies whether to use the Input frequency that is greater than the LO or less than the LO.

LODenominator  Converter  Sets or returns the denominator value of the LO Fractional Multiplier.
<table>
<thead>
<tr>
<th>Method Name</th>
<th>Converter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOFixedFrequency</td>
<td>Converter</td>
<td>Sets or returns the fixed frequency of the specified LO.</td>
</tr>
<tr>
<td>LOName</td>
<td>Converter</td>
<td>Sets or returns the LO name.</td>
</tr>
<tr>
<td>LONumerator</td>
<td>Converter</td>
<td>Sets or returns the numerator value of the LO Fractional Multiplier.</td>
</tr>
<tr>
<td>LOPower</td>
<td>Converter</td>
<td>Sets or returns the value of the LO Power.</td>
</tr>
<tr>
<td>LORangeMode</td>
<td>Converter</td>
<td>Sets or returns the LO sweep mode to fixed or swept.</td>
</tr>
<tr>
<td>LOStage</td>
<td>Converter</td>
<td>Returns the number of stages.</td>
</tr>
<tr>
<td>LOStartFrequency</td>
<td>Converter</td>
<td>Sets or returns the start frequency of the specified LO.</td>
</tr>
<tr>
<td>LOStartPower</td>
<td>Converter</td>
<td>Sets or returns the start value of a LO Power sweep.</td>
</tr>
<tr>
<td>LOStopFrequency</td>
<td>Converter</td>
<td>Sets or returns the start frequency of the specified LO.</td>
</tr>
<tr>
<td>LOStopPower</td>
<td>Converter</td>
<td>Sets or returns the stop value of a LO Power sweep.</td>
</tr>
<tr>
<td>NominalIncidentPowerState</td>
<td>Converter3</td>
<td>Sets or returns whether to use nominal power or measure actual incident power. (SMC ONLY)</td>
</tr>
<tr>
<td>NormalizePoint</td>
<td>Converter3</td>
<td>Sets or returns the data point used for normalizing an SMC phase measurement. (SMC Only)</td>
</tr>
<tr>
<td>OutputFixedFrequency</td>
<td>Converter</td>
<td>Sets or returns the fixed frequency of the mixer output.</td>
</tr>
<tr>
<td>OutputRangeMode</td>
<td>Converter</td>
<td>Sets or returns the Output sweep mode.</td>
</tr>
<tr>
<td>OutputSideband</td>
<td>Converter</td>
<td>Sets or returns the value of the output sideband.</td>
</tr>
<tr>
<td>OutputStartFrequency</td>
<td>Converter</td>
<td>Sets or returns the start frequency of the mixer output.</td>
</tr>
<tr>
<td>OutputStopFrequency</td>
<td>Converter</td>
<td>Sets or returns the stop frequency of the mixer output.</td>
</tr>
<tr>
<td>SegmentCount</td>
<td>Converter5</td>
<td>Read the number of segments. (FCA Only)</td>
</tr>
</tbody>
</table>
SegmentFixedFrequency Converter5 Set and read the CW Frequency for mixer segments in CW Sweep mode. (FCA Only)

SegmentFixedPower Converter5 Set and return the fixed power level for all ranges for mixer segments. (FCA Only)

SegmentIFBandwidth Converter5 Set and return the IF Bandwidth for the sweep segment. (FCA Only)

SegmentIsInputGreaterThanLO Converter5 Set and return whether to use the Input frequency that is greater than the LO. (FCA Only)

SegmentMixingMode Converter5 Set and return whether to set the mixing mode to high side or low side. (FCA Only)

SegmentPoints Converter5 Sets and returns the number of data points to be measured in the sweep segment. (FCA Only)

SegmentRangeMode Converter5 Sets or returns the sweep mode of the segment (all ranges). (FCA Only)

SegmentStartFrequency Converter5 Set and return the Start frequency for mixer segments. (FCA Only)

SegmentState Converter5 Set and return the ON/OFF state for mixer segments. (FCA Only)

SegmentStopFrequency Converter5 Set and return the Stop frequency for mixer segments. (FCA Only)

**Converter History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Converter</td>
<td>8.55</td>
</tr>
<tr>
<td>Converter2</td>
<td>9.0</td>
</tr>
<tr>
<td>Converter3</td>
<td>9.2</td>
</tr>
<tr>
<td>Converter4</td>
<td>9.30</td>
</tr>
<tr>
<td>Converter5</td>
<td>9.33</td>
</tr>
<tr>
<td>Converter6</td>
<td>9.40</td>
</tr>
</tbody>
</table>
ConverterEmbeddedLO Object

Description
Provides access to the properties that allow IMDx and IMSpectrum measurements of converters that contain an embedded LO.

This interface contains all the same properties and methods of the Embedded LO interface (used for FCA measurements) EXCEPT access to the EmbeddedLODiagnostic Object.

Accessing the ConverterEmbeddedLO Interface

<table>
<thead>
<tr>
<th>Access the Interface through the Converter Object.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dim app</td>
</tr>
<tr>
<td>Set app = CreateObject(&quot;AgilentPNA835x.Application&quot;)</td>
</tr>
<tr>
<td>app.Reset</td>
</tr>
<tr>
<td>' Create a Measurement object, in this case using the IMeasurement interface</td>
</tr>
<tr>
<td>Dim meas</td>
</tr>
<tr>
<td>app.CreateCustomMeasurement 1, &quot;Swept IMD Converters&quot;, -1</td>
</tr>
<tr>
<td>set meas = app.activemeasurement</td>
</tr>
<tr>
<td>dim converter</td>
</tr>
<tr>
<td>set converter = app.ActiveChannel.GetConverter</td>
</tr>
<tr>
<td>dim elo</td>
</tr>
<tr>
<td>set elo = converter.ConverterEmbeddedLO</td>
</tr>
<tr>
<td>elo.IsOn = 1</td>
</tr>
</tbody>
</table>

See Also:
PNA Automation Interfaces
The PNA Object Model
Making Embedded LO Measurements
### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ResetLOFrequency</td>
<td>ICELO</td>
<td>Reset LO Delta frequency.</td>
</tr>
<tr>
<td>ResetTuningParameters</td>
<td>ICELO</td>
<td>Resets the tuning parameters to their defaults.</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BroadbandTuningSpan</td>
<td>ICELO</td>
<td>Set broadband sweep span.</td>
</tr>
<tr>
<td>IsOn</td>
<td>ICELO</td>
<td>Set and return Embedded LO ON</td>
</tr>
<tr>
<td>LOFrequencyDelta</td>
<td>ICELO</td>
<td>Sets and returns LO delta frequency.</td>
</tr>
<tr>
<td>MaxPreciseTuningIterations</td>
<td>ICELO</td>
<td>Sets and returns precise tuning iterations.</td>
</tr>
<tr>
<td>NormalizePoint</td>
<td>ICELO</td>
<td>Sets and returns tuning point.</td>
</tr>
<tr>
<td>PreciseTuningTolerance</td>
<td>ICELO</td>
<td>Sets and returns precise tuning tolerance.</td>
</tr>
<tr>
<td>TuningIFBW</td>
<td>ICELO</td>
<td>Sets and returns the IF Bandwidth for tuning sweeps.</td>
</tr>
<tr>
<td>TuningMode</td>
<td>ICELO</td>
<td>Sets and returns the method used to determine the embedded LO Frequency.</td>
</tr>
<tr>
<td>TuningSweepInterval</td>
<td>ICELO</td>
<td>Set how often a tuning sweep is performed.</td>
</tr>
</tbody>
</table>

### ICELO History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICELO</td>
<td>9.00</td>
</tr>
</tbody>
</table>
CorrectionMethods Object

Description
These methods and properties control various error-correction settings for a channel.

Accessing the object

```vba
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim corrMethods as CorrectionMethods
corrMethods = chan.CorrectionMethods
```

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MatchCorrectPower</td>
<td>ICorrectionMethods</td>
</tr>
</tbody>
</table>

IAuxTrigger History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICorrectionMethods</td>
<td>9.30</td>
</tr>
</tbody>
</table>
E5091Testsets Collection

Description
Two testsets can be connected and controlled by the PNA at any time.

The item number in the testsets collection is set by the DIP switches on the testset rear-panel. The valid item numbers are 1 and 2. If the testset DIP switches are set to 1, then item number in the collection is 1, and so forth. See your E5091A documentation for more information.

If the specified testset is not connected to USB or not ON, then setting Enabled = True will return an error. All other properties can be set when the testset is not connected.

Accessing the E5091Testsets collection
Child of the Application Object. Get a handle to one of the E5091Testset objects by specifying an item of the collection.

```
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Dim testsets As E5091Testsets
Set testsets = pna.E5091Testsets
Dim tset1 As E5091Testset
Set tset1 = testsets(1)
```

See Also:
- E5091Testset Control COM Example
- E5091Testset Object
- Collections in the Analyzer
- The PNA Object Model

Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item</td>
<td>Use to get a handle to a testset in the collection.</td>
</tr>
</tbody>
</table>

Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of items in a collection of objects.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current naNetworkAnalyzer application.</td>
</tr>
</tbody>
</table>

E5091Testsets History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IE5091Testsets</td>
<td>5.2</td>
</tr>
</tbody>
</table>
**E5091Testset Object**

**Description**

There can be two test sets connected and controlled by the PNA at any time.

The item number in the testsets collection is set by the DIP switches on the test set rear-panel. The valid item numbers are 1 and 2. If the test set DIP switches are set to 1, then item number in the collection is 1, and so forth. See your E5091A documentation for more information.

If the specified test set is not connected to USB or not ON, then setting Enabled = True will return an error. All other properties can be set when the test set is not connected.

**Accessing the E5091Testset object**

Child of the Application Object. Get a handle to a E5091Testset object by specifying an item of the collection.

```vba
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Dim testsets As E5091Testsets
Set testsets = pna.E5091Testsets
Dim tset1 As E5091Testset
Set tset1 = testsets(1)
```

**See Also:**

- E5091Testset Control COM Example
- E5091 TestSet Control
- E5091Testsets Collection
- TestsetControl Object (for different test sets)
- The PNA Object Model

**Methods**

None

**Properties**

<table>
<thead>
<tr>
<th>Description</th>
<th>ControlLines</th>
<th>Sets the control lines of the specified E5091A.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enabled</td>
<td>Enables and disables (ON/OFF) the port mapping and control line output of the specified testset.</td>
<td></td>
</tr>
<tr>
<td>ID</td>
<td>Returns the test set ID number.</td>
<td></td>
</tr>
<tr>
<td>NumberOfPorts</td>
<td>Reads the number of ports (7 or 9) that are on the specified E5091A test set.</td>
<td></td>
</tr>
</tbody>
</table>
**OutputPort**

Switches an input to one of the valid outputs on the specified E5091A.

**ShowProperties**

Turns ON and OFF the display of the test set control status bar.

---

**E5091Testset History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IE5091Testset</td>
<td>5.2</td>
</tr>
</tbody>
</table>
**ECalModule Object**

Allows access to ECal modules that are connected to the PNA.

**Accessing the ECalModule object**

Get a handle to a ECalModule object by using the ECalModules collection. This is done through the CalManager object with the app.GetCalManager Method.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim pna
pna.Preset
Const chanNum = 1
pna.Channels(chanNum).StopFrequency = 20E9 ' for a 20 GHz ECal mod
Const pnaPortNumber = 1
Const ecalCharacterizationNum = 0
Dim calMgr
Set calMgr = pna.GetCalManager
Dim ecalPortNumber ' The returned ECal port number is a 1-based number
' (1 = Port A, 2 = Port B, etc)
ecalPortNumber = calMgr.ECalModules(1).AutoOrient(chanNum, pnaPortNumber, ecalCharacterizationNum)
MsgBox "ECal port number attached to PNA port 1 = " & ecalPortNumber
```

**See Also:**

- ECalModules Collection
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoOrient</td>
<td>IECalModule</td>
<td>Returns the orientation (which ECal port is connected to which PNA port) outside of the context of a calibration.</td>
</tr>
</tbody>
</table>

### Properties

- None

### IECalModule History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IECalModule</td>
<td>8.50</td>
</tr>
</tbody>
</table>

**Last Modified:**

- 5-Mar-2009  MX New topic
ECalModules Collection

Description
A collection that provides access to ECal modules that are connected to the PNA.

Accessing the ECalModules collection

```
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim eCalMods As ECalModules
Set eCalMods = app.GetCalManager.ECalModules
```

See Also:
- Using ECal
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Methods

- **Item**
  
  Use to get a handle to a [ECalModule Object](#) in the collection.

Properties

<table>
<thead>
<tr>
<th>Description</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns the number of objects in the collection.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Description</th>
<th>Parent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns a handle to the CalManager Object</td>
<td></td>
</tr>
</tbody>
</table>

IECalModules History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IECalModules</td>
<td>8.50</td>
</tr>
</tbody>
</table>

Last Modified:

7-Apr-2009   MX New topic
**ECalUserCharacterizer Object**

**Description**
Controls the settings used to perform an ECal User Characterization. An S-Parameter channel must already be calibrated. These commands will then measure the ECal module with adapters, cables, or fixtures to be included in the User Characterization, allow descriptive text to be entered, then save the User Characterization to the ECal module.

Up to 12 User Characterizations can be stored in an ECal module.

You can NOT perform a **remote** User Characterization of a 4-port ECal module using a 2-port PNA. This can only be done from the front panel user interface.

**Accessing the ECalUserCharacterizer Interface**
Access the Interface through the ICalManager Object.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")

Dim mgr as ICalManager
Set mgr = app.GetCalManager
Dim ecalCharacterizer
Set ecalCharacterizer = mgr.GetECalUserCharacterizer()
```

**See Also:**
- **Example**: Perform an ECal User Characterization

**PNA Automation Interfaces**
The PNA Object Model
About User Characterization

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcquireStep</td>
<td>ECalUserCharacterizer</td>
<td>Measure the ECal module.</td>
</tr>
<tr>
<td>GenerateSteps</td>
<td>ECalUserCharacterizer</td>
<td>Returns the number of steps required to complete the calibration.</td>
</tr>
<tr>
<td>GetStepDescription</td>
<td>ECalUserCharacterizer</td>
<td>Returns the description of the specified step in the calibration process.</td>
</tr>
<tr>
<td>Initialize</td>
<td>ECalUserCharacterizer</td>
<td><strong>Superseded</strong> with InitializeEx Method</td>
</tr>
<tr>
<td>InitializeEx</td>
<td>ECalUserCharacterizer2</td>
<td>Initiates a User Characterization of an ECal module.</td>
</tr>
<tr>
<td>----------------------</td>
<td>------------------------</td>
<td>------------------------------------------------------</td>
</tr>
<tr>
<td>SaveToDiskMemory</td>
<td>ECalUserCharacterizer2</td>
<td>Saves the User Characterization to PNA disk memory.</td>
</tr>
<tr>
<td>SaveToECal</td>
<td>ECalUserCharacterizer</td>
<td>Saves the User Characterization to the ECal module.</td>
</tr>
</tbody>
</table>

**Properties**

<table>
<thead>
<tr>
<th>Property</th>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CharacterizationNumber</td>
<td>ECalUserCharacterizer</td>
<td>Sets and reads the number to which the user characterization will be stored in the ECal module.</td>
</tr>
<tr>
<td>ConnectorType</td>
<td>ECalUserCharacterizer</td>
<td>Sets or queries the connector type for the specified port.</td>
</tr>
<tr>
<td>ECalID</td>
<td>ECalUserCharacterizer</td>
<td>Select the model and serial number of the ECal module to be characterized.</td>
</tr>
<tr>
<td>PortDescription</td>
<td>ECalUserCharacterizer</td>
<td>Sets and reads the description of the adapters, cable, or fixture to be included in the user characterization.</td>
</tr>
<tr>
<td>UserDescriptionofPNA</td>
<td>ECalUserCharacterizer</td>
<td>Sets and reads a user description of the PNA used to perform the User Characterization.</td>
</tr>
<tr>
<td>UserName</td>
<td>ECalUserCharacterizer</td>
<td>Sets and reads the description of the person and/or company who is producing the ECal user characterization.</td>
</tr>
<tr>
<td>ValidConnectorTypes</td>
<td>ECalUserCharacterizer</td>
<td>Returns a list of connector names that are valid for use with user-characterized ECal modules.</td>
</tr>
</tbody>
</table>

**IECalUserCharacterizer History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>PNA Rev.</th>
</tr>
</thead>
<tbody>
<tr>
<td>IECalUserCharacterizer</td>
<td>8.33</td>
</tr>
<tr>
<td>IECalUserCharacterizer2</td>
<td>9.00</td>
</tr>
</tbody>
</table>

597
Last Modified:

2-Nov-2008   MX New topic
EmbeddedLO Object

Description
Provides access to the properties that allow measurement of mixers that contain an embedded LO.

Accessing the EmbeddedLO Interface
Access the Interface through the IMixer Object.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
app.Preset

' FCA Measurements can't share the channel with standard measurements
' Because preset creates a single measurement in channel 1, we first delete the
' standard measurement
Dim standardMeas As IMeasurement
Set standardMeas = app.ActiveMeasurement
standardMeas.Delete

' Create a Measurement object, in this case using the IMeasurement interface
Dim meas As IMeasurement
Set meas = app.CreateCustomMeasurementEx(1, "SMC_Forward.SMC_ForwardMeas", "SC21")

' See if this measurement object supports IMixer
Dim mixer As IMixer
Dim embeddedLO
Set embeddedLO = mixer.EmbeddedLO
```

See an example program that shows how to create and calibrate a standard SMC or VMC measurement or a fixed output SMC measurement.

See Also:
PNA Automation Interfaces
The PNA Object Model
Making Embedded LO Measurements
### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ResetLOFrequency</td>
<td>IEmbeddedLO</td>
<td>Reset LO Delta frequency.</td>
</tr>
<tr>
<td>ResetTuningParameters</td>
<td>IEmbeddedLO</td>
<td>Resets the tuning parameters to their defaults.</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BroadbandTuningSpan</td>
<td>IEmbeddedLO</td>
<td>Set broadband sweep span.</td>
</tr>
<tr>
<td>EmbeddedLODiagnostic</td>
<td>IEmbeddedLO</td>
<td>Provides access to the status of tuning sweeps.</td>
</tr>
<tr>
<td>IsOn</td>
<td>IEmbeddedLO</td>
<td>Set and return Embedded LO ON</td>
</tr>
<tr>
<td>LOFrequencyDelta</td>
<td>IEmbeddedLO</td>
<td>Sets and returns LO delta frequency.</td>
</tr>
<tr>
<td>MaxPreciseTuningIterations</td>
<td>IEmbeddedLO</td>
<td>Sets and returns precise tuning iterations.</td>
</tr>
<tr>
<td>NormalizePoint</td>
<td>IEmbeddedLO</td>
<td>Sets and returns tuning point.</td>
</tr>
<tr>
<td>PreciseTuningTolerance</td>
<td>IEmbeddedLO</td>
<td>Sets and returns precise tuning tolerance.</td>
</tr>
<tr>
<td>TuningIFBW</td>
<td>IEmbeddedLO</td>
<td>Sets and returns the IF Bandwidth for tuning sweeps.</td>
</tr>
<tr>
<td>TuningMode</td>
<td>IEmbeddedLO</td>
<td>Sets and returns the method used to determine the embedded LO Frequency.</td>
</tr>
<tr>
<td>TuningSweepInterval</td>
<td>IEmbeddedLO</td>
<td>Set how often a tuning sweep is performed.</td>
</tr>
</tbody>
</table>

### IEmbeddedLO History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IEmbeddedLO</td>
<td>7.21</td>
</tr>
</tbody>
</table>
EmbeddedLODiagnostic Object

Description

Allows access to the properties that provide information about the broadband and precise tuning of an embedded LO.

Accessing the EmbeddedLODiagnostic Interface

Access the Interface through the EmbeddedLO Object.

```vbnet
dim app as agilentPNA835x.application
set app = createobject("agilentPNA835x.application")
app.preset

' FCA Measurements can't share the channel with standard measurements
' Because preset creates a single measurement in channel 1, we first delete the standard measurement
Dim standardMeas as IMeasurement
set standardMeas = app.activeMeasurement
standardMeas.delete

' Create a Measurement object, in this case using the IMeasurement interface
Dim meas as IMeasurement
set meas = app.createCustomMeasurementEx(1, "SMC_Forward.SMC_ForwardMeas", "SC21")

' See if this measurement object supports IMixer
Dim mixer as IMixer
Dim embeddedLO
set embeddedLO = mixer.EmbeddedLO

Dim embeddedLODiagnostic
set embeddedLODiagnostic = embeddedLO.EmbeddedLODiagnostic
```

See an example program that shows how to create and calibrate a standard SMC or VMC measurement or a fixed output SMC measurement.

See Also:

PNA Automation Interfaces
The PNA Object Model
Making Embedded LO Measurements
EmbeddedLO Object
<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clear</td>
<td>IELODiag</td>
<td>Clear current diagnostic information.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IsMarkerOn</td>
<td>IELODiag</td>
</tr>
<tr>
<td>LODeltaFound</td>
<td>IELODiag</td>
</tr>
<tr>
<td>NumberOfSweeps</td>
<td>IELODiag</td>
</tr>
<tr>
<td>MarkerAnnotation</td>
<td>IELODiag</td>
</tr>
<tr>
<td>MarkerPosition</td>
<td>IELODiag</td>
</tr>
<tr>
<td>Parameter</td>
<td>IELODiag</td>
</tr>
<tr>
<td>StatusAsString</td>
<td>IELODiag</td>
</tr>
<tr>
<td>StepData</td>
<td>IELODiag</td>
</tr>
<tr>
<td>StepTitle</td>
<td>IELODiag</td>
</tr>
<tr>
<td>XAxisAnnotation</td>
<td>IELODiag</td>
</tr>
<tr>
<td>XAxisStart</td>
<td>IELODiag</td>
</tr>
<tr>
<td>XAxisStop</td>
<td>IELODiag</td>
</tr>
<tr>
<td>YAxisAnnotation</td>
<td>IELODiag</td>
</tr>
</tbody>
</table>

**History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IEmbeddedLODiagnostic</td>
<td>7.21</td>
</tr>
</tbody>
</table>
ENRFile Object

Description
Provide commands for creating or editing an ENR file. This is rarely necessary as ENR files, which contain factory calibrated data, are typically provided by the manufacturer of the noise source.

Learn more about Noise Figure Application

Accessing the ENRFile object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim enr As ENRFile
Set enr = app.ENRFile

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Program

Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GetENRData</td>
<td>IENRFile</td>
<td>Read the ENR calibration data from PNA memory.</td>
</tr>
<tr>
<td>PutENRData</td>
<td>IENRFile</td>
<td>Write the ENR calibration data to PNA memory.</td>
</tr>
<tr>
<td>LoadENRFile</td>
<td>IENRFile</td>
<td>Recalls an ENR file from disk into PNA Memory.</td>
</tr>
<tr>
<td>SaveENRFile</td>
<td>IENRFile</td>
<td>Saves an ENR file from PNA memory to disk.</td>
</tr>
</tbody>
</table>

Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENRID</td>
<td>IENRFile</td>
<td>Sets and returns ID of ENR table.</td>
</tr>
<tr>
<td>ENRSN</td>
<td>IENRFile</td>
<td>Sets and returns the serial number of the noise source.</td>
</tr>
</tbody>
</table>

IENRFile History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IENRFFile</td>
<td>8.0</td>
</tr>
</tbody>
</table>

Last Modified:

2-Aug-2007    MX New topic
Equation Object

Description
Provide commands for creating an equation.
Learn more about Equation Editor

Accessing the Equation object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim eq As Equation
Set eq = app.Equation

See Also:
- PNA Automation Interfaces
- The PNA Object Model

Methods

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GetLibraryFunctions</td>
<td>IEquation2</td>
<td>Returns the functions in an imported (loaded) DLL.</td>
</tr>
<tr>
<td>ImportLibrary</td>
<td>IEquation2</td>
<td>Imports an Equation Editor DLL.</td>
</tr>
<tr>
<td>IsLibraryImported</td>
<td>IEquation2</td>
<td>Returns whether a DLL has been imported into the PNA.</td>
</tr>
<tr>
<td>RemoveLibrary</td>
<td>IEquation2</td>
<td>Removes an imported an Equation Editor DLL from the PNA.</td>
</tr>
</tbody>
</table>

Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Text</td>
<td>IEquation</td>
<td>Sets the Equation</td>
</tr>
<tr>
<td>State</td>
<td>IEquation</td>
<td>Sets the Equation enabled state</td>
</tr>
<tr>
<td>Valid</td>
<td>IEquation</td>
<td>Returns whether the equation is presently valid.</td>
</tr>
</tbody>
</table>

Example Program using these commands:
Dim na
Dim meas
Set na = CreateObject("AgilentPNA835x.Application")
Set meas = na.ActiveMeasurement
' Define the measurement
meas.Equation.Text = "mysillyequ=sqrt(AR1_1)"
' Check to see if the equation is valid
valid_e = meas.Equation.Valid
MsgBox valid_e
' Turn on the Equation Editor
meas.Equation.State = True

IEquation History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IEquation  6.03</td>
</tr>
</tbody>
</table>

Last Modified:

4-Dec-2007    Added example
ExternalDevice Object

Description
ExternalDevice objects allow you to set properties and methods for each external device.

Accessing the ExternalDevice Object
You can obtain a handle to an External Device by specifying an item in the External Devices collection.

```vba
Dim app
Set app = CreateObject("AgilentPNA835x.Application")
Dim externalDevices
Set externalDevices = app.ExternalDevices
Dim devicecount
devicecount = externalDevices.count
externalDevices.Add "NewPMAR"
Dim newExternalDevice
Set newExternalDevice = externalDevices.Item("NewPMAR")
newExternalDevice.DeviceType = "Power Meter"
newExternalDevice.IOConfiguration = "GPIB0::14::INSTR"
newExternalDevice.IOEnable = true
```

See Also:
Configure an External Device
ExternalSource Object
PowerSensorAsReceiver Object
The PNA Object Model
### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Active</td>
<td>Sets and returns the state of activation for an external device.</td>
</tr>
<tr>
<td>DeviceType</td>
<td>Sets and returns the DeviceType (source, power meter) for the external device.</td>
</tr>
<tr>
<td>Driver</td>
<td>Sets and returns the external device driver (model).</td>
</tr>
<tr>
<td>ExtendedProperties</td>
<td>Provides access to properties that are unique to the external device.</td>
</tr>
<tr>
<td>IOConfiguration</td>
<td>Sets and returns the method of communication and address for the external device.</td>
</tr>
<tr>
<td>IOEnable</td>
<td>Sets and returns whether an external device is available for IO communication with the PNA.</td>
</tr>
<tr>
<td>Name</td>
<td>Sets and returns the name of the External Device.</td>
</tr>
<tr>
<td>TimeOut</td>
<td>Sets and returns the time out value for communication with the external device.</td>
</tr>
</tbody>
</table>

### ExternalDevice History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IExternalDevices</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

31-Jul-2009    MX New topic (9.0)
ExternalDevices Collection

Description
ExternalDevices collection provides access to an ExternalDevice object.

Accessing the ExternalDevices collection
The ExternalDevices collection is a property of the main Application Object. You can obtain a handle to an External Device by specifying an item in the collection.

```vbnet
Dim app
Set app = CreateObject("AgilentPNA835x.Application")
Dim externalDevices
Set externalDevices = app.ExternalDevices
Dim devicecount
devicecount = externalDevices.count
externalDevices.Add "NewPMAR"
Dim newExternalDevice
Set newExternalDevice = externalDevices.Item("NewPMAR")
newExternalDevice.DeviceType = "Power Meter"
newExternalDevice.IOConfiguration = "GPIB0::14::INSTR"
newExternalDevice.IOEnable = true
```

See Also:

Example: Create a PMAR Device and Measurement
 Configure an External Device
 ExternalDevice Object
 The PNA Object Model
## Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds an external device to the system.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to an external device in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes an external device from the system</td>
</tr>
</tbody>
</table>

## Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of devices in the ExternalDevices collection.</td>
</tr>
<tr>
<td>HasItem</td>
<td>Returns a value indicating whether the specified external devices is configured.</td>
</tr>
<tr>
<td>Items</td>
<td>Returns an array of configured devices in the system.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current application object.</td>
</tr>
</tbody>
</table>

### ExternalDevices History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IExternalDevices</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

30-Jul-2009   MX New topic
ExternalSource Object

Description
The ExternalSource object allows you to set unique properties and methods for each external source.

Accessing the ExternalSource Object
You can obtain a handle to an ExternalSource Object through `ExtendedProperties`

```vba
    dim app
    Set app = CreateObject("AgilentPNA835x.Application")
    dim externalDevices
    Set externalDevices = app.ExternalDevices
    dim devicecount
    devicecount = externalDevices.count
    externalDevices.Add "NewPSG"
    dim newExternalDevice
    Set newExternalDevice = externalDevices.Item("NewPSG")
    newExternalDevice.DeviceType = "Source"
    newExternalDevice.IOConfiguration = "GPIB0::14::INSTR"
    dim PSG
    Set PSG = newExternalDevice.ExtendedProperties
    PSG.DwellPerPoint = 5
```

See Also:
- [Configure an External Device](#)
- [The PNA Object Model](#)
<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DwellPerPoint</strong></td>
<td>Sets and returns the dwell time for an external source.</td>
</tr>
<tr>
<td><strong>TriggerMode</strong></td>
<td>Sets and returns the trigger mode (Software / Hardware) for an external source.</td>
</tr>
<tr>
<td><strong>TriggerPort</strong></td>
<td>Sets and returns the PNA port through which an external source is to be triggered.</td>
</tr>
</tbody>
</table>

**ExtendedProperties History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IExternalSource</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

31-Jul-2009  MX New topic
ExternalTestsets Collection

Description
ExternalTestsets collection provides access to a TestsetControl object. Only one external testset can be controlled by the PNA at any time.

Accessing the ExternalTestsets collection
The ExternalTestsets collection is a property of the main Application Object. You can obtain a handle to a testset by specifying an item in the collection.

Visual Basic Example

```vba
Dim pna
Dim testsets As ExternalTestsets
Dim tset1 As TestsetControl
Set pna = CreateObject("AgilentPNA835x.Application")
Set testsets = pna.ExternalTestsets
Set tset1 = testsets(1)
' make COM calls on tset1 object
End Sub
```

See Also:
ExternalTestset Control COM Example
About External TestSet Control
TestsetControl Object
The PNA Object Model

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a testset to the collection and loads a test set configuration file.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to a testset in the collection.</td>
</tr>
<tr>
<td>TestsetCatalog</td>
<td>Returns a list of supported test sets.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of items in a collection of objects.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current naNetworkAnalyzer application.</td>
</tr>
</tbody>
</table>

ExternalTestsets History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IExternalTestsets</td>
<td>6.0</td>
</tr>
<tr>
<td>IExternalTestsets</td>
<td>6.2</td>
</tr>
</tbody>
</table>
**FIFO Object**

**Description**
These properties control the First IN, First OUT (FIFO) buffer settings for the PNA-X and N5264A.
The 4 GB FIFO data buffer is available with Option 118 on the PNA-X and N5264A.

**Accessing the FIFO object**

```vbnet
Dim app as AgilentPNA835x.Application
Dim fifo as IFIFO
Set fifo = app.IFIFO
```

**See Also:**
- [About FIFO](#)
- [FIFO example program](#)
- [PNA Automation Interfaces](#)
- [The PNA Object Model](#)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clear</td>
<td>IFIFO</td>
</tr>
<tr>
<td></td>
<td>Clears the FIFO buffer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data Property</td>
<td>IFIFO</td>
<td>Reads the next specified number of data points from the FIFO buffer.</td>
</tr>
<tr>
<td>DataCount Property</td>
<td>IFIFO</td>
<td>Returns the total number of data points in the FIFO buffer.</td>
</tr>
<tr>
<td>DataInCompactForm</td>
<td>IFIFO</td>
<td>Reads data from the FIFO buffer. Same as Data but in a compact form.</td>
</tr>
<tr>
<td>State</td>
<td>IFIFO</td>
<td>Turns FIFO ON and OFF</td>
</tr>
</tbody>
</table>

**IFIFO History**
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA</th>
<th>Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFIFO</td>
<td></td>
<td>8.35</td>
</tr>
</tbody>
</table>

Last Modified:

6-Oct-2008  MX New topic
Fixturing Object

Description
Contains the properties for Embedding and De-embedding test fixtures.

Accessing the Fixturing object

```vbnet
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim fixt as Fixturing
Set fixt = chan.Fixturing
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- About Fixturing
- Example Programs

### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoPortExtMeasure</td>
<td>IFixturing2 Measures either an OPEN or SHORT standard.</td>
</tr>
<tr>
<td>AutoPortExtReset</td>
<td>IFixturing2 Clears old port extension delay and loss data.</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoPortExtConfig</td>
<td>IFixturing2</td>
<td>Sets the frequency span that is used to calculate Automatic Port Extension.</td>
</tr>
<tr>
<td>AutoPortExtDCOffset</td>
<td>IFixturing2</td>
<td>Specifies whether or not to include DC Offset as part of Automatic port extension.</td>
</tr>
<tr>
<td>AutoPortExtLoss</td>
<td>IFixturing2</td>
<td>Specifies whether or not to include loss correction as part of Automatic Port Extension.</td>
</tr>
<tr>
<td>AutoPortExtSearchStart</td>
<td>IFixturing2</td>
<td>Set the start frequency for custom user span.</td>
</tr>
<tr>
<td>AutoPortExtSearchStop</td>
<td>IFixturing2</td>
<td>Set the stop frequency for custom user span.</td>
</tr>
<tr>
<td><strong>Command</strong></td>
<td><strong>IFixturing2</strong></td>
<td><strong>Description</strong></td>
</tr>
<tr>
<td>--------------------------</td>
<td>-----------------</td>
<td>-----------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>AutoPortExtState</strong></td>
<td>IFixturing2</td>
<td>Enables and disables automatic port extensions on the specified port.</td>
</tr>
<tr>
<td><strong>CmnModeZConvPortImag</strong></td>
<td>IFixturing2</td>
<td>Sets imaginary value for common port impedance conversion.</td>
</tr>
<tr>
<td><strong>CmnModeZConvPortReal</strong></td>
<td>IFixturing2</td>
<td>Sets real value for common port impedance conversion.</td>
</tr>
<tr>
<td><strong>CmnModeZConvPortZ0</strong></td>
<td>IFixturing2</td>
<td>Sets impedance value for common port impedance conversion.</td>
</tr>
<tr>
<td><strong>DiffPortMatch_C</strong></td>
<td>IFixturing2</td>
<td>Sets Capacitance value of the differential matching circuit.</td>
</tr>
<tr>
<td><strong>DiffPortMatch_G</strong></td>
<td>IFixturing2</td>
<td>Sets Conductance value of the differential matching circuit.</td>
</tr>
<tr>
<td><strong>DiffPortMatch_L</strong></td>
<td>IFixturing2</td>
<td>Sets Inductance value of the differential matching circuit.</td>
</tr>
<tr>
<td><strong>DiffPortMatch_R</strong></td>
<td>IFixturing2</td>
<td>Sets Resistance value of the differential matching circuit.</td>
</tr>
<tr>
<td><strong>DiffPortMatchMode</strong></td>
<td>IFixturing2</td>
<td>Sets type of circuit to embed.</td>
</tr>
<tr>
<td><strong>DiffPortMatchUserFilename</strong></td>
<td>IFixturing2</td>
<td>Specifies the 4-port touchstone file for user-defined differential matching circuit.</td>
</tr>
<tr>
<td><strong>DiffPortMatchState</strong></td>
<td>IFixturing2</td>
<td>Turns ON/OFF differential matching circuit function.</td>
</tr>
<tr>
<td><strong>DiffZConvPortImag</strong></td>
<td>IFixturing2</td>
<td>Sets imaginary value for differential port impedance conversion.</td>
</tr>
<tr>
<td><strong>DiffZConvPortReal</strong></td>
<td>IFixturing2</td>
<td>Sets real value for differential port impedance conversion.</td>
</tr>
<tr>
<td><strong>DiffZConvPortZ0</strong></td>
<td>IFixturing2</td>
<td>Sets impedance value for differential port impedance conversion.</td>
</tr>
<tr>
<td><strong>DiffZConvState</strong></td>
<td>IFixturing2</td>
<td>Turns ON/OFF differential port impedance conversion.</td>
</tr>
<tr>
<td><strong>Embed4PortA</strong></td>
<td>IFixturing2</td>
<td>Returns PNA portA connections.</td>
</tr>
<tr>
<td><strong>Embed4PortB</strong></td>
<td>IFixturing2</td>
<td>Returns PNA portB connections.</td>
</tr>
<tr>
<td><strong>Embed4PortC</strong></td>
<td>IFixturing2</td>
<td>Returns PNA portC connections.</td>
</tr>
<tr>
<td>Function</td>
<td>Fixturing Level</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------------</td>
<td>----------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Embed4PortD</td>
<td>IFixturing2</td>
<td>Returns PNA portD connections.</td>
</tr>
<tr>
<td>Embed4PortList</td>
<td>IFixturing2</td>
<td>Specifies all PNA port connections.</td>
</tr>
<tr>
<td>Embed4PortNetworkFilename</td>
<td>IFixturing2</td>
<td>Specifies *.s4p filename.</td>
</tr>
<tr>
<td>Embed4PortNetworkMode</td>
<td>IFixturing2</td>
<td>Specify embed, de-embed, or none.</td>
</tr>
<tr>
<td>Embed4PortState</td>
<td>IFixturing2</td>
<td>Turns ON or OFF 4-port Network Embed/De-embed.</td>
</tr>
<tr>
<td>Embed4PortTopology</td>
<td>IFixturing2</td>
<td>Specifies the PNA / DUT topology.</td>
</tr>
<tr>
<td>EnablePowerCompensation</td>
<td>IFixturing5</td>
<td>Compensates source power for combined loss through all fixturing functions.</td>
</tr>
<tr>
<td>EnableSnPDataExtrapolation</td>
<td>IFixturing6</td>
<td>Turns ON and OFF SNP file extrapolation for both 2-port and 4-port embedding/de-embedding.</td>
</tr>
<tr>
<td>FixturingState</td>
<td>IFixturing</td>
<td>Turns Fixturing ON and OFF on this channel.</td>
</tr>
<tr>
<td>NetworkPortMap</td>
<td>IFixturing6</td>
<td>Set and return the port mapping for a 4-port SNP file to be embedded.</td>
</tr>
<tr>
<td>Port2PdeembedCktModel</td>
<td>IFixturing</td>
<td>Sets and returns the 2 port De-embedding circuit model for the specified port number.</td>
</tr>
<tr>
<td>Port2PdeembedState</td>
<td>IFixturing</td>
<td>Turns 2 port de-embedding ON and OFF on this channel.</td>
</tr>
<tr>
<td>PortArbzImag</td>
<td>IFixturing3</td>
<td>Sets and returns the imaginary impedance value for the specified single-ended port number.</td>
</tr>
<tr>
<td>PortArbzReal</td>
<td>IFixturing3</td>
<td>Sets and returns the real impedance value for the specified single-ended port number.</td>
</tr>
<tr>
<td>PortArbzState</td>
<td>IFixturing</td>
<td>Turns single-ended port impedance ON and OFF on the specified channel.</td>
</tr>
<tr>
<td>PortArbzZ0</td>
<td>IFixturing3</td>
<td>Sets and returns the real and imaginary impedance value for the specified single-ended port number.</td>
</tr>
<tr>
<td>PortCoupleToSystemMedia</td>
<td>IFixturing4</td>
<td>Couples to system Media type</td>
</tr>
<tr>
<td>PortCoupleToSystemVelocity</td>
<td>IFixturing4</td>
<td>Couples to system Velocity Factor</td>
</tr>
<tr>
<td>Function</td>
<td>IFxturing</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------</td>
<td>------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>PortDelay</td>
<td>IFxturing</td>
<td>Sets and returns the Port Delay value for the specified port number.</td>
</tr>
<tr>
<td>PortDistance</td>
<td>IFxturing4</td>
<td>Sets Port Ext in distance</td>
</tr>
<tr>
<td>PortDistanceUnit</td>
<td>IFxturing4</td>
<td>Sets distance units</td>
</tr>
<tr>
<td>PortExtState</td>
<td>IFxturing</td>
<td>Turns Port Extension ON and OFF on this channel.</td>
</tr>
<tr>
<td>PortExtUse1</td>
<td>IFxturing</td>
<td>Sets and returns the USE1 ON/OFF state for the Loss1 and Freq1 values for the specified port number.</td>
</tr>
<tr>
<td>PortExtUse2</td>
<td>IFxturing</td>
<td>Sets and returns the USE2 ON/OFF state for the Loss2 and Freq2 values for the specified port number.</td>
</tr>
<tr>
<td>PortFreq1</td>
<td>IFxturing</td>
<td>Sets and returns the 1st Port Frequency value for the specified port number.</td>
</tr>
<tr>
<td>PortFreq2</td>
<td>IFxturing</td>
<td>Sets and returns the 2nd Port Frequency value for the specified port number.</td>
</tr>
<tr>
<td>PortLoss1</td>
<td>IFxturing</td>
<td>Sets and returns the 1st Port Loss value for the specified port number.</td>
</tr>
<tr>
<td>PortLoss2</td>
<td>IFxturing</td>
<td>Sets and returns the 2nd Port Loss value for the specified port number.</td>
</tr>
<tr>
<td>PortLossDC</td>
<td>IFxturing</td>
<td>Sets and returns the Port Loss at DC value for the specified port number.</td>
</tr>
<tr>
<td>PortMatching_C</td>
<td>IFxturing</td>
<td>Sets and returns the Capacitance, 'C' value for the specified port number.</td>
</tr>
<tr>
<td>PortMatching_G</td>
<td>IFxturing</td>
<td>Sets and returns the Conductance, 'G' value for the specified port number.</td>
</tr>
<tr>
<td>PortMatching_L</td>
<td>IFxturing</td>
<td>Sets and returns the Inductance, 'L' value for the specified port number.</td>
</tr>
<tr>
<td>PortMatching_R</td>
<td>IFxturing</td>
<td>Sets and returns the Resistance, 'R' value for the specified port number.</td>
</tr>
<tr>
<td>PortMatchingCktModel</td>
<td>IFxturing</td>
<td>Sets and returns the Port Matching circuit model for the specified port number.</td>
</tr>
</tbody>
</table>
PortMatchingState IFixturing Turns Port Matching ON and OFF on this channel.

PortMedium IFixturing4 Sets Media per port

PortVelocityFactor IFixturing4 Set Velocity Factor per port

PortWGCutoffFreq IFixturing4 Sets waveguide cutoff frequency per port

Reverse2PortAdapter IFixturing6 Set and read whether or not to reverse ports on a 2-port fixture or adapter to be de-embedded.

strPort2Pdeembed_S2PFile IFixturing Sets and returns the 2 port De-embedding 'S2P' file name for the specified port number.

strPortMatch_S2PFile IFixturing Sets and returns the Port Matching 'S2P' file name for the specified port number.

IFixturing History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFixturing</td>
<td>5.0</td>
</tr>
<tr>
<td>IFixturing2</td>
<td>5.2</td>
</tr>
<tr>
<td>IFixturing3</td>
<td>5.25</td>
</tr>
<tr>
<td>IFixturing4</td>
<td>8.50</td>
</tr>
<tr>
<td>IFixturing5</td>
<td>9.20</td>
</tr>
</tbody>
</table>
FOM Collection

Description
The FOM collection provides access to the source and receiver range objects which are used for configuring frequency offset measurements.

The FOM range items are typically numbered as follows:

1. Primary
2. Source
3. Receivers
4. Source2 (if present)

External devices can appear in the list of range names. Learn more.

Accessing the FOM Collection and FOMRange objects

```vba
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel

Dim ifom as FOM
Set ifom = chan.FOM

ifom.item(2).Coupled = false
```

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- About FOM
### Example Programs

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>See History</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Item</th>
<th>IFOM</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DisplayRange</td>
<td>IFOM</td>
<td>Sets the range to be displayed on the PNA x-axis.</td>
</tr>
<tr>
<td>FOMRange</td>
<td>IFOM</td>
<td>Object</td>
</tr>
<tr>
<td>RangeCount</td>
<td>IFOM</td>
<td>Returns the number of FOM ranges available on the PNA.</td>
</tr>
<tr>
<td>State</td>
<td>IFOM</td>
<td>Turns Frequency Offset ON and OFF.</td>
</tr>
</tbody>
</table>

### FOM History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFOM</td>
<td>7.10</td>
</tr>
</tbody>
</table>

Last Modified:

8-Mar-2007  Modified Access
FOMRange Object

Description
The FOM Range object provides access to the properties and methods for configuring a specific Range for frequency offset measurements.

Accessing an FOMRange object
Get a handle to a FOM Range by specifying an item in the FOM collection.
The FOM range items are typically numbered as follows:

1. Primary
2. Source
3. Receivers
4. Source2 (if present)

External devices can appear in the list of range names. Learn more.

```vba
Dim app as AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim ranges as FOM
Set ranges = app.ActiveChannel.FOM
ranges.item(2).Coupled = False
```

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- About FOM
- Example Programs
## Method

### See History

None

## Property

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coupled</td>
<td>IFOMRange</td>
<td>Sets and returns the state of coupling (ON or OFF) of this range to the primary range.</td>
</tr>
<tr>
<td>CWFrequency</td>
<td>IFOMRange</td>
<td>Set the Continuous Wave (CW) frequency.</td>
</tr>
<tr>
<td>Divisor</td>
<td>IFOMRange</td>
<td>Sets and returns the Divisor value to be used when coupling this range to the primary range.</td>
</tr>
<tr>
<td>Multiplier</td>
<td>IFOMRange</td>
<td>Sets and returns the Multiplier value to be used when coupling this range to the primary range.</td>
</tr>
<tr>
<td>Name</td>
<td>IFOMRange</td>
<td>Returns the name of this FOM range object.</td>
</tr>
<tr>
<td>Offset</td>
<td>IFOMRange</td>
<td>Sets and returns the offset value to be used when coupling this range to the primary range.</td>
</tr>
<tr>
<td>rangeNumber</td>
<td>IFOMRange</td>
<td>Returns the index number of the range within the FOM collection.</td>
</tr>
<tr>
<td>Segments</td>
<td>IFOMRange</td>
<td>Collection - Used to add segment sweep capability to a range.</td>
</tr>
<tr>
<td>StartFrequency</td>
<td>IFOMRange</td>
<td>Sets or returns the start frequency of this FOM Range.</td>
</tr>
<tr>
<td>StopFrequency</td>
<td>IFOMRange</td>
<td>Sets or returns the stop frequency of this FOM Range.</td>
</tr>
<tr>
<td>Sweep Type</td>
<td>IFOMRange</td>
<td>Sets the type of range sweep.</td>
</tr>
</tbody>
</table>

**Note:** Use the [Start Power](#) and [Stop Power](#) settings from the [channel object](#).

## FOM History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFOM</td>
<td>7.10</td>
</tr>
</tbody>
</table>

Last Modified:
<table>
<thead>
<tr>
<th>Date</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>7-Jan-2008</td>
<td>Added Start/Stop power note</td>
</tr>
<tr>
<td>7-Mar-2007</td>
<td>Modified Receivers</td>
</tr>
</tbody>
</table>
Gain Compression Measurement Object

Description
Controls the Gain Compression Analysis settings.

Accessing the GainCompressionMeas object

```vba
Set app = CreateObject("AgilentPNA835x.Application")
app.Preset
app.ActiveMeasurement.Delete
' create a GCA measurement
app.CreateCustomMeasurementEx 1, "Gain Compression","CompIn21", 1
Set meass = app.Measurements ' get the measurements collection
Set ana = meass(1).CustomMeasurementConfiguration 'get the measurement
ana.AnalysisEnable = true ' enable the analysis mode
```

See Also:

- Example Program Create and Cal a Gain Compression Measurement (includes Analysis)
- GainCompression Object
- GainCompressionCal Object
- About Gain Compression Application
- PNA Automation Interfaces
- The PNA Object Model

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>See History</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

None

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AnalysisCWFreq</td>
<td>IGainCompressionMeas</td>
<td>Set CW frequency.</td>
</tr>
<tr>
<td>AnalysisEnable</td>
<td>IGainCompressionMeas</td>
<td>Enable a compression analysis trace.</td>
</tr>
<tr>
<td>AnalysisIsDiscreteFreq</td>
<td>IGainCompressionMeas</td>
<td>Set to discrete or interpolated CW frequencies.</td>
</tr>
</tbody>
</table>
AnalysisXAxis  
IGainCompressionMeas  Sets X-axis display.

## IGainCompressionMeas History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGainCompressionMeas</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

3-Sep-2009  MX New topic
Gain Compression Object

Description
Controls the Gain Compression Application settings.

Accessing the GainCompression object

```vba
Dim app as AgilentPNA835x.Application
app.CreateCustomMeasurementEx(1, "Gain Compression", "CompIn21", 1)
Dim GCA
Set GCA = app.ActiveChannel.CustomChannelConfiguration
```

See Also:

- Example Programs
  - Create and Cal a Gain Compression Measurement
  - Create and Cal a GCX Measurement
- GainCompressionCal Object
- About Gain Compression Application
- PNA Automation Interfaces
- The PNA Object Model

**Note:** Set the Start/Stop Frequency and Start/Stop Power Settings using the Channel Object.

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GetRaw2DData</td>
<td>IGainCompression</td>
<td>Reads Gain Compression data from specified location.</td>
</tr>
<tr>
<td>GetDataIm</td>
<td>IGainCompression</td>
<td>Reads Imaginary part of specified frequency or power points.</td>
</tr>
<tr>
<td>GetDataRe</td>
<td>IGainCompression</td>
<td>Reads REAL part of specified frequency or power points.</td>
</tr>
<tr>
<td>SetPortMap</td>
<td>IGainCompression</td>
<td>Maps the PNA ports to the DUT ports</td>
</tr>
<tr>
<td>Property</td>
<td>Interface</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------------</td>
<td>-----------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>AcquisitionMode</td>
<td>IGainCompression</td>
<td>Set and read the method by which gain compression data is acquired.</td>
</tr>
<tr>
<td>CompressionAlgorithm</td>
<td>IGainCompression</td>
<td>Set and read the algorithm method used to compute gain compression.</td>
</tr>
<tr>
<td>CompressionBackoff</td>
<td>IGainCompression</td>
<td>Set and read value for the BackOff compression algorithm.</td>
</tr>
<tr>
<td>CompressionDeltaX</td>
<td>IGainCompression</td>
<td>Set and read the 'X' value in the delta X/Y compression algorithm.</td>
</tr>
<tr>
<td>CompressionDeltaY</td>
<td>IGainCompression</td>
<td>Set and read the 'Y' value in the delta X/Y compression algorithm.</td>
</tr>
<tr>
<td>CompressionInterpolation</td>
<td>IGainCompression</td>
<td>Sets whether or not to interpolate the final power level when the measured compression level deviates from the specified level.</td>
</tr>
<tr>
<td>CompressionLevel</td>
<td>IGainCompression</td>
<td>Set and read the decrease in gain which indicates that the amplifier is compressing.</td>
</tr>
<tr>
<td>DeviceInputPort</td>
<td>IGainCompression</td>
<td>Read the PNA port number which is connected to the DUT input.</td>
</tr>
<tr>
<td>DeviceOutputPort</td>
<td>IGainCompression</td>
<td>Read the PNA port number which is connected to the DUT output.</td>
</tr>
<tr>
<td>EndOfSweepOperation</td>
<td>IGainCompression</td>
<td>Set and read the action which should be taken at the end of the last frequency or power sweep in the measurement.</td>
</tr>
<tr>
<td>InputLinearPowerLevel</td>
<td>IGainCompression</td>
<td>Set and read the input power level that should produce linear gain.</td>
</tr>
<tr>
<td>MaximumNumberOfPoints</td>
<td>IGainCompression</td>
<td>Returns the maximum possible number of data points.</td>
</tr>
<tr>
<td>NumberOfFrequencyPoints</td>
<td>IGainCompression</td>
<td>Set and read the number of data points in each frequency sweep.</td>
</tr>
<tr>
<td>NumberOfPowerPoints</td>
<td>IGainCompression</td>
<td>Set and read the number of data points in each power sweep.</td>
</tr>
</tbody>
</table>
**ReverseLinearPowerLevel**  
IGainCompression  
Set and read the reverse power level to the DUT.

**SafeSweepCoarsePowerAdjustment**  
IGainCompression  
Set and read the Safe Sweep COARSE power adjustment.

**SafeSweepEnable**  
IGainCompression  
Set and read the (ON | OFF) state of Safe Sweep mode.

**SafeSweepFinePowerAdjustment**  
IGainCompression  
Set and read the Safe Sweep FINE power adjustment.

**SafeSweepFineThreshold**  
IGainCompression  
Set and read the compression level in which Safe Sweep changes from the COARSE power adjustment to the FINE power adjustment.

**SafeSweepMaximumLimit**  
IGainCompression  
When the DUT Output reaches this value, the input power to the DUT is no longer incremented at that frequency.

**SaturationLevel**  
IGainCompression  
Set and read the deviation dB from the maximum Pout.

**SearchFailures**  
IGainCompression  
Read number of points that did not achieve compression.

**SearchSummary**  
IGainCompression  
Returns if a compression search is complete, and if the search was a success or failure.

**SmartSweepMaximumIterations**  
IGainCompression  
Set and read the maximum number of iterations to be used to find the compression level in a SMART sweep.

**SmartSweepSettlingTime**  
IGainCompression  
Set and read SMART sweep settling time.

**SmartSweepShowIterations**  
IGainCompression  
Set and read whether to show results for each SMART sweep iteration.

**SmartSweepTolerance**  
IGainCompression  
Set and read the level of tolerance to be used to find the compression level in a SMART sweep.

**TotalIterations**  
IGainCompression  
Returns the total number of iteration required for the SMART Sweep.

**TotalNumberOfPoints**  
IGainCompression  
Set and read the total number of data points. (Freq x Power)
## IGainCompression History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGainCompression</td>
<td>8.0</td>
</tr>
<tr>
<td>IGainCompression2</td>
<td>8.2</td>
</tr>
<tr>
<td>IGainCompression3</td>
<td>9.0</td>
</tr>
<tr>
<td>IGainCompression4</td>
<td>9.2</td>
</tr>
</tbody>
</table>

**Last Modified:**
- 3-Sep-2009   Added GC3 (9.0)
- 10-Jun-2009  Added SearchSummary
- 12-May-2008  Added 8.2
- 11-Sep-2007  MX New topic
Gain CompressionCal Object

Description
Sets properties that are unique to a Gain Compression Cal (opt 086).
The remaining commands to perform a GCA Cal use the Guided Calibration commands.

Accessing the GainCompressionCal object

```vba
Dim app as AgilentPNA835x.Application
Set GCAcal = pna.GetCalmanager.CreateCustomCalEx(channelNum)
Set GCACalExtension = GCAcal.CustomCalConfiguration
GCACalExtension.PowerLevel = 5
```

See Also:
- Example Program Create and Cal a Gain Compression Measurement
- GainCompression Object
- About Gain Compression Application
- The PNA Object Model
- PNA Automation Interfaces

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>See History</td>
</tr>
</tbody>
</table>

None

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PowerLevel</td>
<td>IGainCompressionCal</td>
<td>Set and read the power level at which to perform the Source Power Cal.</td>
</tr>
<tr>
<td>PowerSensorCalKitType</td>
<td>IGainCompressionCal2</td>
<td>Set and read the cal kit to be used for calibrating at the port 1 reference plane.</td>
</tr>
<tr>
<td>PowerSensorConnectorType</td>
<td>IGainCompressionCal2</td>
<td>Superseded by PowerSensorConnectorType on the GuidedCal Object.</td>
</tr>
</tbody>
</table>

633
### IGainCompressionCal History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGainCompressionCal</td>
<td>8.0</td>
</tr>
<tr>
<td>IGainCompressionCal2</td>
<td>8.04</td>
</tr>
</tbody>
</table>

**Last Modified:**
- **12-Oct-2009** Superseded property (9.1)
- **11-Apr-2008** Added 8.04 properties
- **27-Nov-2007** MX New topic
Gating Object

Description
Contains the methods and properties that control Time Domain Gating.

Accessing the Gating Object

```vbscript
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim gate As Gating
Set gate = app.ActiveMeasurement.Gating
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Time Domain Topics
- Example Programs

Methods
None

Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center</td>
<td>IGating</td>
<td>Sets or returns the Center time.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the Transform Object</td>
</tr>
<tr>
<td>CoupledParameters</td>
<td>IGating2</td>
<td>Select Gating parameters to couple</td>
</tr>
<tr>
<td>Shape</td>
<td>IGating</td>
<td>Specifies the shape of the gate filter.</td>
</tr>
<tr>
<td>Span</td>
<td>IGating</td>
<td>Sets or returns the Span time.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the Transform Object</td>
</tr>
<tr>
<td>Start</td>
<td>IGating</td>
<td>Sets or returns the Start time.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the Transform Object</td>
</tr>
<tr>
<td>State</td>
<td>IGating</td>
<td>Turns an Object ON and OFF.</td>
</tr>
</tbody>
</table>
Stop

IGating
Sets or returns the Stop time.
Shared with the Transform Object

Type

IGating
Specifies the type of gate filter used.

History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGating</td>
<td>1.0</td>
</tr>
<tr>
<td>IGating2</td>
<td>4.2</td>
</tr>
</tbody>
</table>
GlobalPowerLimit Object

Description
Provides access to the properties and methods for setting power limits for PNA ports.

Accessing the GlobalPowerLimit object

```vbnet
Dim app as AgilentPNA835x.Application
Dim gpl as IGlobalPowerLimit
Set gpl = app.GlobalizationLimit
```

See Also:
- About GlobalPowerLimit
- PNA Automation Interfaces
- The PNA Object Model

## Methods

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

## Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Limit</td>
<td>IGlobalPowerLimit</td>
<td>Sets and returns the power limit for the specified port.</td>
</tr>
<tr>
<td>Lock</td>
<td>IGlobalPowerLimit</td>
<td>Enables or disables the ability to change the power limit values through the user interface.</td>
</tr>
<tr>
<td>State</td>
<td>IGlobalPowerLimit</td>
<td>Turns GlobalPowerLimit ON and OFF</td>
</tr>
</tbody>
</table>

IGlobalPowerLimit History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGlobalPowerLimit</td>
<td>9.0</td>
</tr>
</tbody>
</table>
GroupDelayAperture Object

Description
Contains the methods and properties that set Group Delay Aperture.

Accessing the GroupDelayAperture Object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")

Dim GDAperture As GroupDelayAperture
Set GDAperture = app.ActiveMeasurement.GroupDelayAperture
```

See Also:
- Group Delay Measurements
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

### Methods

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>None</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
<td>IGroupDelayAperture</td>
<td>Sets group delay aperture using a fixed frequency range.</td>
</tr>
<tr>
<td>Percent</td>
<td>IGroupDelayAperture</td>
<td>Sets group delay aperture using a percent of the channel frequency span.</td>
</tr>
<tr>
<td>Points</td>
<td>IGroupDelayAperture</td>
<td>Sets group delay aperture using a fixed number of data points.</td>
</tr>
</tbody>
</table>

IGroupDelayAperture History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGroupDelayAperture</td>
<td>A.09.20</td>
</tr>
</tbody>
</table>

Last Modified:

23-Feb-2010    MX New topic
**GuidedCalibration Object**

**Description**
Contains the methods and properties used to perform a Guided Calibration.

**Important!**
Do *NOT* use commands from the `Calibrator` (Unguided calibration) object when performing a Guided calibration. Use ONLY the `GuidedCalibration` object.

The ONLY exception is:
Use `OrientECalModule_Property` and `ECalPortMapEx_Property` on the `Calibrator Object` to specify orientation for both guided and unguided calibrations.

A Guided Calibration must be performed on the Active Channel. To activate a channel, activate any measurement on that channel. Do this using `meas.Activate`, which requires you already have a handle to the measurement.

**Accessing the GuidedCalibration object**

For standard S-parameter channels:

```vba
Dim app as AgilentPNA835x.Application
Dim CalMgr
Set CalMgr = App.GetCalManager
Dim guidedCal
Set guidedCal = CalMgr.GuidedCalibration
```

To calibrate an `Application` channel, see `CreateCustomCalEx` Method.

**See Also:**
- [PNA Automation Interfaces](#)
- [The PNA Object Model](#)
- [Example Programs](#)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>AcquireStep</code></td>
<td>IGuidedCalibration Acquire data for a cal step.</td>
</tr>
<tr>
<td><code>ApplyDeltaMatchFromCalSet Method</code></td>
<td>IGuidedCalibration2 Apply a cal as Delta Match Cal.</td>
</tr>
<tr>
<td><code>GenerateErrorTerms</code></td>
<td>IGuidedCalibration Generates the error terms for the calibration.</td>
</tr>
<tr>
<td><code>GenerateGlobalDeltaMatchSequence</code></td>
<td>IGuidedCalibration2 Initiates a global delta match calibration.</td>
</tr>
</tbody>
</table>
**GenerateSteps**
IGuidedCalibration  Request to generate a connection list and return the number of steps required.

**GetCompatibleCalKits**
IGuidedCalibration5  Returns the list of cal kits for the connector type.

**GetIsolationPaths**
IGuidedCalibration3  Gets the list of port pairings for which isolation standards will be measured during calibration.

**GetStepDescription**
IGuidedCalibration  Query description of a step.

**Initialize**
IGuidedCalibration  Initial setup with channel context for the remote cal object.

**SetIsolationPaths**
IGuidedCalibration3  Sets the list of port pairings for which isolation standards will be measured during calibration.

**SetupMeasurementsForStep**
IGuidedCalibration4  Show the Cal Window, or custom Cal Window, before acquiring a Cal standard.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CalKitType</strong></td>
<td>IGuidedCalibration</td>
<td>Sets the cal kit for the port.</td>
</tr>
<tr>
<td><strong>CompatibleCalKits</strong></td>
<td>IGuidedCalibration</td>
<td>Superseded with <strong>GetCompatibleCalKits Method</strong></td>
</tr>
<tr>
<td><strong>ConnectorType</strong></td>
<td>IGuidedCalibration</td>
<td>Sets the connector type for the port.</td>
</tr>
<tr>
<td><strong>CustomCalConfiguration</strong></td>
<td>IGuidedCalibration4</td>
<td>Provides access to additional Properties and Methods which extends the GuidedCal Object.</td>
</tr>
<tr>
<td><strong>IsolationAveragingIncrement</strong></td>
<td>IGuidedCalibration3</td>
<td>Value by which to increment the channel's averaging factor during measurement of isolation standards.</td>
</tr>
<tr>
<td><strong>PathCalMethod</strong></td>
<td>IGuidedCalibration3</td>
<td>Specifies the calibration method for each port pair.</td>
</tr>
<tr>
<td><strong>PathThruMethod</strong></td>
<td>IGuidedCalibration3</td>
<td>Specifies the calibration <strong>THRU</strong> method for each port pair.</td>
</tr>
<tr>
<td><strong>PerformPowerCalibration</strong></td>
<td>IGuidedCalibration7</td>
<td>Perform Guided Power Cal</td>
</tr>
<tr>
<td><strong>PortsNeedingDeltaMatch</strong></td>
<td>IGuidedCalibration2</td>
<td>Returns port numbers that need delta match cal.</td>
</tr>
</tbody>
</table>
### PowerCalibrationPowerLevel
```
IGuidedCalibration6
```
Sets power level for power cal in several applications.

### PowerSensorCalKitType
```
IGuidedCalibration6
```
Sets Cal Kit for power cal in several applications.

### PowerSensorConnectorType
```
IGuidedCalibration6
```
Sets Power sensor connector for power cal in several applications.

### ThruCalMethod
```
IGuidedCalibration
```
*Superseded with PathCalMethod and PathThruMethod*

### ThruPortList
```
IGuidedCalibration
```
Sets the thru connection port pairs.

### UseCalWindow
```
IGuidedCalibration
```
Turns Cal window ON or OFF

### ValidConnectorTypes
```
IGuidedCalibration
```
Gets Valid Connector Types.

### IGuidedCalibration History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGuidedCalibration</td>
<td>5.0</td>
</tr>
<tr>
<td>IGuidedCalibration2</td>
<td>5.25</td>
</tr>
<tr>
<td>IGuidedCalibration3</td>
<td>7.11</td>
</tr>
<tr>
<td>IGuidedCalibration4</td>
<td>8.0</td>
</tr>
<tr>
<td>IGuidedCalibration5</td>
<td>9.0</td>
</tr>
<tr>
<td>IGuidedCalibration6</td>
<td>9.10</td>
</tr>
<tr>
<td>IGuidedCalibration7</td>
<td>9.30</td>
</tr>
</tbody>
</table>

Last Modified:
5-Jan-2011  Added Calibrator note
8-Oct-2010  Fixed PowerCalLevel
9-Sep-2010  Added IGuidedCalibration7
25-Aug-2009  Added GetCompatibleCalKits
22-Jul-2009  Added CustomCalConfiguration
9-Nov-2007  Added Setup command and Activate note
GuidedCalibrationPowerSensor Object

Description
Contains the methods and properties to configure multiple power sensors to be used during a guided power calibration.

Note: These commands are supported ONLY on standard channels.

Accessing the GuidedCalibrationPowerSensor Object
A GuidedCalibrationPowerSensor object is accessed as an item of the GuidedCalibrationPowerSensors Collection.

```vbscript
Option explicit
Dim app as AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
Dim CalMgr
Set CalMgr = App.GetCalManager
Dim guidedCal
Set guidedCal = CalMgr.GuidedCalibration
Dim powerSensors
Dim port: port = 1
Set powerSensors = guidedCal.GuidedCalibrationPowerSensors(port)
Dim powerSensor2
Set powerSensor2 = GuidedCalibrationPowerSensors.Item(2)
powerSensor2.Name="26GHzSensor"
```

See Also:
- GuidedCalibrationPowerSensors Collection
- See Example Programs
  - Perform a Guided Power Cal using Multiple Power Sensors
  - Create a PMAR Device and Measurement
- PNA Automation Interfaces
- The PNA Object Model
# Methods

None

## Properties

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>Sets and returns the name of this power sensor.</td>
</tr>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>Set and read the cal kit to be used for calibrating at the reference plane.</td>
</tr>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>Sets or returns the connector type of the power sensor</td>
</tr>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>Sets or returns the start frequency of the power sensor coverage</td>
</tr>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>Sets or returns the stop frequency of the power sensor coverage</td>
</tr>
</tbody>
</table>

**IGuidedCalibrationPowerSensors History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IGuidedCalibrationPowerSensors</td>
<td>9.33</td>
</tr>
</tbody>
</table>

**Last Modified:**

- 16-Jun-2011  Updated example and links
- 29-Nov-2010  New topic
GuidedCalibrationPowerSensors Collection

Description
Contains the methods and properties to enable and configure multiple power sensors to be used during a guided calibration.

Note: Guided Power Cal, and "multiple sensors" are allowed ONLY on standard channels.

Accessing the GuidedCalibrationPowerSensors Collection

```vba
Option explicit
Dim app as AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
Dim CalMgr
Set CalMgr = app.GetCalManager
Dim guidedCal
Set guidedCal = CalMgr.GuidedCalibration
Dim port: port = 1
Dim powerSensors
Set powerSensors = guidedCal.GuidedCalibrationPowerSensors(port)
powerSensors = powerSensors.Add "26GHzPowerSensor"
```

See Also:
- GuidedCalibrationPowerSensor Object
- PNA Automation Interfaces
- The PNA Object Model
- See Example Program
<table>
<thead>
<tr>
<th><strong>Methods</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Add</strong></td>
<td><code>IGuidedCalibrationPowerSensors</code> Add a power sensor by name to the collection.</td>
</tr>
<tr>
<td><strong>Item</strong></td>
<td><code>IGuidedCalibrationPowerSensors</code> Specify a power sensor by name in the collection.</td>
</tr>
<tr>
<td><strong>Remove</strong></td>
<td><code>IGuidedCalibrationPowerSensors</code> Remove a power sensor from the collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Properties</strong></th>
<th><strong>Interface</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Count</strong></td>
<td><code>IGuidedCalibrationPowerSensors</code> Return the number of power sensors in the collection.</td>
<td></td>
</tr>
<tr>
<td><strong>UseMultipleSensors</strong></td>
<td><code>IGuidedCalibrationPowerSensors</code> Enable the use of multiple power sensors.</td>
<td></td>
</tr>
</tbody>
</table>

**IGuidedCalibrationPowerSensors History**

<table>
<thead>
<tr>
<th><strong>Interface</strong></th>
<th><strong>Introduced with PNA Rev:</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>IGuidedCalibrationPowerSensors</code></td>
<td>9.33</td>
</tr>
</tbody>
</table>

Last Modified:

- 29-Nov-2010 New topic
HWAuxIO Object

Description
Contains the methods and properties that control the rear panel Auxiliary Input / Output connector.

**Note:** PNA-X models do NOT have this connector. However, the get/put Input/Output voltage commands can be used on the PNA-X to control ADC voltages on the Power I/O connector: Sending other Control:AUX commands to a PNA-X may result in unusual behavior.

Accessing the HWAuxIO object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim AuxIO As HWAuxIO
Set AuxIO = app.GetAuxIO
```

See Also:
- Pinout of the Aux IO Connector
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>get_InputVoltage</td>
<td>IHWAuxIO</td>
<td>Superseded by get InputVoltageEX</td>
</tr>
<tr>
<td>get_OutputVoltage</td>
<td>IHWAuxIO</td>
<td>Reads ADC output voltages.</td>
</tr>
<tr>
<td>get_OutputVoltageMode</td>
<td>IHWAuxIO2</td>
<td>Reads mode setting for either DAC output.</td>
</tr>
<tr>
<td>get_PortCData</td>
<td>IHWAuxIO</td>
<td>Reads a 4-bit value from Port C</td>
</tr>
<tr>
<td>put_OutputVoltage</td>
<td>IHWAuxIO</td>
<td>Writes voltages to the DAC/Analog Output 1 and Output 2</td>
</tr>
<tr>
<td>put_OutputVoltageMode</td>
<td>IHWAuxIO2</td>
<td>Writes the mode setting for either DAC output.</td>
</tr>
<tr>
<td>put_PortCData</td>
<td>IHWAuxIO</td>
<td>Writes a 4-bit value to Port C</td>
</tr>
<tr>
<td><strong>FootSwitch</strong></td>
<td>IHWAuxIO</td>
<td>Reads the Footswitch Input</td>
</tr>
<tr>
<td>---------------</td>
<td>----------</td>
<td>---------------------------</td>
</tr>
<tr>
<td><strong>FootswitchMode</strong></td>
<td>IHWAuxIO3</td>
<td>Determines the action that occurs when the footswitch is pressed.</td>
</tr>
<tr>
<td><strong>InputVoltageEX</strong></td>
<td>IHWAuxIO5</td>
<td>Reads the ADC input voltage</td>
</tr>
<tr>
<td><strong>PassFailLogic</strong></td>
<td>IHWAuxIO</td>
<td>Sets and reads the logic of the PassFail line</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
<tr>
<td><strong>PassFailMode</strong></td>
<td>IHWAuxIO</td>
<td>Sets and reads the mode of the PassFail line</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
<tr>
<td><strong>PassFailPolicy</strong></td>
<td>IHWAuxIO4</td>
<td>Sets the policy used to determine how global pass/fail is computed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
<tr>
<td><strong>PassFailScope</strong></td>
<td>IHWAuxIO</td>
<td>Sets and reads the scope of the PassFail line</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
<tr>
<td><strong>PassFailStatus</strong></td>
<td>IHWAuxIO4</td>
<td>Returns the most recent pass/fail status value.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
<tr>
<td><strong>PortCLogic</strong></td>
<td>HWAuxIO</td>
<td>Sets and reads the logic mode of Port C</td>
</tr>
<tr>
<td><strong>PortCMode</strong></td>
<td>HWAuxIO</td>
<td>Sets and reads the mode of Port C</td>
</tr>
<tr>
<td><strong>SweepEndMode</strong></td>
<td>HWAuxIO</td>
<td>Sets and reads the event that causes the Sweep End line to go to a false state.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Shared with the HWMaterialHandler Object</td>
</tr>
</tbody>
</table>

**IHWAuxIO History**
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IHWAuxIO</td>
<td>2.0</td>
</tr>
<tr>
<td>IHWAuxIO2</td>
<td>3.0</td>
</tr>
<tr>
<td>IHWAuxIO3</td>
<td>3.0</td>
</tr>
<tr>
<td>IHWAuxIO4</td>
<td>5.0</td>
</tr>
<tr>
<td>IHWAuxIO5</td>
<td>7.5</td>
</tr>
</tbody>
</table>

Last Modified:

- 5-Aug-2008   Updated for InputVoltageEX command
- 10-Jul-2007  Added new command
- 29-Jun-2007  Updated for PNA-X ADC commands
HWExternalTestSetIO Object

Description
Contains the methods and properties that control the rear panel External Test Set Input / Output connector

Accessing the HWExternalTestSetIO object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim ExtTS As HWExternalTestSetIO
Set ExtTS = app.GetExternalTestSetIO

See Also:
- Pinout of the Aux IO Connector
- Pinout for the External Test Set Connector
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ReadData</td>
<td>Reads data and generates the appropriate timing signals</td>
</tr>
<tr>
<td>ReadRaw</td>
<td>Reads data, but does NOT generate appropriate timing signals</td>
</tr>
<tr>
<td>WriteData</td>
<td>Writes data and generates the appropriate timing signals</td>
</tr>
<tr>
<td>WriteRaw</td>
<td>Writes data, but does NOT generate the appropriate timing signals</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interrupt</td>
<td>Returns the state of the Interrupt line</td>
</tr>
<tr>
<td>SweepHoldOff</td>
<td>Returns the state of the Sweep Holdoff line</td>
</tr>
</tbody>
</table>

IHWExternalTestSetIO History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>HWExternalTestSetIO</td>
<td>2.0</td>
</tr>
</tbody>
</table>
HWMaterialHandlerIO Object

Description
Contains the methods and properties that control the rear panel Material Handler Input / Output connector.

Accessing the HWMaterialHandlerIO object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim MatHdlr As HWMaterialHandlerIO
Set MatHdlr = app.GetMaterialHandlerIO
```

See Also:
- Pinout for the Material HandlerIO Connector
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>get_Input1</td>
<td>HWMaterialHandlerIO</td>
<td>Reads a hardware latch that captures low to high transition on Input1</td>
</tr>
<tr>
<td>get_Output</td>
<td>HWMaterialHandlerIO</td>
<td>Returns the last value written to the selected output pin.</td>
</tr>
<tr>
<td>get_Port</td>
<td>HWMaterialHandlerIO</td>
<td>Returns the value from the specified &quot;readable&quot; port.</td>
</tr>
<tr>
<td>put_Output</td>
<td>HWMaterialHandlerIO</td>
<td>Writes a TTL HI or TTL Low to output pins 3 or 4.</td>
</tr>
<tr>
<td>put_Port</td>
<td>HWMaterialHandlerIO</td>
<td>Writes a value to the specified port.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IndexState</td>
<td>HWMaterialHandlerIO2</td>
</tr>
<tr>
<td></td>
<td>Determines the control of Material Handler connector Pin 20</td>
</tr>
<tr>
<td>READYForTriggerState</td>
<td>HWMaterialHandlerIO2</td>
</tr>
<tr>
<td></td>
<td>Determines the control of Material Handler connector Pin 21</td>
</tr>
</tbody>
</table>
PassFailLogic | HWMaterialHandlerIO | Sets and reads the logic of the PassFail line
Shared with the HWAuxIO Object

PassFailMode | HWMaterialHandlerIO | Sets and reads the mode for the PassFail line
Shared with the HWAuxIO Object

PassFailPolicy | HWMaterialHandlerIO2 | Sets the policy used to determine how global pass/fail is computed.
Shared with the HWAuxIO Object

PassFailScope | HWMaterialHandlerIO | Sets and reads the scope for the PassFail line
Shared with the HWAuxIO Object

PassFailStatus | HWMaterialHandlerIO2 | Returns the most recent pass/fail status value.
Shared with the HWAuxIO Object

PortLogic | HWMaterialHandlerIO | Sets and returns the logic mode of data ports A-H

PortMode | HWMaterialHandlerIO | Sets and returns whether Port C or Port D is used for writing or reading data

SweepEndMode | HWMaterialHandlerIO | Sets and reads the event that cause the Sweep End line to go to a low state.
Shared with the HWAuxIO Object

**HWMaterialHandlerIO History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>HWMaterialHandlerIO</td>
<td>2.0</td>
</tr>
<tr>
<td>HWMaterialHandlerIO2</td>
<td>5.0</td>
</tr>
</tbody>
</table>
IFConfiguration Object

Description
These properties control the IF gain and source path settings for the following:

- E836x Opt H11 - all IFConfiguration and IFConfiguration2 commands
- PNA-X - IFConfiguration3 commands ONLY

Accessing the IFConfiguration object

```vbscript
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim cfg as IIFConfiguration
Set cfg = chan.IFConfiguration
```

See Also:

- IF Path Configuration
- SignalProcessingModuleFour Object (PNA-X ONLY)
- PulseGenerator Object (PNA-X ONLY)
- The PNA Object Model
- Pulsed Application
- Pulsed Measurement Example

### Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFFilterSampleCount</td>
<td>IFConfiguration2</td>
<td>Sets or returns the number of taps in the IF filter.</td>
</tr>
<tr>
<td>IFFilterSamplePeriod</td>
<td>IFConfiguration2</td>
<td>Sets or returns the IF filter sample period time.</td>
</tr>
<tr>
<td>IFFilterSamplePeriodList</td>
<td>IFConfiguration2</td>
<td>Returns the list of available IF filter sample periods for the instrument.</td>
</tr>
</tbody>
</table>
**IFFilterSamplePeriodMode** IFConfiguration2 Sets or returns the IF filter sample period mode.(Auto or Manual).

**IFFilterSource** IFConfiguration2 Sets or retrieves type of IF filter to be used.

**IFFrequency** IFConfiguration3 Sets IF frequency in manual mode.

**IFFrequencyMode** IFConfiguration3 Sets IF frequency mode to automatic or manual.

**IFGainLevel** IFConfiguration Sets the gain level for the specified receiver.

**IFGainMode** IFConfiguration Sets the gain state for ALL receivers.

**IFGateEnable** IFConfiguration2 Sets or retrieves the state of the IF Gate.

**IFSourcePath** IFConfiguration Sets the source path of the specified receiver to Internal or External.

**MaximumIFFilterSampleCount** IFConfiguration2 Returns the maximum allowed value for the IFFilterSampleCount.

**MinimumIFFilterSampleCount** IFConfiguration2 Returns the minimum allowed value for the IFFilterSampleCount.

**MaximumIFFrequency** IFConfiguration3 Returns the maximum IF frequency setting

**MinimumIFFrequency** IFConfiguration3 Returns the minimum IF frequency setting

### IFConfiguration History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IIFConfiguration</td>
<td>4.0</td>
</tr>
<tr>
<td>IIFConfiguration2</td>
<td>4.0</td>
</tr>
<tr>
<td>IIFConfiguration3</td>
<td>7.2</td>
</tr>
</tbody>
</table>
**IMixer Interface (Option 083) - Superseded**

**Note:** This object and all properties and methods are replaced with Converter Object which can be used for all converter application.

**Description**
Contains the methods and properties to setup FCA Mixer measurements.
For performing calibrations, use either the SMC Type Object or the VMC Type Object.

**Accessing the IMixer Interface**
Access the IMixer Interface through the Measurement Object. If the particular type of Measurement that was created supports IMixer, then the program determines this at run time and can access the functionality exposed by IMixer. Because the determination of IMixer support is not made until runtime, the program should handle the case where IMixer is not supported on the object.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", "analyzerName")
app.Preset

' FCA Measurements can't share the channel with standard measurements
' Because preset creates a single measurement in channel 1, we first delete the standard measurement
Dim standardMeas As IMeasurement
Set standardMeas = app.ActiveMeasurement
standardMeas.Delete

' Create a Measurement object, in this case using the IMeasurement interface
Dim meas As IMeasurement
Set meas = app.CreateCustomMeasurementEx(1, "SMC_Forward.SMC_ForwardMeas", "SC21")

' See if this measurement object supports IMixer
Dim mixer As IMixer
Set mixer = meas
```

**See Also:**
PNA Automation Interfaces
The PNA Object Model
Example: How to create and calibrate a standard SMC or VMC measurement or a fixed output SMC measurement.

**Methods**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Apply</td>
<td>IMixer3</td>
</tr>
<tr>
<td></td>
<td>Applies mixer settings.</td>
</tr>
</tbody>
</table>
Calculate
IMixer
Automatically calculate Input and Output frequencies for mixer setup.

LoadFile
IMixer
Loads a previously-configured mixer attributes file (.mxr)

SaveFile
IMixer
Saves the settings for the mixer/converter test setup to a mixer attributes file.

SetDutPorts
IMixer8
Sets the PNA to DUT port mapping.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActiveXAxisRange</td>
<td>IMixer3 Sets or returns the swept parameter to display on the X-axis.</td>
</tr>
<tr>
<td>AvoidSpurs</td>
<td>IMixer Sets and returns the state of the avoid spurs feature.</td>
</tr>
<tr>
<td>EmbeddedLO</td>
<td>IMixer7 Provides measurements of mixers with an embedded LO.</td>
</tr>
<tr>
<td>DeviceInputPort</td>
<td>IMixer8 Reads the PNA port that is mapped to the DUT Input port</td>
</tr>
<tr>
<td>DeviceOutputPort</td>
<td>IMixer8 Reads the PNA port that is mapped to the DUT Output port</td>
</tr>
<tr>
<td>EnablePhase</td>
<td>IMixer12 Include phase in SMC measurements and calibrations.</td>
</tr>
<tr>
<td>IFSideband</td>
<td>IMixer Sets or returns the value of the IF sideband.</td>
</tr>
<tr>
<td>IFStartFrequency</td>
<td>IMixer Returns the start frequency of the mixer IF.</td>
</tr>
<tr>
<td>IFStopFrequency</td>
<td>IMixer Returns the stop frequency of the mixer IF.</td>
</tr>
<tr>
<td>IncludeReverseSweep</td>
<td>IMixer12 Sets or returns whether to include SC12 sweep.</td>
</tr>
<tr>
<td>InputDenominator</td>
<td>IMixer Sets or returns the denominator value of the Input Fractional Multiplier.</td>
</tr>
<tr>
<td>InputFixedFrequency</td>
<td>IMixer6 Sets or returns the mixer fixed Input frequency value.</td>
</tr>
<tr>
<td>InputNumerator</td>
<td>IMixer Sets or returns the numerator value of the Input Fractional Multiplier.</td>
</tr>
<tr>
<td>InputPower</td>
<td>IMixer Sets or returns the value of the Input Power.</td>
</tr>
<tr>
<td>InputRangeMode</td>
<td>IMixer6 Sets or returns the Input sweep mode.</td>
</tr>
<tr>
<td>InputStartFrequency</td>
<td>IMixer Sets or returns the start frequency of the mixer input.</td>
</tr>
</tbody>
</table>
InputStopFrequency IMixer Sets or returns the stop frequency of the mixer input.

IServiceGreaterThanLO IMixer2 Specifies whether to use the Input frequency that is greater than the LO or less than the LO.

LODenominator IMixer Sets or returns the denominator value of the LO Fractional Multiplier.

LOFixedFrequency IMixer Sets or returns the fixed frequency of the specified LO.

LOName IMixer Sets or returns the LO name.

LONumerator IMixer Sets or returns the numerator value of the LO Fractional Multiplier.

LOPower IMixer Sets or returns the value of the LO Power.

LORangeMode IMixer3 Sets or returns the LO sweep mode to fixed or swept.

LOStage IMixer Returns the number of LOs (1 or 2).

LOStartFrequency IMixer3 Sets or returns the start frequency of the specified LO.

LOStopFrequency IMixer3 Sets or returns the start frequency of the specified LO.

NominalIncidentPowerState IMixer4 Toggles Nominal Incident Power ON and OFF.

NormalizePoint IMixer12 Set or return the data point used to normalize SMC phase measurements.

OutputFixedFrequency IMixer3 Sets or returns the fixed frequency of the mixer output.

OutputRangeMode IMixer6 Sets or returns the Output sweep mode.

OutputSideband IMixer Sets or returns the value of the output sideband.

OutputStartFrequency IMixer Sets or returns the start frequency of the mixer output.

OutputStopFrequency IMixer Sets or returns the stop frequency of the mixer output.

IMixer History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMixer</td>
<td>1.0</td>
</tr>
<tr>
<td>IMixer2</td>
<td>3.5</td>
</tr>
<tr>
<td>IMixer3</td>
<td>4.0</td>
</tr>
<tr>
<td>IMixer4</td>
<td>4.8</td>
</tr>
<tr>
<td>IMixer5</td>
<td>6.04</td>
</tr>
<tr>
<td>IMixer6</td>
<td>6.20</td>
</tr>
<tr>
<td>IMixer7</td>
<td>7.21</td>
</tr>
<tr>
<td>IMixer8</td>
<td>7.5/8.2</td>
</tr>
<tr>
<td>IMixer12</td>
<td>7.5/9.2</td>
</tr>
</tbody>
</table>
**IMSpectrum Object**

**Description**
Controls the IM Spectrum settings.

**Accessing the IMSpectrum object**

```vbnet
Dim app as AgilentPNA835x.Application
app.CreateCustomMeasurementEx(1, "IMSpectrum", "Output", 1)
Dim IMSpec
Set IMSpec = app.ActiveChannel.CustomChannelConfiguration
```

See Also:

- Example Program Create and Cal a IM Spectrum Measurement
- About IM Spectrum Measurements
- SweptIMD Object
- PNA Automation Interfaces
- The PNA Object Model

### Method

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SetPortMap</td>
<td>IIMSpectrum</td>
<td>Sets limited port mapping</td>
</tr>
</tbody>
</table>

### Property

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CoupleTonePower</td>
<td>IIMSpectrum</td>
<td>ON</td>
</tr>
<tr>
<td>DeviceInputPort</td>
<td>IIMSpectrum</td>
<td>Read input port map</td>
</tr>
<tr>
<td>DeviceOutputPort</td>
<td>IIMSpectrum</td>
<td>Read output port map</td>
</tr>
<tr>
<td>DeltaFrequency</td>
<td>IIMSpectrum</td>
<td>Fixed tone spacing value.</td>
</tr>
<tr>
<td>EqualTonePower</td>
<td>IIMSpectrum2</td>
<td>Set Equal Tone Power state.</td>
</tr>
</tbody>
</table>
F1Frequency IIMSpectrum Frequency of the F1 tone.

F2Frequency IIMSpectrum Frequency of the F2 tone.

FrequencyCenter IIMSpectrum Center frequency of the main tones.

ResolutionBW IIMSpectrum Resolution Bandwidth for the measurement.

SpectrumCenterFrequency IIMSpectrum Receiver Center frequency

SpectrumSpanFrequency IIMSpectrum Receiver frequency span.

SpectrumStartFrequency IIMSpectrum Receiver Start frequency.

SpectrumStopFrequency IIMSpectrum Receiver Stop frequency.

TonePowerSetAt IIMSpectrum2 Set power at DUT input or Output

TrackingChannel IIMSpectrum IMD channel number to which the IM Spectrum channel is coupled.

TrackingEnable IIMSpectrum Enables tracking with an IMD channel.

TrackingManualStepEnable IIMSpectrum Step sweep mode for the IM Spectrum channel.

TrackingStepIndex IIMSpectrum IMD data point number at which the IM spectrum measurement occurs.

SweepOrder IIMSpectrum IM product to view when SweepType = NTH is specified.

SweepType IIMSpectrum Type of sweep for an IMSpectrum measurement.

TonePower IIMSpectrum Power level of the Main Tones

IIMSpectrum History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IIMSpectrum</td>
<td>8.35</td>
</tr>
<tr>
<td>IIMSpectrum2</td>
<td>9.40</td>
</tr>
</tbody>
</table>

Last Modified:
5-May-2011  Added tone power settings
11-Aug-2009  Added limited port mapping
19-Aug-2008  MX New topic
**InterfaceControl Object**

**Description**
Contains the methods and properties that support Interface Control.

**Accessing the InterfaceControl object**

```
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim IntControl As InterfaceControl
Set IntControl = app.InterfaceControl
```

**See Also:**
- PNA Automation Interfaces
- The PNA Object Model
- Interface Control Feature
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConfigurationFile</td>
<td>InterfaceControl</td>
<td>Recalls an Interface Control file</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>InterfaceControl</td>
</tr>
<tr>
<td></td>
<td>Turns Interface Control ON and OFF</td>
</tr>
</tbody>
</table>

**InterfaceControl History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>InterfaceControl</td>
<td>5.2</td>
</tr>
</tbody>
</table>
Limit Test Collection

Description
Child of the Measurement Object. A collection that provides a mechanism for iterating through the Measurement’s Limit Segment objects (Limit Lines). The collection has 100 limit lines by default.

Accessing the LimitTest collection
Get a handle to an individual limit segment by specifying an item of the LimitTest collection.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim limSegs As LimitTest
Set limSegs = app.ActiveMeasurement.LimitTest
limSegs.Item(1).BeginStimulus = 1000000000
limSegs.Item(1).EndStimulus = 1000000000
limSegs.Item(1).BeginResponse = 3.5
limSegs.Item(1).EndResponse = 3.5
```

See Also:
- LimitSegment Object
- Collections in the Analyzer
- The PNA Object Model
- Limit Line Testing Example

**Methods**

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GetTestResult</td>
<td>Retrieves the Pass/Fail results of the Limit Test (State).</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle on a limit line in the collection.</td>
</tr>
</tbody>
</table>

**Properties**

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of limit lines used in the measurement.</td>
</tr>
<tr>
<td>LineDisplay</td>
<td>Displays the limit lines on the screen.</td>
</tr>
<tr>
<td>SoundOnFail</td>
<td>Enables a beep on Limit Test fails.</td>
</tr>
<tr>
<td>State</td>
<td>Turns ON and OFF limit testing.</td>
</tr>
</tbody>
</table>

LimitTest History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ILimitTest</td>
<td>1.0</td>
</tr>
</tbody>
</table>
LimitSegment Object

Description
The LimitSegment object is an individual limit line.

Accessing the LimitSegment object
Get a handle to an individual limit line by using the LimitTest collection.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim limSegs As LimitTest
Set limSegs = app.ActiveMeasurement.LimitTest
limSegs(1).BeginResponse = 1000000000#
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BeginResponse</td>
<td>Specifies the Y-axis value that corresponds with Begin Stimulus (X-axis) value.</td>
</tr>
<tr>
<td>BeginStimulus</td>
<td>Specifies the beginning X-axis value of the Limit Line.</td>
</tr>
<tr>
<td>EndResponse</td>
<td>Specifies the Y-axis value that corresponds with End Stimulus (X-axis) value.</td>
</tr>
<tr>
<td>EndStimulus</td>
<td>Specifies the End X-axis value of the Limit Line.</td>
</tr>
<tr>
<td>Type</td>
<td>Specifies the Limit Line type.</td>
</tr>
</tbody>
</table>

LimitSegment History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ILimitSegment</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Marker Object

Description
Contains the methods and properties that control Markers. There are 10 markers available per measurement:

- 1 reference marker
- 9 markers for absolute data or data relative to the reference marker (delta markers).

There are two ways to control markers through COM.

1. The Measurement object has properties that apply to ALL of the markers for that measurement. For example, `meas.MarkerFormat = naLinMag` applies formatting to all markers.

2. Marker object properties override the Measurement object properties. For example, you can then override the format setting for an individual marker by specifying `mark.Format = naLogMag` on the marker object.

Note: SearchFilterBandwidth is available through the measurement object.

Accessing the Marker object
To turn ON a marker, get a handle to the marker through the measurement object. If not already activated, this command will turn ON marker 1

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
app.ActiveMeasurement.marker(1).Format = naLinMag
```

You can also set the marker object to an object variable:

```vba
Dim m1 As Marker
Set m1 = app.ActiveMeasurement.Marker(1)
m1.Format = naMarkerFormat_LinMag
```

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
- All about Markers

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Activate</td>
<td>IMarker</td>
<td>Makes an object the Active Object. Shared with the Marker Object</td>
</tr>
</tbody>
</table>
**SearchCompressionPoint** (IMarker4)
Searches the marker domain for the compression level.

**SearchMax** (IMarker)
Searches the marker domain for the maximum value.

**SearchMin** (IMarker)
Searches the marker domain for the minimum value.

**SearchNextPeak** (IMarker)
Searches the marker's domain for the next largest peak value.

**SearchPeakLeft** (IMarker)
Searches the marker's domain for the next VALID peak to the left of the marker.

**SearchPeakRight** (IMarker)
Searches the marker's domain for the next VALID peak to the right of the marker.

**SearchTarget** (IMarker)
Searches the marker's domain for the target value.

**SearchTargetLeft** (IMarker)
Moving to the left of the marker position, searches the marker's domain for the target value.

**SearchTargetRight** (IMarker)
Moving to the right of the marker position, searches the marker's domain for the target value.

**SetCenter** (IMarker)
Changes the analyzer's center frequency to the X-axis position of the marker.

**SetCW** (IMarker)
Changes the sweep type to CW mode and makes the CW frequency the marker's frequency.

**SetCWFreq** (IMarker3)
Changes the CW frequency to the frequency of the active marker.

**SetElectricalDelay** (IMarker)
Changes the measurement's electrical delay to the marker's delay value.

**SetReferenceLevel** (IMarker)
Changes the measurement's reference level to the marker's Y-axis value.

**SetStart** (IMarker)
Changes the analyzer's start frequency to the X-axis position of the marker.

**SetStop** (IMarker)
Changes the analyzer's stop frequency to the X-axis position of the marker.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bucket Number</strong></td>
<td>IMarker             Marker data point number</td>
</tr>
<tr>
<td><strong>CompressionLevel</strong></td>
<td>IMarker4            Set and read the marker compression level.</td>
</tr>
<tr>
<td><strong>CompressionPin</strong></td>
<td>IMarker4            Reads the input power at the marker compression level.</td>
</tr>
</tbody>
</table>
**CompressionPout**
IMarker4
Reads the output power at the marker compression level.

**DeltaMarker**
IMarker
Makes a marker relative to the reference marker.

**Distance**
IMarker2
Sets or returns distance value for time domain trace.

**Format**
IMarker
Linear, SWR, and so forth

**Interpolated**
IMarker
Turn marker interpolation ON and OFF

**Number**
IMarker
Read the number of the active marker

**PeakExcursion**
IMarker
Sets and reads the peak excursion value for the specified marker.

**PeakThreshold**
IMarker
Sets peak threshold for the specified marker.

**SearchFunction**
IMarker
Emulates the Tracking function in the marker search dialog box.

**Stimulus**
IMarker
Sets and reads the X-Axis value of the marker.

**Target Value**
IMarker
Sets the target value for the marker when doing Target Searches.

**Tracking**
IMarker
The tracking function finds the selected search function every sweep.

**Type**
IMarker
Sets and reads the marker type.

**UserRange**
IMarker
Assigns the marker to the specified User Range.

**UserRangeMax**
IMarker
Sets the stimulus stop value for the specified User Range.

**UserRangeMin**
IMarker
Sets the stimulus start value for the specified User Range.

**Value**
IMarker
Reads the Y-Axis value of the marker.

---

**Marker History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMarker</td>
<td>1.0</td>
</tr>
<tr>
<td>IMarker2</td>
<td>4.2</td>
</tr>
<tr>
<td>IMarker3</td>
<td>8.33</td>
</tr>
<tr>
<td>IMarker4</td>
<td>8.50</td>
</tr>
</tbody>
</table>
Measurement Object

See IArrayTransfer Interface for putting and getting typed data.
See IMixer Interface (used with Option 083)

Description
The Measurement object is probably the most used object in the PNA Object Model. A measurement object represents the chain of data processing algorithms that take raw data from the channel and make it ready for display, which then becomes the scope of the Trace object.

A Measurement object is defined by its parameter (S11, S22, A/R1, B and so forth). The measurement object is associated with a channel which drives the hardware that produces the data that feeds the measurement. The root of a measurement is the raw data. This buffer of complex paired data then flows through a number of processing blocks: error-correction, trace math, phase correction, time domain, gating, formatting. All of these are controlled through the measurement object.

The ACTIVE measurement is the measurement that will be acted upon if you make a setting from the front panel. It is the measurement whose "button" is pressed in the window with the red "active window" frame. If you create a new measurement, that measurement becomes the active measurement.

Therefore, all automation methods with the word "Active" in them refer to the object associated with the Active measurement, whether that object is a Channel, Window, Trace or Limit line.

Learn about the IMeasurement2 Interface for reading stimulus properties.

Accessing the Measurement object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim meas As IMeasurement
Set meas = app.ActiveMeasurement

or

Set meas = app.Measurements(n)

You can access four other objects through the Measurement object: markers, limit test, transform, and gating. For example, because each measurement has its own set of markers, you can set a marker by doing this:

Dim meas as measurement
Set meas = app.ActiveMeasurement
meas.marker(1).Stimulus = 900e6

IMeasurement2 Interface

Some of the properties and methods for the IMeasurement2 Interface return stimulus values that are set using the channel object. The following is the reason these properties and methods are duplicated.

Every measurement carries with it a snapshot of the stimulus properties of the channel that were in effect when the measurement last acquired data. Therefore, it is the measurement that provides the most accurate stimulus description of its data. Any change made to the channel after the measurement was acquired renders the IChannel interface unreliable in terms of describing the measurement.

See Also:
(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Activate</td>
<td>IMeasurement</td>
<td>Makes a measurement the active measurement. Shared with the Marker Object</td>
</tr>
<tr>
<td>ActivateMarker</td>
<td>IMeasurement</td>
<td>Makes a marker the Active Marker.</td>
</tr>
<tr>
<td>ChangeParameter</td>
<td>IMeasurement</td>
<td>Changes the parameter of the measurement.</td>
</tr>
<tr>
<td>DataToDivisor</td>
<td>IMeasurement</td>
<td>Superseded with DoReceiverPowerCal Method</td>
</tr>
<tr>
<td>DataToMemory</td>
<td>IMeasurement</td>
<td>Stores the active measurement into memory.</td>
</tr>
<tr>
<td>Delete</td>
<td>IMeasurement</td>
<td>Deletes the measurement object.</td>
</tr>
<tr>
<td>DeleteAllMarkers</td>
<td>IMeasurement</td>
<td>Deletes all of the markers from the measurement.</td>
</tr>
<tr>
<td>DeleteMarker</td>
<td>IMeasurement</td>
<td>Deletes a marker from the active measurement.</td>
</tr>
<tr>
<td>getData</td>
<td>IMeasurement</td>
<td>Retrieves Complex data from analyzer memory</td>
</tr>
<tr>
<td>getDataByString</td>
<td>IMeasurement</td>
<td>Retrieves variant data from the specified location in your choice of formats.</td>
</tr>
<tr>
<td>GetFilterStatistics</td>
<td>IMeasurement</td>
<td>Returns all four Filter Statistics</td>
</tr>
<tr>
<td>GetReferenceMarker</td>
<td>IMeasurement</td>
<td>Returns a handle to the reference marker.</td>
</tr>
<tr>
<td>GetSnpData</td>
<td>IMeasurement3</td>
<td>Returns SnP data.</td>
</tr>
<tr>
<td>GetSnpDataWithSpecifiedPorts</td>
<td>IMeasurement7</td>
<td>Returns sNp data for the specified ports.</td>
</tr>
<tr>
<td>GetTraceStatistics</td>
<td>IMeasurement</td>
<td>Returns the Trace Statistics.</td>
</tr>
<tr>
<td>GetXAxisValues</td>
<td>IMeasurement2</td>
<td>Returns the stimulus values for the measurement.</td>
</tr>
<tr>
<td>InterpolateMarkers</td>
<td>IMeasurement</td>
<td>Turns All Marker Interpolation ON and OFF for the measurement.</td>
</tr>
<tr>
<td>putDataComplex</td>
<td>IMeasurement</td>
<td>Puts complex data into one of five data buffers.</td>
</tr>
<tr>
<td>putDataScalar</td>
<td>IMeasurement</td>
<td>Puts formatted variant data into the measurement results buffer.</td>
</tr>
<tr>
<td>Method/Property</td>
<td>Interface</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------------</td>
<td>------------</td>
<td>------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SearchFilterBandwidth</td>
<td>IMeasurement</td>
<td>Searches the domain with the current BW target.</td>
</tr>
<tr>
<td>WriteSnpFileWithSpecifiedPorts</td>
<td>IMeasurement7</td>
<td>Write sNp data for specified ports to a file.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActiveMarker</td>
<td>IMeasurement</td>
<td>Returns a handle to the Active Marker object.</td>
</tr>
<tr>
<td>BalancedMeasurement</td>
<td>IMeasurement</td>
<td>Sets the measurement type that is used with balanced topologies.</td>
</tr>
<tr>
<td>BandwidthTarget</td>
<td>IMeasurement</td>
<td>The insertion loss value at which the bandwidth of a filter is measured.</td>
</tr>
<tr>
<td>BandwidthTracking</td>
<td>IMeasurement</td>
<td>Turns Bandwidth Tracking function ON and OFF.</td>
</tr>
<tr>
<td>CalibrationName</td>
<td>IMeasurement2</td>
<td>Returns the name of the cal type.</td>
</tr>
<tr>
<td>CalibrationType</td>
<td>IMeasurement</td>
<td>Superseded with CalibrationTypeID_property</td>
</tr>
<tr>
<td>CalibrationTypeID</td>
<td>IMeasurement2</td>
<td>Sets or returns the cal type for the current measurement.</td>
</tr>
<tr>
<td>Center</td>
<td>IMeasurement2</td>
<td>Returns the stimulus value of the center point for the measurement.</td>
</tr>
<tr>
<td>channelNumber</td>
<td>IMeasurement</td>
<td>Returns the channel number. Shared with the Channel Object</td>
</tr>
<tr>
<td>CustomMeasurementConfiguration</td>
<td>IMeasurement12</td>
<td>Provides access to custom measurement properties and methods.</td>
</tr>
<tr>
<td>Domain</td>
<td>IMeasurement2</td>
<td>Returns the domain (frequency, time, power) for the measurement.</td>
</tr>
<tr>
<td>ElectricalDelay</td>
<td>IMeasurement</td>
<td>Sets electrical delay.</td>
</tr>
<tr>
<td>ElecDelayMedium</td>
<td>IMeasurement2</td>
<td>Sets or returns the characteristic of the electrical delay medium.</td>
</tr>
<tr>
<td>ElecDistanceDelay</td>
<td>IMeasurement11</td>
<td>Sets delay in distance</td>
</tr>
<tr>
<td>ElecDistanceDelayUnit</td>
<td>IMeasurement11</td>
<td>Sets distance units</td>
</tr>
<tr>
<td>Equation</td>
<td>IMeasurement6</td>
<td>Access Equation Editor</td>
</tr>
<tr>
<td>ErrorCorrection</td>
<td>IMeasurement</td>
<td>Set or get the state of error correction for the measurement.</td>
</tr>
<tr>
<td>ErrorCorrectionIndicator</td>
<td>IMeasurement14</td>
<td>Returns the error correction status of the measurement.</td>
</tr>
<tr>
<td>Property</td>
<td>Class</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------</td>
<td>---------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>FilterBW</td>
<td>IMeasurement</td>
<td>Returns the results of the SearchBandwidth method.</td>
</tr>
<tr>
<td>FilterCF</td>
<td>IMeasurement</td>
<td>Returns the Center Frequency result of the SearchBandwidth method.</td>
</tr>
<tr>
<td>FilterLoss</td>
<td>IMeasurement</td>
<td>Returns the Loss value of the SearchBandwidth method.</td>
</tr>
<tr>
<td>FilterQ</td>
<td>IMeasurement</td>
<td>Returns the Q (quality factor) result of the SearchBandwidth method.</td>
</tr>
<tr>
<td>Format</td>
<td>IMeasurement</td>
<td>Sets display format.</td>
</tr>
<tr>
<td>FormatUnit</td>
<td>IMeasurement</td>
<td>Sets units for unratioed measurements.</td>
</tr>
<tr>
<td>Gating</td>
<td>IMeasurement</td>
<td>Controls Time Domain Gating.</td>
</tr>
<tr>
<td>GroupDelayAperture</td>
<td>IMeasurement</td>
<td>Provides access to the Group Delay Aperture settings.</td>
</tr>
<tr>
<td>InterpolateCorrection</td>
<td>IMeasurement</td>
<td>Turns ON and OFF the calculation of new error terms when stimulus values change.</td>
</tr>
<tr>
<td>InterpolateNormalization</td>
<td>IMeasurement</td>
<td>Superseded with DoReceiverPowerCal Method</td>
</tr>
<tr>
<td>IsSparameter</td>
<td>IMeasurement</td>
<td>Returns true if measurement represents an S-Parameter.</td>
</tr>
<tr>
<td>LimitTest</td>
<td>IMeasurement</td>
<td>Collection for iterating through the Limit Segment objects (Limit Lines).</td>
</tr>
<tr>
<td>LimitTestFailed</td>
<td>IMeasurement</td>
<td>Returns the results of limit testing</td>
</tr>
<tr>
<td>LoadPort</td>
<td>IMeasurement</td>
<td>Returns the load port number associated with an S-parameter reflection measurement.</td>
</tr>
<tr>
<td>LogMagnitudeOffset</td>
<td>IMeasurement</td>
<td>Superseded with DoReceiverPowerCal Method</td>
</tr>
<tr>
<td>MagnitudeOffset</td>
<td>IMeasurement</td>
<td>Offsets the magnitude of the entire data trace to a specified value.</td>
</tr>
<tr>
<td>MagnitudeSlopeOffset</td>
<td>IMeasurement</td>
<td>Offsets the magnitude of the data trace to a value that changes linearly with frequency.</td>
</tr>
<tr>
<td>Marker</td>
<td>IMeasurement</td>
<td>Provides access to Marker settings.</td>
</tr>
<tr>
<td>MarkerFormat</td>
<td>IMeasurement</td>
<td>Sets or returns the format of all the markers in the measurement.</td>
</tr>
<tr>
<td>Marker State</td>
<td>IMeasurement</td>
<td>Sets or returns the ON / OFF state of a marker.</td>
</tr>
<tr>
<td>Method</td>
<td>Type</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Mean</td>
<td>IMeasurement</td>
<td>Returns the mean value of the measurement.</td>
</tr>
<tr>
<td>Name</td>
<td>IMeasurement</td>
<td>Sets or returns the name of the measurement.</td>
</tr>
<tr>
<td>NAWindow</td>
<td>IMeasurement</td>
<td>Controls the part of the display that contains the graticule, or what is written on the display.</td>
</tr>
<tr>
<td>Normalization</td>
<td>IMeasurement</td>
<td><strong>Superseded with</strong> DoReceiverPowerCal Method</td>
</tr>
<tr>
<td>Number</td>
<td>IMeasurement</td>
<td>Returns the number of the measurement.</td>
</tr>
<tr>
<td>NumberOfPoints</td>
<td>IMeasurement2</td>
<td>Returns the Number of Points of the measurement.</td>
</tr>
<tr>
<td>Parameter</td>
<td>IMeasurement</td>
<td>Returns the measurement Parameter.</td>
</tr>
<tr>
<td>PeakToPeak</td>
<td>IMeasurement</td>
<td>Returns the Peak to Peak value of the measurement.</td>
</tr>
<tr>
<td>PhaseOffset</td>
<td>IMeasurement</td>
<td>Sets the Phase Offset for the active channel.</td>
</tr>
<tr>
<td>PSaturation</td>
<td>IMeasurement13</td>
<td>Provides access to the Power Saturation marker search object.</td>
</tr>
<tr>
<td>ReceivePort</td>
<td>IMeasurement2</td>
<td>Returns the receiver port of the measurement.</td>
</tr>
<tr>
<td>ReferenceMarkerState</td>
<td>IMeasurement</td>
<td>Turns the reference marker ON or OFF</td>
</tr>
<tr>
<td>ShowStatistics</td>
<td>IMeasurement</td>
<td>Displays and hides the measurement statistics (peak-to-peak, mean, standard deviation) on the screen.</td>
</tr>
<tr>
<td>Smoothing</td>
<td>IMeasurement</td>
<td>Turns ON and OFF data smoothing.</td>
</tr>
<tr>
<td>SmoothingAperture</td>
<td>IMeasurement</td>
<td>Specifies or returns the amount of smoothing as a ratio of the number of data points in the measurement trace.</td>
</tr>
<tr>
<td>SourcePort</td>
<td>IMeasurement2</td>
<td>Returns the source port of the measurement.</td>
</tr>
<tr>
<td>Span</td>
<td>IMeasurement2</td>
<td>Returns the stimulus span (stop - start) for the measurement.</td>
</tr>
<tr>
<td>StandardDeviation</td>
<td>IMeasurement</td>
<td>Returns the standard deviation of the measurement.</td>
</tr>
<tr>
<td>Start</td>
<td>IMeasurement2</td>
<td>Returns the stimulus value of the first point for the measurement.</td>
</tr>
<tr>
<td>StatisticsRange</td>
<td>IMeasurement</td>
<td>Sets the User Range number for calculating measurement statistics.</td>
</tr>
<tr>
<td>Stop</td>
<td>IMeasurement2</td>
<td>Returns the stimulus value of the last point for the measurement.</td>
</tr>
<tr>
<td>Trace</td>
<td>IMeasurement</td>
<td>Controls scale, reference position, and reference line.</td>
</tr>
</tbody>
</table>
TraceMath

IMeasurement

Performs math operations on the measurement object and the trace stored in memory.

TraceMax

IMeasurement10

Maximizes the active trace.

TraceTitle

IMeasurement8

Writes and reads a trace title.

TraceTitleState

IMeasurement8

Turns trace title ON and OFF

Transform

IMeasurement

Controls Time Domain transforms.

UserRangeMax

IMeasurement15

Sets the stimulus stop value for the specified User Range.

UserRangeMin

IMeasurement15

Sets the stimulus start value for the specified User Range.

View

IMeasurement

Sets (or returns) the type of trace displayed on the screen.

WGCutoffFreq

IMeasurement2

Sets or returns the value of the waveguide cut off frequency.

IMeasurement History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMeasurement</td>
<td>1.0</td>
</tr>
<tr>
<td>IMeasurement2</td>
<td>3.0</td>
</tr>
<tr>
<td>IMeasurement3</td>
<td>4.0</td>
</tr>
<tr>
<td>IMeasurement4</td>
<td>4.2</td>
</tr>
<tr>
<td>IMeasurement5</td>
<td>5.0</td>
</tr>
<tr>
<td>IMeasurement7</td>
<td>6.2</td>
</tr>
<tr>
<td>IMeasurement8</td>
<td>7.2</td>
</tr>
<tr>
<td>IMeasurement9</td>
<td>8.35</td>
</tr>
<tr>
<td>IMeasurement10</td>
<td>8.35</td>
</tr>
<tr>
<td>IMeasurement11</td>
<td>8.50</td>
</tr>
<tr>
<td>IMeasurement12</td>
<td>9.00</td>
</tr>
</tbody>
</table>
**IArrayTransfer Interface**

**Description**
Contains methods for putting data in and getting data out of the analyzer using typed data. This interface transfers data more efficiently than the IMeasurement Interface. However, this interface is only usable from VB6, C, & C++.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>getComplex</td>
<td>Retrieves real and imaginary data from the specified buffer.</td>
</tr>
<tr>
<td>getNAComplex</td>
<td>Retrieves typed NAComplex data from the specified buffer.</td>
</tr>
<tr>
<td>getPairedData</td>
<td>Retrieves magnitude and phase data pairs from the specified buffer.</td>
</tr>
<tr>
<td>getScalar</td>
<td>Retrieves scalar data from the specified buffer.</td>
</tr>
<tr>
<td>putComplex</td>
<td>Puts real and imaginary data into the specified buffer.</td>
</tr>
<tr>
<td>putNAComplex</td>
<td>Puts typed NAComplex data into the specified buffer.</td>
</tr>
<tr>
<td>putScalar</td>
<td>Puts scalar data into the measurement result buffer.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

**IArrayTransfer History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IArrayTransfer</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Measurement Collection

Description
A collection object that provides a mechanism for iterating through the Application measurements.

Accessing the Measurements collection

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim measurements As Measurements
Set measurements = app.Measurements
```

See Also:
- Measurement Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a Measurement to the collection.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle on a measurement in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes a measurement from the measurements collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of measurements in the analyzer.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current Application.</td>
</tr>
</tbody>
</table>
NAWindow Object

Description
The NAWindow object controls the part of the display that contains the graticule, or what is written on the display.

Accessing the NaWindow object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim window As NAWindow
Set window = app.NAWindows(1)
window.AutoScale
```

or

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", "analyzerName")

app.NAWindows(1)..AutoScale
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autoscale</td>
<td>Autoscales all measurements in the window. Shared with the Trace Object</td>
</tr>
<tr>
<td>ShowMarkerReadout</td>
<td>Shows and Hides the Marker readout for the active marker in the upper-right corner of the window object.</td>
</tr>
<tr>
<td>ShowTable</td>
<td>Shows or Hides the specified table for the active measurement in the lower part of the window object.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ActiveTrace</td>
<td>Sets a trace to the Active Trace.</td>
</tr>
<tr>
<td>MarkerReadout</td>
<td>Sets and reads the state of the Marker readouts.</td>
</tr>
</tbody>
</table>

MarkerReadoutResponsePlaces
**MarkerReadoutSize**  
Specifies the size of font used when displaying Marker readout in the selected window.

**MarkerReadoutsPerTrace**

**MarkerReadoutStimulusPlaces**

**MarkerReadoutXPosition**

**MarkerReadoutYPosition**

**MarkerSymbol**

<table>
<thead>
<tr>
<th><strong>OneMarkerReadoutPerTrace</strong></th>
<th>Superceded with MarkerReadoutsPerTrace Property</th>
</tr>
</thead>
</table>

**ScaleCouplingMethod**  
Sets and returns the method of scale coupling.

**ScaleCouplingState**  
Enables and disables scale coupling for the specified window.

**Title**  
Writes or reads a custom title for the window.

**TitleState**  
Turns ON and OFF the window title.

**Traces**  
Collection for getting a handle to a trace or iterating through the traces in a window.

**WindowNumber**  
Reads the number of the active window.

**WindowState**  
Maximizes or minimizes a window.

Shared with the Application Object

---

**INaWindow History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>INaWindow</td>
<td>1.0</td>
</tr>
<tr>
<td>INaWindow2</td>
<td>9.0</td>
</tr>
</tbody>
</table>
NAWindows Collection

Description
A collection object that provides a mechanism for iterating through the Application windows.

Accessing the NaWindows collection

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim windows As NAWindows
Set windows = app.NAWindows
```

See Also:
- NAWindow Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a window to the NAWindows collection.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to a window in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes a window from the NAWindows collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of windows on the analyzer.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current Application.</td>
</tr>
</tbody>
</table>
NoiseCal Object

Description
Controls the noise figure calibration settings for amplifiers and converters. These commands supplement the standard calibration commands on the GuidedCalibration Object.

Accessing the NoiseCal object

```vba
Dim app as AgilentPNA835x.Application
Set noisecal = pna.GetCalmanager.CreateCustomCalEx(channelNum)
Set noiseCalExtension = noisecal.CustomCalConfiguration
noiseCalExtension.NoiseSourceCold = 300
```

See Also

- **Examples:**
  - Create and Cal a Noise Figure Measurement
  - Create and Cal an NFX Measurement
- NoiseFigure Object
- About Noise Figure Measurements
- About NFX Measurements
- PNA Automation Interfaces
- The PNA Object Model

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>See History</td>
</tr>
</tbody>
</table>

None

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoOrientTuner</td>
<td>INoiseCal2</td>
<td>Sets the state of auto orientation for a noise tuner during Noise Figure for NFX.</td>
</tr>
</tbody>
</table>

CalMethod | INoiseCal | Sets and returns the method for performing calibration on a noise channel. |
EnableLOPowerCal  INoiseCal2 Enables and disables LO power calibration for NFX.

ENRFile  INoiseCal Sets and returns the name of the ENR file associated with the noise source.

ForceDeEmbedENRAdapter  INoiseCal2 Sets and reads the state of ENR adapter de-embedding.

ForceDeEmbedSensorAdapter  INoiseCal2 Sets and reads the state of noise source adapter de-embedding.

NoiseSourceCalKitType  INoiseCal Sets and reads the Cal Kit type used to perform a cal at the adapter which is used to connect the noise source (if required.)

NoiseSourceCold  INoiseCal Sets and returns the current temperature at the noise source.

NoiseSourceConnectorType  INoiseCal Sets and reads the connector type of the noise source used during the cal.

RcvCharMethod  INoiseCal3 Set and read the method used to characterize the noise receivers.

NoiseConfiguration History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>INoiseCal</td>
<td>8.0</td>
</tr>
<tr>
<td>INoiseCal2</td>
<td>9.10</td>
</tr>
<tr>
<td>INoiseCal3</td>
<td>9.41</td>
</tr>
</tbody>
</table>

Last Modified:

9-Jun-2011  Added INoise3
30-May-2007  MX New topic
NoiseFigure Object

Description
Controls the Noise Figure application settings.

Accessing the NoiseFigure object

```vba
Dim app as AgilentPNA835x.Application
app.CreateCustomMeasurementEx(1, "NoiseFigure", "NF", 1)
Dim NoiseFig
Set NoiseFig = app.ActiveChannel.CustomChannelConfiguration
```

See Also:

- Example programs
  - Create and Cal a NoiseFigure Measurement
  - Create and Cal an NFX Measurement
- About Noise Figure Measurements
- Noise Figure Calibration Object
- app.NoiseSourceState (ON and OFF)
- ENRFile Object
- PNA Automation Interfaces
- The PNA Object Model

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SetPortMap</td>
<td>INoiseFigure6</td>
<td>Maps DUT ports to PNA-X ports (Opt 028)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AmbientTemperature</td>
<td>INoiseFigure</td>
<td>Sets the air temperature at which the measurement is being performed.</td>
</tr>
<tr>
<td>DeviceInputPort</td>
<td>INoiseFigure6</td>
<td>Read DUT input port map</td>
</tr>
</tbody>
</table>
**DeviceOutputPort**  
INoiseFigure6  
Read DUT output port map

**ImpedanceStates**  
INoiseFigure  
Sets the number of impedance states to use during calibrated measurements.

**NoiseAverageFactor**  
INoiseFigure  
Set averaging of noise receiver.

**NoiseAverageState**  
INoiseFigure  
Turn noise averaging ON and OFF.

**NoiseBandwidth**  
INoiseFigure  
Set bandwidth of noise receiver.

**NoiseGain**  
INoiseFigure  
Set gain state of noise receiver.

**NoiseReceiver**  
INoiseFigure5  
Sets and returns the receiver to use for noise measurements.

**NoiseReceiverSweepTime**  
INoiseFigure3  
Returns an estimate of sweep time.

**NoiseTuner**  
INoiseFigure  
Sets and returns the noise tuner identifier,

**NoiseTunerIn**  
INoiseFigure  
Sets and returns the port identifier of the ECal noise tuner Input

**NoiseTunerOut**  
INoiseFigure  
Sets and returns the port identifier of the ECal noise tuner Output

**SourcePullForSParameters**  
INoiseFigure4  
Set and read the use of source pull technique to compute S22 on Noise Figure on Converters.

### NoiseFigure History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>INoiseFigure</td>
<td>8.0</td>
</tr>
<tr>
<td>INoiseFigure3</td>
<td>9.0</td>
</tr>
<tr>
<td>INoiseFigure4</td>
<td>9.1</td>
</tr>
<tr>
<td>INoiseFigure5</td>
<td>9.2</td>
</tr>
<tr>
<td>INoiseFigure6</td>
<td>9.22</td>
</tr>
</tbody>
</table>
PathConfiguration Object

Description
Provides access to the path configuration currently active on the channel object.
To load, store, or delete a configuration, see ConfigurationManager Object.

Accessing the PathConfiguration object in VB

```vbnet
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim chan as Channel
Set chan = app.ActiveChannel

Dim pathConfig As PathConfiguration
Set pathConfig = chan.PathConfiguration
```

Accessing the PathConfiguration object in C#

```csharp
Type pnaType = Type.GetTypeFromProgID("AgilentPNA835x.Application", "PNA-NAME-HERE");
AgilentPNA835x.Application pna = (AgilentPNA835x.Application)Activator.CreateInstance(pnaType);
AgilentPNA835x.Channel chan = (AgilentPNA835x.Channel)pna.ActiveChannel;
AgilentPNA835x.PathConfiguration path = (AgilentPNA835x.PathConfiguration)chan.get_PathConfiguration();
```

Note:
To learn how to make configuration (element) settings, see this Path Configuration Example
Also see this list of configurable elements and settings.

See Also:

- PathConfigurationManager Object
- PathElement Object
- Path Configurator UI
- PNA Automation Interfaces
- The PNA Object Model
## Methods

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CopyFrom</td>
<td>Copy the mechanical switch settings and attenuator settings from the specified channel to the active channel.</td>
</tr>
<tr>
<td>Store</td>
<td>Saves the current configuration to the specified name.</td>
</tr>
</tbody>
</table>

## Properties

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DescriptiveText</td>
<td>Write and read descriptive text associated with the configuration.</td>
</tr>
<tr>
<td>Elements</td>
<td>Collection of Elements that can be configured (switches and so forth). See the list of elements and settings.</td>
</tr>
<tr>
<td>Element</td>
<td>Returns a handle to a IPathElement object.</td>
</tr>
<tr>
<td>Name</td>
<td>Returns the name of the current configuration.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a pointer to the parent COM object (Channel).</td>
</tr>
</tbody>
</table>

## IPathConfiguration History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPathConfiguration</td>
<td>7.2</td>
</tr>
<tr>
<td>IPathConfiguration2</td>
<td>9.4</td>
</tr>
</tbody>
</table>
PathConfigurationManager Object

Description
These commands allow configurations to be stored, loaded, or deleted on the PNA.
To make path configuration settings, see PathConfiguration Object and the PathElement Object

Accessing the PathConfigurationManager object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim pathConfig As PathConfigurationManager
Set pathConfig = app.PathConfigurationManager

Note:
To learn how to make configuration (element) settings, see this Path Configuration Example
Also see this list of configurable elements and settings.

See Also:

- Path Configuration Example
- Path Configurator
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeleteConfiguration</td>
<td>IPathConfigurationManager</td>
<td>Deletes the specified configuration from the PNA.</td>
</tr>
<tr>
<td>LoadConfiguration</td>
<td>IPathConfigurationManager</td>
<td>Loads the named configuration.</td>
</tr>
<tr>
<td>StoreConfiguration</td>
<td>IPathConfigurationManager</td>
<td>Saves the path configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configurations</td>
<td>IPathConfigurationManager</td>
<td>Returns a list of configuration names stored in the PNA.</td>
</tr>
</tbody>
</table>
IPathConfigurationManager

Returns a handle to the Application object.

IPathConfigurationManager History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPathConfigurationManager</td>
<td>7.2</td>
</tr>
</tbody>
</table>
PathElement Object

Description
Provides access to the settings for the PathElement object.

Accessing the PathElement object in VB

```vbnet
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim chan as Channel
Set chan = app.ActiveChannel
Dim pathConfig As PathConfiguration
Set pathConfig = chan.PathConfiguration
Dim element as PathElement
Set element = pathConfig.PathElement("Src1")
```

Accessing the PathElement object in C#

```csharp
Type pnaType = Type.GetTypeFromProgID("AgilentPNA835x.Application", "PNA-NAME-HERE");
AgilentPNA835x.Application pna = (AgilentPNA835x.Application)Activator.CreateInstance(pnaType);
AgilentPNA835x.Channel chan = (AgilentPNA835x.Channel)pna.ActiveChannel;
AgilentPNA835x.PathConfiguration path = (AgilentPNA835x.PathConfiguration)chan.get_PathConfiguration();
path.get_Element("Port1RefMxr").Value = "External";
```

Note:
To learn how to make configuration (element) settings, see this [Path Configuration Example](#).
Also see this list of configurable elements and settings.

See Also:
- Path Configurator
- PathConfigurationManager Object
- PathConfiguration Object
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>See History</td>
</tr>
</tbody>
</table>

None

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>See History</td>
</tr>
</tbody>
</table>

| Name       | IPathElement | Returns the name of the element. |
| Parent     | IPathElement | Returns a pointer to the Parent Object ([PathConfiguration](#)) |
| Value      | IPathElement | Read / Write get the current setting for the element. |
| Values     | IPathElement | Returns all valid settings for the element. |

**IPathElement History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPathElement</td>
<td>7.2</td>
</tr>
</tbody>
</table>
PhaseControl Object

Description
Contains the properties for configuring Phase Sweep (Opt 088) in the PNA.

Accessing the PhaseControl object

```vba
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim phase as PhaseControl
Set phase = chan.PhaseControl
```

See Also:

- About Phase Control
- Set Phase Sweep using SweepType Property
- PNA Automation Interfaces
- The PNA Object Model
- Example Program

Methods

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CouplePhasePortSettings</td>
<td>IPhaseControl</td>
<td>Set and return whether to couple phase control settings.</td>
</tr>
<tr>
<td>FixedPhase</td>
<td>IPhaseControl</td>
<td>Set and return the fixed phase value.</td>
</tr>
<tr>
<td>FixedRatioedPower</td>
<td>IPhaseControl</td>
<td>Set and return the fixed ratioed power value.</td>
</tr>
<tr>
<td>PhaseControlMode</td>
<td>IPhaseControl</td>
<td>Set and return the ON/Off state of phase control.</td>
</tr>
<tr>
<td>PhaseCorrectionData</td>
<td>IPhaseControl</td>
<td>Set and return an array of phase offsets.</td>
</tr>
<tr>
<td>PhaseCorrectionEnabled</td>
<td>IPhaseControl</td>
<td>Set and return whether to use the phase correction offset array.</td>
</tr>
<tr>
<td>PhaseParameterNumber</td>
<td>IPhaseControl</td>
<td>Set and return max number of leveling sweeps</td>
</tr>
<tr>
<td>---------------------</td>
<td>--------------</td>
<td>------------------------------------------</td>
</tr>
<tr>
<td>PhaseParameter</td>
<td>IPhaseControl</td>
<td>Set and return the ratioed receivers (parameter) to use for phase control.</td>
</tr>
<tr>
<td>PhaseParameterModes</td>
<td>IPhaseControl</td>
<td>Returns the available phase control modes for the specified port.</td>
</tr>
<tr>
<td>PhaseReferencePort</td>
<td>IPhaseControl</td>
<td>Sets and returns the reference port for the Phase Control measurement.</td>
</tr>
<tr>
<td>PhaseTolerance</td>
<td>IPhaseControl</td>
<td>Set and return tolerance value for leveling sweeps</td>
</tr>
<tr>
<td>RatioedPowerCorrectionData</td>
<td>IPhaseControl</td>
<td>Set and return ratioed power offset data</td>
</tr>
<tr>
<td>RatioedPowerCorrectionEnabled</td>
<td>IPhaseControl</td>
<td>Set and return whether to use the ratioed power offset array</td>
</tr>
<tr>
<td>StartPhase</td>
<td>IPhaseControl</td>
<td>Set and return the start value of phase sweep.</td>
</tr>
<tr>
<td>StartRatioedPower</td>
<td>IPhaseControl</td>
<td>Set and return the start ratioed power value.</td>
</tr>
<tr>
<td>StopPhase</td>
<td>IPhaseControl</td>
<td>Set and return the stop value of phase sweep.</td>
</tr>
<tr>
<td>StopRatioedPower</td>
<td>IPhaseControl</td>
<td>Set and return the stop ratioed power value.</td>
</tr>
</tbody>
</table>

**IPhaseControl History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPhaseControl</td>
<td>9.33</td>
</tr>
</tbody>
</table>

Last Modified:

3-Dec-2010    MX New topic
PNOP Object

Description
Contains the methods and properties that initiate and return Power Normal Operating Power markers.

Accessing the PNOP Object

```vbs
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")

Dim pnop As PNOP
Set pnop = app.ActiveMeasurement.PNOP
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- PNOP Markers
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SearchPowerNormalOperatingPoint</td>
<td>IPNOP</td>
<td>Initiates a PNOP search</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BackOff</td>
<td>Sets and returns the backoff value.</td>
</tr>
<tr>
<td>BackOffGain</td>
<td>Returns the BackOffGain result.</td>
</tr>
<tr>
<td>BackOffPin</td>
<td>Returns the BackOffPin result.</td>
</tr>
<tr>
<td>BackOffPout</td>
<td>Returns the BackOffPout result.</td>
</tr>
<tr>
<td>Compression</td>
<td>Returns the Compression result.</td>
</tr>
<tr>
<td>CompressionMax</td>
<td>Returns the Compression Max result.</td>
</tr>
<tr>
<td>Gain</td>
<td>Returns the Gain result.</td>
</tr>
<tr>
<td>GainMax</td>
<td>Returns the Gain Max result.</td>
</tr>
</tbody>
</table>
### Pin
- **Description**: Returns the Pin result.

### PinOffset
- **Description**: Sets and returns the PinOffset value.

### PMaxIn
- **Description**: Returns the PMaxIn result.

### PMaxOut
- **Description**: Returns the PMaxOut result.

### POut
- **Description**: Returns the P Out result.

#### IPNOP History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPNOP</td>
<td>A.09.20</td>
</tr>
</tbody>
</table>

---

**Last Modified:**

- **19-Feb-2010**   MX New topic
PortExtension Object  Superseded

ALL methods and properties on the PortExtension Object are Superseded with the Fixturing Object.

Description
Contains the methods and properties that control Port Extensions.

Accessing a PortExtension object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim PortExt As PortExtension
Set PortExt = app.PortExtension
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
- Superseded commands

Methods

None

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input A</td>
<td>Sets the Input A extension value.</td>
</tr>
<tr>
<td>Input B</td>
<td>Sets the Input B extension value.</td>
</tr>
<tr>
<td>Input C</td>
<td>Sets the Input C extension value.</td>
</tr>
<tr>
<td>Port 1</td>
<td>Sets the Port 1 extension value.</td>
</tr>
<tr>
<td>Port 2</td>
<td>Sets the Port 2 extension value.</td>
</tr>
<tr>
<td>Port 3</td>
<td>Sets the Port 3 extension value.</td>
</tr>
<tr>
<td>State</td>
<td>Turns Port Extensions ON and OFF.</td>
</tr>
</tbody>
</table>

IPort Extension History
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPort Extension</td>
<td>1.0</td>
</tr>
</tbody>
</table>
PowerLossSegment Object

Description
Contains the properties describing a segment of the power loss table used in source power calibration.
You can get a handle to one of these segments through the segments.Item Method of the PowerLossSegments collection.

Accessing the PowerLossSegment object
You can get a handle to one of these segments through  PowerLossSegments.Item(n)

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim PwrLossSeg As PowerLossSegment
Set PwrLossSeg = app.SourcePowerCalibrator.PowerLossSegments(1)
```

See Also:
- About Source Power Cal
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Methods
None

Properties

<table>
<thead>
<tr>
<th>Description</th>
<th>Frequency</th>
<th>The frequency (Hz) associated with this segment.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Loss</td>
<td>The loss value (dB) associated with this segment.</td>
</tr>
<tr>
<td>Description</td>
<td>SegmentNumber</td>
<td>Returns the number of this segment</td>
</tr>
</tbody>
</table>

IPowerLossSegment History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerLossSegment</td>
<td>2.0</td>
</tr>
</tbody>
</table>
PowerLossSegments Collection

Description
A collection object that provides a mechanism for iterating through the segments of the power loss table used in source power calibration. The power loss table can contain up to 100 segments.

Accessing the PowerLossSegments collection

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim PwrLossSegs As PowerLossSegments
Set PwrLossSegs = app.SourcePowerCalibrator.PowerLossSegments

See Also:
- PowerLossSegment Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a PowerLossSegment object to the collection.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to a PowerLossSegment object in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes an object from the collection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of objects in the collection.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the Parent object (SourcePowerCalibrator) of this collection.</td>
</tr>
</tbody>
</table>

Last Modified:

2-Jun-2008   Added segment limit
PowerLossSegmentPMAR Object

Description
Contains the properties describing a segment of the power loss table used with a Power Meter as Receiver.

Accessing the PowerLossSegmentPMAR object
You can get a handle to one of these segments through PowerLossSegmentsPMAR.Item(n)

See Also
- Example: Create a PMAR Device and Measurement
- Configure an External Device
- About PMAR
- ExternalDevice Object
- The PNA Object Model

Methods

None

Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
<td>The frequency (Hz) associated with this segment.</td>
</tr>
<tr>
<td>Loss</td>
<td>The loss value (dB) associated with this segment.</td>
</tr>
<tr>
<td>SegmentNumber</td>
<td>Returns the number of this segment</td>
</tr>
</tbody>
</table>

IPowerLossSegment History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerLossSegmentPMAR</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

25-Aug-2009  MX New topic
**PowerLossSegmentsPMAR Collection**

**Description**
A collection object that provides a mechanism for iterating through the segments of the power loss table used with a Power Meter as Receiver. The power loss table can contain up to 100 segments.

**Accessing the PowerLossSegmentsPMAR collection**
- **Example:** Create a PMAR Device and Measurement

**See Also**
- Configure an External Device
- About PMAR
- ExternalDevice Object
- The PNA Object Model

### Methods

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a PowerLossSegmentPMAR object to the collection.</td>
</tr>
<tr>
<td>Item</td>
<td>Use to get a handle to a PowerLossSegmentPMAR object in the collection.</td>
</tr>
<tr>
<td>Remove</td>
<td>Removes a PowerLossSegmentPMAR object from the collection.</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of PowerLossSegmentPMAR objects in the collection.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the Parent object of this collection.</td>
</tr>
</tbody>
</table>

Last Modified:

25-Aug-2009   MX New topic
PowerMeterInterface Object

Description
Contains the properties used to select a power meter and sensor to be used for a source power calibration.

Note: This object replaces the PowerMeterGPIBAddress Property.

Accessing the PowerMeterInterface object
Get a handle to a power meter object using the PowerMeterInterfaces collection.

```vbscript
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim pwrMtrInterfaces As PowerMeterInterfaces
Set pwrMtrInterfaces = app.SourcePowerCalibrator.PowerMeterInterfaces
If pwrMtrInterfaces.Count > 0 Then
    Dim pwrMtrInterface As PowerMeterInterface
    Set pwrMtrInterface = pwrMtrInterfaces(1)
pwrMtrInterface.Path = naUSB
    pwrMtrInterface.Locator = "Agilent Technologies,U2000A,MY12345678"
End If
```

See Also:
- Source Power Calibration
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Methods
None

Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Path</td>
<td>Specifies the interface to use: GPIB, USB, LAN</td>
</tr>
<tr>
<td>Locator</td>
<td>Specifies the location (address) of the power meter/sensor.</td>
</tr>
</tbody>
</table>
### IPowerMeterInterface History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerMeterInterface</td>
<td>7.50</td>
</tr>
</tbody>
</table>

**Last Modified:**

5-Jul-2007   MX New topic
PowerMeterInterfaces Collection

Description
A collection object that provides a mechanism for accessing the PowerMeterInterface objects.
The collection size is limited to one PowerMeterInterface object. By default, that PowerMeterInterface object refers
to GPIB, and to the GPIB address that is currently set for the power meter on that PNA.
The power meter is specified by using the Path property.

Accessing the PowerMeterInterfaces collection
Get a handle to a power meter object using the PowerMeterInterfaces collection.

```vbscript
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim pwrMtrInterfaces As PowerMeterInterfaces
Set pwrMtrInterfaces = app.SourcePowerCalibrator.PowerMeterInterfaces
If pwrMtrInterfaces.Count > 0 Then
    Dim pwrMtrInterface As PowerMeterInterface
    Set pwrMtrInterface = pwrMtrInterfaces(1)
    pwrMtrInterface.Path = naUSB
    pwrMtrInterface.Locator = "Agilent Technologies,U2000A,MY12345678"
End If
```

See Also:
- Source Power Calibration
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Methods
- Item
  Use to get a handle to a PowerMeterInterface object in the collection.

Properties
- Description

  Count
  Returns the number of objects in the collection.

  Parent
  Returns a handle to the SourcePowerCalibrator
## IPowerMeterInterfaces History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerMeterInterfaces</td>
<td>7.50</td>
</tr>
</tbody>
</table>

Last Modified:

- 5-Mar-2009       Added Parent Property
- 9-Jul-2007       MX New topic
PowerSensor Object

Description
Each power sensor connected to the power meter associated with Source Power Calibration will have a PowerSensor object created to represent it. These PowerSensor objects reside in the PowerSensors collection within the SourcePowerCalibrator object. You cannot directly create PowerSensor objects, but can only retrieve existing ones from the PowerSensors collection.

The PowerSensorCalFactorSegment object is also accessed through the PowerSensor object. These are accessed through the CalFactorSegments collection in the PowerSensor object.

Accessing a PowerSensor object

Dim pna As AgilentPNA835x.Application
Set pna = CreateObject("AgilentPNA835x.Application", analyzerName)

Dim powerCalibrator as SourcePowerCalibrator
Dim powerSensor as PowerSensor
Dim calFactorSegment as PowerSensorCalFactorSegment

Set powerCalibrator = pna.SourcePowerCalibrator

' Specify GPIB address of the power meter.
powerCalibrator.PowerMeterGPIBAddress = 13

' Each time the PowerSensors collection is accessed, the power meter is queried to determine which channels have sensors attached. The collection is updated accordingly.
If powerCalibrator.PowerSensors.Count > 0 Then
  ' If channel B of the meter has a sensor attached but channel A does not, then element 1 of the collection is sensor B. Whenever channel A has a sensor, sensor A will be element 1.
  Set powerSensor = powerCalibrator.PowerSensors(1)
  ' Insert one new PowerSensorCalFactorSegment at the beginning of the collection (index 1).
  powerSensor.CalFactorSegments.Add(1)
  ' Assign our variable to refer to that object.
  Set calFactorSegment = powerSensor.CalFactorSegments(1)

  ' Set property values for that object.
  calFactorSegment.Frequency = 300000
  ' frequency in Hz
  calFactorSegment.CalFactor = 98
  ' cal factor in percent
End If

See Also:
PNA Automation Interfaces
The PNA Object Model
Example Programs

**Bold** Methods or Properties provide access to a child object

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CalFactorSegments</strong></td>
<td>Collection for iterating through the segments of a power sensor cal factor table.</td>
</tr>
<tr>
<td><strong>MaximumFrequency</strong></td>
<td>Maximum usable frequency (Hz) specified for this power sensor.</td>
</tr>
<tr>
<td><strong>MinimumFrequency</strong></td>
<td>Minimum usable frequency (Hz) specified for this power sensor.</td>
</tr>
<tr>
<td><strong>PowerMeterChannel</strong></td>
<td>Identifies which power sensor this object corresponds to (or which channel of the power meter the sensor is connected to).</td>
</tr>
<tr>
<td><strong>ReferenceCalFactor</strong></td>
<td>Reference cal factor (%) associated with this power sensor.</td>
</tr>
</tbody>
</table>

**IPowerSensor History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerSensor</td>
<td>2.0</td>
</tr>
</tbody>
</table>
PowerSensors Collection

Description
A collection object that provides a mechanism for iterating through the PowerSensor objects which are connected to the power meter. Each time this collection object is accessed, the power meter is queried to determine how many sensors are connected to it. The collection size and order of objects is then adjusted accordingly before the requested method or property operation is performed. The power meter is specified by using the `PowerMeterGPIBAddress` property of the `SourcePowerCalibrator` object.

Accessing the PowerSensors Collection

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim PwrSensors As PowerSensors
Set PwrSensors = app.SourcePowerCalibrator.PowerSensors
```

See Also:
- **PowerSensor Object**
- **Collections in the Analyzer**
- **The PNA Object Model**
- **Example Programs**

## Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>Item</code></td>
<td>Use to get a handle to a PowerSensor object in the collection.</td>
</tr>
</tbody>
</table>

## Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>Count</code></td>
<td>Returns the number of objects in the collection.</td>
</tr>
<tr>
<td><code>Parent</code></td>
<td>Returns a handle to the Parent object (<code>SourcePowerCalibrator</code>) of this collection.</td>
</tr>
</tbody>
</table>
PowerSensorCalFactorSegment Object

Description
Contains the properties describing a segment of a power sensor cal factor table.

Accessing the PowerSensorCalFactorSegment object
You can get a handle to one of these segments through `CalFactorSegments.Item(n)`

```
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim calFactSeg As CalFactorSegments
Set calFactSeg = app.SourcePowerCalibrator.PowerSensors(1).CalFactorSegments(1)
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

### Methods
None

### Properties | Description
--- | ---
**Frequency** | The frequency (Hz) associated with this segment. Shared with the PowerLossSegment Object
**CalFactor** | The cal factor (%) associated with this segment.
**SegmentNumber** | Returns the number of this segment. Shared with the PowerLossSegment Object

### IPowerSensorCalFactorSegment History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerSensorCalFactorSegment</td>
<td>2.0</td>
</tr>
</tbody>
</table>
PowerSensorCalFactorSegmentPMAR Object

Description
Contains the properties describing a segment of a power sensor cal factor table that is used with Power Meter as Receiver.

Accessing the PowerSensorCalFactorSegment PMAR object
You can get a handle to one of these segments through CalFactorSegmentsPMAR.Item(n).

See Also
- Example: Create a PMAR Device and Measurement
- Configure an External Device
- ExternalDevice Object
- The PNA Object Model

Methods
None

Properties
<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
</tr>
<tr>
<td>CalFactor</td>
</tr>
<tr>
<td>SegmentNumber</td>
</tr>
</tbody>
</table>

IPowerSensorCalFactorSegmentPMAR History

| Interface | Introduced with PNA Rev: |
|----------------|
| IPowerSensorCalFactorSegmentPMAR | 9.0 |

Last Modified:
25-Aug-2009 MX New topic
PowerSensorAsReceiver Object

Description
Provides the settings for configuring a Power Meter to be used as a PNA Receiver (PMAR).

Accessing a PowerSensorAsReceiver object
This object is accessed through ExternalDevice.ExtendedProperties. When an external device is added to the ExternalDevices collection, and the DeviceType property is set to Power Meter, then ExtendedProperties is used to get a handle to this object.

```vba
externalDevices.Add "NewPMAR"
dim newExternalDevice
Set newExternalDevice = externalDevices.Item("NewPMAR")
newExternalDevice.DeviceType = "Power Meter"
dim PMAR
Set PMAR = newExternalDevice.ExtendedProperties
PMAR.ReadingsPerPoint = 10
```

See Also:
- Example: Create a PMAR Device and Measurement
- Configure an External Device
- ExternalDevice Object
- The PNA Object Model

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>None</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalFactorSegmentsPMAR</td>
<td>Provides access to the collection of segments of a power sensor cal factor table.</td>
</tr>
<tr>
<td>LimitFrequency</td>
<td>Enable or disable the use of the power meter min and max frequencies.</td>
</tr>
<tr>
<td>MaximumFrequency</td>
<td>Maximum usable frequency (Hz) specified for this power sensor.</td>
</tr>
<tr>
<td>MinimumFrequency</td>
<td>Minimum usable frequency (Hz) specified for this power sensor.</td>
</tr>
<tr>
<td>PowerLossSegmentsPMAR</td>
<td>Provides access to the collection of segments of the power loss table.</td>
</tr>
</tbody>
</table>
**PowerMeterChannel**
Identifies which power sensor this object corresponds to (or which channel of the power meter the sensor is connected to).

**ReadingsPerPoint**
Allows for settling of the power sensor READINGS.

**ReadingsTolerance**
Allows for settling of the power sensor READINGS.

**ReferenceCalFactor**
Reference cal factor (%) associated with this power sensor.

**SensorIndex**
Sets the power sensor channel (1 or 2) to be used.

**UsePowerLossSegments**
Specifies if subsequent power readings will use of the loss table.

---

**IPowerSensorAsReceiver History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPowerSensorAsReceiver</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Last Modified:

27-Aug-2009   MX New topic
## Preferences Object

### Description
Sets the preferences for the behavior of several properties.

### Accessing the Preferences object

```vbnet
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim pref As Preferences
Set pref = app.Preferences
```

### See Also:
- PNA Preferences
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

**Bold** Methods or Properties provide access to a child object

### Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>RestoreDefaults</strong></td>
<td>IPreferences9</td>
<td>Restores preference settings to their factory defaults.</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AuxTriggerScopeIsGlobal</strong></td>
<td>IPreferences5</td>
<td>Sets the External Trigger OUT behavior to have either Global or Channel scope.</td>
</tr>
<tr>
<td><strong>CitiContents</strong></td>
<td>IPreferences</td>
<td>Specifies the contents of subsequent citifile saves. Superseded with SaveData</td>
</tr>
<tr>
<td><strong>CitiFormat</strong></td>
<td>IPreferences</td>
<td>Specifies the format of subsequent citifile saves. Superseded with SaveData</td>
</tr>
<tr>
<td><strong>DisplayColors</strong></td>
<td>IPreferences10</td>
<td>Provides access to the ComColors Object.</td>
</tr>
<tr>
<td><strong>EnableSourceUnleveledEvents</strong></td>
<td>IPreferences6</td>
<td>Specifies whether or not to report Source Unleveled errors as system events.</td>
</tr>
</tbody>
</table>
**ExternalDeviceDeActivatePolicy**  
IPreferences10  
External Devices remain activated or are deactivated when the PNA is Preset or when an Instrument State is recalled.

**FrequencyOffsetRangeForCalComputations**  
IPreferences10  
Specifies the FOM frequency range to use when performing calibration.

**OffsetReceiverAttenuator**  
IPreferences6  
Mathematically offset the test port receiver.

**OffsetSourceAttenuator**  
IPreferences6  
Mathematically offset the reference receiver.

**Port1NoiseTunerSwitchPresetsToExternal**  
IPreferences8  
Sets default setting for Noise Figure switch.

**PowerOnDuringRetraceMode**  
IPreferences4  
Specify whether to turn RF power ON or OFF during a retrace for single-band frequency or segment sweeps ONLY.

**PowerSweepRetracePowerMode**  
IPreferences3  
At the end of a power sweep, specifies whether to maintain source power at the start or stop power level.

**PreferInternalTriggerOnChannelSingle**  
IPreferences2  
Sets the preference for chan.Single behavior.

**PreferInternalTriggerOnUnguidedCal**  
IPreferences2  
Set the preference for the trigger behavior when performing an Unguided calibration.

**PresetPowerState**  
IPreferences11  
Set and return the Preset Power ON/OFF state.

**PrintColors**  
IPreferences10  
Provides access to the **ComColors** Object.

**RemoteCalStoragePreference**  
IPreferences7  
Specifies the default manner in which calibrations performed via SCPI or COM are to be stored.

**ReportReceiverOverload**  
IPreferences12  
Set whether to display receiver overload warnings.

**RFOffOnReceiverOverload**  
IPreferences12  
Set whether to turn source power OFF when a receiver is overloaded.

**SnPFormat**  
IPreferences  
Specifies the format of subsequent .S1P, .S2P, .S3P file saves.

**TwoPointGroupDelayAperture**  
IPreferences11  
Sets Group Delay Aperture default to 2 points.

**IPreferences History**
<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPrefs</td>
<td>4.0</td>
</tr>
<tr>
<td>IPrefs2</td>
<td>6.0</td>
</tr>
<tr>
<td>IPrefs3</td>
<td>7.2</td>
</tr>
<tr>
<td>IPrefs4</td>
<td>6.04</td>
</tr>
<tr>
<td>IPrefs5</td>
<td>7.10</td>
</tr>
<tr>
<td>IPrefs6</td>
<td>7.20</td>
</tr>
<tr>
<td>IPrefs7</td>
<td>7.21</td>
</tr>
<tr>
<td>IPrefs8</td>
<td>8.0</td>
</tr>
<tr>
<td>IPrefs9</td>
<td>8.2</td>
</tr>
<tr>
<td>IPrefs10</td>
<td>9.0</td>
</tr>
<tr>
<td>IPrefs11</td>
<td>9.20</td>
</tr>
<tr>
<td>IPrefs12</td>
<td>9.30</td>
</tr>
</tbody>
</table>
PSaturation Object

Description
Contains the methods and properties that initiate a Power Saturation marker search and returns PSAT data.

Accessing the PSaturation Object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
Dim PSat As PSaturation
Set PSat = app.ActiveMeasurement.PSaturation

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- PSaturation Markers
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SearchPowerSaturation</td>
<td>IPSaturation</td>
<td>Initiates a Power Saturation marker search.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CompressionMax</td>
<td>IPSaturation Returns the Compression Max result of a PSat marker search.</td>
</tr>
<tr>
<td>CompressionSaturation</td>
<td>IPSaturation Returns the Compression Saturation result of a PSat marker search.</td>
</tr>
<tr>
<td>GainLinear</td>
<td>IPSaturation Returns the Linear Gain result of a PSat marker search.</td>
</tr>
<tr>
<td>GainMax</td>
<td>IPSaturation Returns the GainMax result of a PSAT marker search.</td>
</tr>
<tr>
<td>GainSaturation</td>
<td>IPSaturation Returns the GainSaturation result of a PSAT marker search.</td>
</tr>
<tr>
<td>Pin</td>
<td>IPSaturation Returns the Pin result of a PSAT marker search.</td>
</tr>
<tr>
<td>PMaxBackOff</td>
<td>IPSaturation Sets and returns the backoff value used to calculate various PSAT parameters.</td>
</tr>
<tr>
<td>PMaxIn</td>
<td>IPSaturation Returns the PMaxIn result of a PSAT marker search.</td>
</tr>
</tbody>
</table>
**PMaxOut**

Returns the PMaxOut result of a PSAT marker search.

**POut Property**

Returns the POut result of a PSAT marker search.

## IPSaturation History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPSaturation</td>
<td>A.09.20</td>
</tr>
</tbody>
</table>
PulseGenerator Object

Description
Contains the properties for configuring the five internal pulse generators in the PNA-X.

Learn more about the PNA-X Pulse Generators.

Accessing the PulseGenerator object

Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim pulse as PulseGenerator
Set pulse = chan.PulseGenerator

Each pulse generator is specified in the Pulse Generator properties. See below.

Pulse Definitions

- D = Delay; the time before each pulse begins
- W = Width; the time the pulse is ON
- P = Period; one complete pulse cycle
- W/P = Duty Cycle; the ratio of pulse ON/OFF

Important: If D + W is greater than P, then undefined PNA behavior results. There is NO error message or warning.

See Also:

- IF Path Block Diagram.
- PNA Automation Interfaces
- The PNA Object Model
- About PNA-X Pulse Measurements
- Example Programs
### Methods

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Delay</strong></td>
<td>IPulsedGenerator</td>
<td>Sets the pulse delay.</td>
</tr>
<tr>
<td><strong>DelayIncrement</strong></td>
<td>IPulsedGenerator</td>
<td>Sets the pulse delay increment.</td>
</tr>
<tr>
<td><strong>Invert</strong></td>
<td>IPulsedGenerator3</td>
<td>Sets whether to invert the polarity of the pulse.</td>
</tr>
<tr>
<td><strong>Period</strong></td>
<td>IPulsedGenerator</td>
<td>Sets the pulse-period (1/PRF) for ALL PNA-X internal pulse generators.</td>
</tr>
<tr>
<td><strong>State</strong></td>
<td>IPulsedGenerator</td>
<td>Turns the specified pulse generator ON and OFF.</td>
</tr>
<tr>
<td><strong>SubPointTrigger</strong></td>
<td>IPulsedGenerator2</td>
<td>Enables / Disables subpoint triggering.</td>
</tr>
<tr>
<td><strong>TriggerInPolarity</strong></td>
<td>IPulsedGenerator3</td>
<td>Sets the polarity of trigger to which the internal pulse generators will respond when being externally triggered.</td>
</tr>
<tr>
<td><strong>TriggerInType</strong></td>
<td>IPulsedGenerator3</td>
<td>Sets the type of trigger to which the internal pulse generators will respond when being externally triggered.</td>
</tr>
<tr>
<td><strong>Width</strong></td>
<td>IPulsedGenerator</td>
<td>Sets the pulse width for the specified pulse generator.</td>
</tr>
</tbody>
</table>

### IPulseGenerator History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPulseGenerator</td>
<td>7.2</td>
</tr>
<tr>
<td>IPulseGenerator2</td>
<td>8.55.09</td>
</tr>
<tr>
<td>IPulseGenerator3</td>
<td>9.10</td>
</tr>
</tbody>
</table>

Last Modified:
8-Oct-2010    Added Invert
9-Dec-2009    Added 9.10 commands
20-Jul-2009   Added SubPointTrig
1-Jan-2007    MX New topic
PulseMeasurementControl Object

Description
Contains the properties for configuring pulse measurements in the PNA-X. Some of these settings require Opt H08 / 008.
Learn more about Pulse Measurements

Accessing the PulseMeasurementControl object

Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim pulse as PulseMeasurementControl
Set pulse = chan.PulseMeasurementControl

See Also:
- IF Path Block Diagram.
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Methods

Description
None

Properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoCWSweepTime</td>
<td>IPulseMeasurementControl2</td>
<td>Sets the state of automatic CW sweep time in Pulse Profile mode.</td>
</tr>
<tr>
<td>AutoDetection</td>
<td>IPulseMeasurementControl</td>
<td>Automatically or manually set pulse mode (Narrowband or Wideband)</td>
</tr>
<tr>
<td>AutoIFBandWidth</td>
<td>IPulseMeasurementControl</td>
<td>Autoselect the IFBW</td>
</tr>
<tr>
<td>AutoIFGain</td>
<td>IPulseMeasurementControl</td>
<td>For future use.</td>
</tr>
<tr>
<td>AutoOptimizePRF</td>
<td>IPulseMeasurementControl</td>
<td>Auto-optimize pulse clock period</td>
</tr>
<tr>
<td><strong>AutoPulseTiming</strong></td>
<td>IPulseMeasurementControl</td>
<td>Autoselect Width and Delay</td>
</tr>
<tr>
<td>---------------------</td>
<td>--------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td><strong>AutoSelectPulseGen</strong></td>
<td>IPulseMeasurementControl</td>
<td>Autoselect Pulse Generators</td>
</tr>
<tr>
<td><strong>MasterFrequency</strong></td>
<td>IPulseMeasurementControl2</td>
<td>Sets the pulse repetition frequency (PRF) for ALL internal pulse generators.</td>
</tr>
<tr>
<td><strong>MasterPeriod</strong></td>
<td>IPulseMeasurementControl2</td>
<td>Sets the period for ALL internal pulse generators.</td>
</tr>
<tr>
<td><strong>MasterWidth</strong></td>
<td>IPulseMeasurementControl2</td>
<td>Sets the pulse width for ALL internal pulse generators.</td>
</tr>
<tr>
<td><strong>Parent</strong></td>
<td>IPulseMeasurementControl</td>
<td>Returns the channel object</td>
</tr>
<tr>
<td><strong>PulseMeasMode</strong></td>
<td>IPulseMeasurementControl</td>
<td>Select Pulse Measurement selection</td>
</tr>
<tr>
<td><strong>SoftwareGateState</strong></td>
<td>IPulseMeasurementControl2</td>
<td>This setting is used for troubleshooting purposes.</td>
</tr>
<tr>
<td><strong>WideBandDectionState</strong></td>
<td>IPulseMeasurementControl</td>
<td>Select Narrowband or Wideband pulse detection.</td>
</tr>
</tbody>
</table>

**IPulseMeasurementControl History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPulseMeasurementControl</td>
<td>9.2</td>
</tr>
<tr>
<td>PulseMeasurementControl2</td>
<td>9.40</td>
</tr>
</tbody>
</table>

Last Modified:

- 3-May-2011 Added master commands
- 11-Mar-2010 MX New topic
RxLevelingConfiguration Object

Description
Contains the properties for configuring Receiver Leveling in the PNA.

Accessing the RxLevelingConfiguration object

```vba
Dim app as AgilentPNA835x.Application  
Dim chan as Channel  
Set chan = app.ActiveChannel  
Dim RxLevel as RxLevelingConfiguration  
Set RxLevel = chan.RxLevelingConfiguration
```

See Also:
- About Receiver Leveling
- PNA Automation Interfaces
- The PNA Object Model
- Example Program

### Methods

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FastMode</td>
<td>IRxLevelingConfiguration  Select separate IFBW for leveling sweeps</td>
</tr>
<tr>
<td>IterationNumber</td>
<td>IRxLevelingConfiguration  Max number of leveling sweeps</td>
</tr>
<tr>
<td>LastLevelingAsSPC</td>
<td>IRxLevelingConfiguration2  Turn ON Source Power Cal using latest correction data</td>
</tr>
<tr>
<td>LevelingIFBW</td>
<td>IRxLevelingConfiguration  IFBW for leveling sweeps</td>
</tr>
<tr>
<td>PowerMax</td>
<td>IRxLevelingConfiguration  Max power for safe mode</td>
</tr>
<tr>
<td>PowerMin</td>
<td>IRxLevelingConfiguration  Min power for safe mode</td>
</tr>
</tbody>
</table>
**PowerOffset** IRxLevelingConfiguration Offset power for external components

**PowerStep** IRxLevelingConfiguration Power step for safe mode

**ReferenceReceiver** IRxLevelingConfiguration Select a receiver

**SafeMode** IRxLevelingConfiguration Enable safe mode

**State** IRxLevelingConfiguration Enable receiver leveling

**Tolerance** IRxLevelingConfiguration Tolerance value for leveling sweeps

**ReceiverLevelingConfiguration History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IRxLevelingConfiguration</td>
<td>8.5</td>
</tr>
<tr>
<td>IRxLevelingConfiguration2</td>
<td>9.30</td>
</tr>
</tbody>
</table>

Last Modified:

- 16-Sep-2010 Added IRxLevelingConfiguration2
- 13-Feb-2009 MX New topic
**SCPIStringParser Object**

**Description**
Provides the ability to send a SCPI command from within the COM command. The two commands differ in the following ways:

**Execute** - will not return an error unless the Execute command itself fails, which is unlikely. Otherwise, you are required to read the SCPI error queue for errors that were caused by the SCPI command. The Execute command operates with minimal interference between you, the programmer, and the SCPI parser. It does not presume how you want to handle errors: handle by ignore, handle by reading the status byte, etc. This command was defined because automation engines like VB throw runtime errors when a COM method returns a failed HRESULT.

**Parse** - parses the input command, and then reads the SCPI error queue until the queue is empty. If the queue contains errors, Parse returns a failed HRESULT (E_NA_BAD_SCPI_EXECUTE). It then creates an IErrorInfo object and bundles the error numbers and descriptions into the error object. This object is available so that you can detect the failed HRESULT and interrogate the errorInfo object for more details.

**The SCPIStringParser Methods can NOT be used with:**
- SCPI Status Reporting. However, the *OPC? will work.
- Transferring Binary (block) data with commands such as: MMEM:TRAN or CALC:DATA with Format:Data set to Real32 or Real64

**Accessing the ScpiStringParser object**

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim SCPI As IScpiStringParser
Set SCPI = app.ScpiStringParser

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs
- See an example of how to return error information when using the Parse method.
```
<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parse</td>
<td>ISCPIStringParser</td>
<td>Provides the ability to send a SCPI command from within the COM command.</td>
</tr>
<tr>
<td>Execute</td>
<td>ISCPIStringParser2</td>
<td>Does not convert scpi errors. Use :SYST:ERR?</td>
</tr>
</tbody>
</table>

### Properties

None

### History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISCPIStringParser</td>
<td>1.0</td>
</tr>
<tr>
<td>ISCPIStringParser2</td>
<td>3.0</td>
</tr>
</tbody>
</table>
Segment Object

Description
Contains the methods and properties that affect a sweep segment.

**Note:** All of these properties are shared with at least one of the following objects: Channel, Cal Set, PowerSensorCalFactorSegment, or PowerLossSegment.

Accessing a Segment object and setting Segment Properties
You can get a handle to a sweep segment through the segments collection.

```
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim segs As ISegments
Set segs = app.ActiveChannel.Segments

segs.Add(1)
segs(1).NumberOfPoints = 30
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Segment Sweep
- Example Programs

Methods
None

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>centerFrequency</td>
<td>ISegment</td>
<td>Sets or returns the center frequency of the segment. Shared with the Channel Object</td>
</tr>
<tr>
<td>DwellTime</td>
<td>ISegment</td>
<td>Dwell time value. Shared with the Channel Object</td>
</tr>
<tr>
<td>FrequencySpan</td>
<td>ISegment</td>
<td>Sets or returns the frequency span of the segment. Shared with the Channel Object</td>
</tr>
<tr>
<td>IFBandwidth</td>
<td>ISegment</td>
<td>Sets or returns the IF Bandwidth of the segment. Shared with the Channel Object and Cal Set object.</td>
</tr>
</tbody>
</table>
**NumberOfPoints**  ISegment  Sets or returns the Number of Points of the segment.
Shared with the Channel Object

**SegmentNumber**  ISegment  Returns the number of the current segment.

**StartFrequency**  ISegment  Sets or returns the start frequency of the segment.
Shared with the Channel Object

**State**  ISegment  Turns On or OFF a segment.

**StopFrequency**  ISegment  Sets or returns the stop frequency of the segment.
Shared with the Channel Object

**SweepTime**  ISegment2  Sets or returns the sweep time of the segment.
Shared with the Channel Object

**TestPortPower**  ISegment  Sets or returns the RF power level of the segment.
Shared with the Channel Object

**ISegment History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISegment</td>
<td>1.0</td>
</tr>
<tr>
<td>ISegment2</td>
<td>7.1</td>
</tr>
</tbody>
</table>

Last modified:

9/29/06   MQQ Added Sweep time
Segments Collection

Description
The segment collection provides a mechanism for iterating through the sweep segments of a channel. Sweep segments are a potentially faster method of sweeping the analyzer through only the frequencies of interest. Learn more about Segment Sweep.

Accessing the Segments collection
There are two paths to the Segments Collection:

1. From the Channel object
2. From the FOMRange object

```
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim segs As ISegments
Set segs = app.ActiveChannel.Segments

or

Set segs = app.ActiveChannel.FOM.FOMRange(1).Segments
```

See Also:
- Segment Object to learn how to set the properties for individual segments.
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>ISegments</td>
<td>Adds an item to either the Segments collection.</td>
</tr>
<tr>
<td>Item</td>
<td>ISegments</td>
<td>Use to get a handle to a segment in the collection.</td>
</tr>
<tr>
<td>GetAllSegments</td>
<td>ISegments5</td>
<td>Downloads a segment table from the PNA</td>
</tr>
<tr>
<td>Remove</td>
<td>ISegments</td>
<td>Removes an item from a collection of objects.</td>
</tr>
<tr>
<td>SetAllSegments</td>
<td>ISegments2</td>
<td>Uploads a segment table to the PNA.</td>
</tr>
<tr>
<td>Properties</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>AllowArbitrarySegments</td>
<td>ISegments3 Enables the setup of arbitrary segment sweep</td>
<td></td>
</tr>
<tr>
<td>Count</td>
<td>ISegments Returns the number of items in a collection of objects.</td>
<td></td>
</tr>
<tr>
<td>IF Bandwidth Option</td>
<td>ISegments Enables the IFBandwidth to be set on individual sweep segments.</td>
<td></td>
</tr>
<tr>
<td>Parent</td>
<td>ISegments Returns a handle to the current naNetworkAnalyzer application.</td>
<td></td>
</tr>
<tr>
<td>Source Power Option</td>
<td>ISegments Enables setting the Source Power for a segment.</td>
<td></td>
</tr>
<tr>
<td>SweepTimeOption</td>
<td>ISegments4 Enables the Sweep time or Dwell time to be set independently on sweep segments.</td>
<td></td>
</tr>
</tbody>
</table>

### ISegments History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISegments</td>
<td>1.0</td>
</tr>
<tr>
<td>ISegments2</td>
<td>3.5</td>
</tr>
<tr>
<td>ISegments3</td>
<td>4.2</td>
</tr>
<tr>
<td>ISegments4</td>
<td>7.1</td>
</tr>
<tr>
<td>ISegments5</td>
<td>8.6</td>
</tr>
</tbody>
</table>

Last modified:

- 23-Jul-2009   Added more description
- 28-Apr-2009   Added GetAllSegments
- 8-Mar-2007    Modified access via fom
- 9/29/06       Added ISegments4
**SignalProcessingModuleFour Object**

**Description**
Contains the properties for configuring the DSP (digital filters) in the PNA-X.

![Digital Signal Processor (DSP) Block Diagram]

See the entire IF Path Block diagram.

**DSP Version 5 Notes**
Programs that control these settings, or state files that are saved, will yield different results when run or recalled on PNAs with DSP 4 versions versus DSP 5 Versions.
Stage 2 settings are IGNORED with DSP 5 Versions.

Learn more about DSP Versions

**Accessing the SignalProcessingModuleFour object**

```vba
Dim app as AgilentPNA835x.Application
Dim chan as Channel
Set chan = app.ActiveChannel
Dim digFilter as SignalProcessingModuleFour
Set digFilter = chan.SignalProcessingModuleFour
```

**See Also:**
- [PNA Automation Interfaces](#)
- [The PNA Object Model](#)
- [About PNA-X Pulse Capabilities](#)
- [Example Programs](#)

**Methods**

None

**Properties**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISPM4</td>
<td>Sets ADC capture mode: auto or manual</td>
</tr>
</tbody>
</table>

ADCCaptureMode
<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FilterErrors</td>
<td>ISPM4 Returns errors with manual digital filter settings</td>
</tr>
<tr>
<td>FilterMode</td>
<td>ISPM4 Sets digital filter mode: auto or manual</td>
</tr>
<tr>
<td>Stage1Coefficients</td>
<td>ISPM4 Sets Stage1Coefficients</td>
</tr>
<tr>
<td>Stage1Frequency</td>
<td>ISPM4 Sets Stage1 NCO frequency</td>
</tr>
<tr>
<td>Stage1MaximumCoefficient</td>
<td>ISPM4 Returns the maximum value of any single stage1 coefficient.</td>
</tr>
<tr>
<td>Stage1MaximumCoefficientCount</td>
<td>ISPM4 Returns the maximum number of Stage1 coefficients.</td>
</tr>
<tr>
<td>Stage1MaximumCoefficientSum</td>
<td>ISPM4 Returns the maximum sum of all Stage1 coefficients.</td>
</tr>
<tr>
<td>Stage1MinimumCoefficientCount</td>
<td>ISPM4 Returns the minimum number of Stage1 coefficients.</td>
</tr>
<tr>
<td>Stage2Coefficients</td>
<td>ISPM4 Sets Stage2 Coefficients</td>
</tr>
<tr>
<td>Stage2MaximumCoefficient</td>
<td>ISPM4 Returns the maximum value of any single stage2 coefficient.</td>
</tr>
<tr>
<td>Stage2MaximumCoefficientCount</td>
<td>ISPM4 Returns the maximum number of Stage2 coefficients.</td>
</tr>
<tr>
<td>Stage2MaximumCoefficientSum</td>
<td>ISPM4 Returns the maximum sum of all Stage2 coefficients.</td>
</tr>
<tr>
<td>Stage2MinimumCoefficientCount</td>
<td>ISPM4 Returns the minimum number of Stage2 coefficients.</td>
</tr>
<tr>
<td>Stage3FilterType</td>
<td>ISPM4 Sets and returns stage3 filter type</td>
</tr>
<tr>
<td>Stage3FilterTypes</td>
<td>ISPM4 Returns the names of supported types of Stage3 filters.</td>
</tr>
<tr>
<td>Stage3Parameter</td>
<td>ISPM4 Sets and returns the parameter value of the current filter type.</td>
</tr>
<tr>
<td>Stage3ParameterMaximum</td>
<td>ISPM4 Returns maximum parameter value for the current filter type.</td>
</tr>
</tbody>
</table>
Stage3ParameterMinimum ISPM4 Returns minimum parameter value for the current filter type.

Stage3Parameters ISPM4 Returns the names of parameters for the current filter type.

**ISignalProcessingModuleFour History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISignalProcessingModuleFour</td>
<td>7.2</td>
</tr>
</tbody>
</table>

Last Modified:

5-Jan-2007   MX New topic
SMCType Object

Description
Contains the methods and properties to perform an Scalar Measurement Calibration for the Frequency Converter Application (option 083).

Accessing the SMCType object
See an example which creates and calibrates an SMC measurement.
You can also do the following:

```vbscript
Set app = CreateObject("AgilentPNA835x.Application")
Set CalMgr = app.GetCalManager
Set guidedCal = CalMgr.CreateCustomCalEx(1)
Set SMC = guidedCal.CustomCalConfiguration
SMC.ConnectorType(1) = "APC 3.5 male"
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcquireStep</td>
<td>ISMCType</td>
<td>Acquire the measurement data for the specified step in the calibration process.</td>
</tr>
<tr>
<td>GenerateErrorTerms</td>
<td>ISMCType</td>
<td>Generates the error terms for the calibration.</td>
</tr>
<tr>
<td>GenerateSteps</td>
<td>ISMCType</td>
<td>Returns the number of steps required to complete the calibration.</td>
</tr>
<tr>
<td>GetStepDescription</td>
<td>ISMCType</td>
<td>Returns the description of the specified step calibration process.</td>
</tr>
<tr>
<td>ImportDataSet</td>
<td>ISMCType4</td>
<td>Imports separate power meter data for SMC cal.</td>
</tr>
<tr>
<td>Initialize</td>
<td>ISMCType</td>
<td>Begins a calibration.</td>
</tr>
</tbody>
</table>

739
<table>
<thead>
<tr>
<th>Property</th>
<th>ISMCType</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoOrient</td>
<td>ISMCType</td>
<td>Sets ECAL module automatic orientation ON or OFF.</td>
</tr>
<tr>
<td>CalibrationPort</td>
<td>ISMCType</td>
<td>Sets or returns the calibration source port for the calibration.</td>
</tr>
<tr>
<td>CalKitType</td>
<td>ISMCType</td>
<td>Sets and returns a calibration kit type for calibration.</td>
</tr>
<tr>
<td>CompatibleCalKits</td>
<td>ISMCType</td>
<td>Returns a list of cal kits that are compatible with the connector type for the specified port.</td>
</tr>
<tr>
<td>ConnectorType</td>
<td>ISMCType</td>
<td>Sets or queries the connector type for the specified port.</td>
</tr>
<tr>
<td>Do2PortEcal</td>
<td>ISMCType</td>
<td><strong>Superseded</strong> - Replaced by CalKitType Specify ECAL or Mechanical calibration.</td>
</tr>
<tr>
<td>EcalCharacterization</td>
<td>ISMCType</td>
<td><strong>Superseded</strong> - Replaced by CalKitType Specifies the characterization data within an ECal module to be used for the calibration.</td>
</tr>
<tr>
<td>EcalOrientation</td>
<td>ISMCType</td>
<td>Specifies which port of the ECal module is connected to which port of the PNA when the AutoOrient property = False.</td>
</tr>
<tr>
<td>FixedDelay</td>
<td>ISMCType5</td>
<td>Set and return the known delay through the calibration mixer.</td>
</tr>
<tr>
<td>MixerCharacterizationFile</td>
<td>ISMCType5</td>
<td>Set the filename of the S2P file used to characterize the calibration mixer.</td>
</tr>
<tr>
<td>NetworkFilename</td>
<td>ISMCType2</td>
<td>Specifies the S2P filename to embed or de-embed on the input or output of your mixer measurement.</td>
</tr>
<tr>
<td>NetworkMode</td>
<td>ISMCType2</td>
<td>Embed (add) or de-embed (remove) circuit network effects on the input and output of your mixer measurement.</td>
</tr>
<tr>
<td>OmitIsolation</td>
<td>ISMCType</td>
<td><strong>Superseded</strong> - Replaced by SetIsolationPaths and GetIsolationPaths Sets and returns whether Isolation portion of the calibration will be performed or not.</td>
</tr>
<tr>
<td>SeparatePowerCal</td>
<td>ISMCType4</td>
<td>Use a Thru standard or to use two power sensor connections during the SMC power cal</td>
</tr>
<tr>
<td>ThruCalMethod</td>
<td>ISMCType</td>
<td><strong>Superseded</strong> - Replaced by PathThruMethod Property Sets and returns the method for performing the thru portion of the calibration.</td>
</tr>
</tbody>
</table>
ValidConnectorTypes ISMCType Returns a list of connector types for which there are calibration kits.

### ISMCType History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISMCType</td>
<td>3.5</td>
</tr>
<tr>
<td>ISMCType2</td>
<td>6.0</td>
</tr>
<tr>
<td>ISMCType4</td>
<td>9.0</td>
</tr>
</tbody>
</table>
SourcePowerCalibrator Object

Description
This object is a child of the Application object and is a vehicle for performing source power calibrations.

Accessing the SourcePowerCalibrator Object

Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim ispc As ISourcePowerCalibrator
Set ispc = app.SourcePowerCalibrator

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

Note: Interface ISourcePowerCalibrator is abbreviated as ISPC in the following table.

(Bold Methods or Properties provide access to a child object)

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AbortPowerAcquisition</td>
<td>ISPC</td>
<td>Aborts a source power cal acquisition sweep that is currently in progress.</td>
</tr>
<tr>
<td>AcquirePowerReadings</td>
<td>ISPC</td>
<td>Superseded with AcquirePowerReadingsEx</td>
</tr>
<tr>
<td>AcquirePowerReadingsEx</td>
<td>ISPC4</td>
<td>Initiates a source power cal acquisition.</td>
</tr>
<tr>
<td>ApplyPowerCorrectionValues</td>
<td>ISPC</td>
<td>Superseded with ApplyPowerCorrectionValuesEx</td>
</tr>
<tr>
<td>ApplyPowerCorrectionValuesEx</td>
<td>ISPC5</td>
<td>Applies correction values after completing a source power cal acquisition sweep. Optionally perform a calibration of the reference receiver used in the source power cal.</td>
</tr>
<tr>
<td>CheckPower</td>
<td>ISPC2</td>
<td>Measures power at a specific frequency. Used to test power level before and/or after applying a source power calibration.</td>
</tr>
<tr>
<td>LaunchPowerMeterSettingsDialog</td>
<td>ISPC2</td>
<td>Launches the Power Meter Settings dialog on the PNA.</td>
</tr>
<tr>
<td>SetCalInfo</td>
<td>ISPC</td>
<td>Superseded with SetCalInfoEx Method</td>
</tr>
<tr>
<td>Method</td>
<td>Interface</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SetCalInfo2</td>
<td>ISPC3</td>
<td>Superseded with <strong>SetCalInfoEx Method</strong></td>
</tr>
<tr>
<td>SetCalInfoEx Method</td>
<td>ISPC4</td>
<td>Specifies the channel and source port to be used for the source power calibration.</td>
</tr>
<tr>
<td>SetPowerAcquisitionDevice</td>
<td>ISPC3</td>
<td>Sets the power sensor channel (A or B) to be used. This method is ONLY necessary when performing an SMC calibration.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalPower</td>
<td>ISPC</td>
<td>Specifies the power level that is expected at the desired reference plane.</td>
</tr>
<tr>
<td>IterationsTolerance</td>
<td>ISPC3</td>
<td>Sets the maximum desired deviation from the sum of the test port power and the offset value.</td>
</tr>
<tr>
<td>LastCalPassedTolerance</td>
<td>ISPC7</td>
<td>Returns pass / fail status of the user-specified tolerance limits on the target cal power.</td>
</tr>
<tr>
<td>MaximumIterationsPerPoint</td>
<td>ISPC3</td>
<td>Specifies maximum number of readings to take at each data point for iterating the source power.</td>
</tr>
<tr>
<td>PowerAcquisitionDevice</td>
<td>ISPC2</td>
<td>Specifies the power sensor channel (A or B) that is currently selected for use at a specific frequency.</td>
</tr>
<tr>
<td>PowerLossSegments (collection)</td>
<td>ISPC2</td>
<td>Collection for iterating through the segments of the power loss table used in source power calibration.</td>
</tr>
<tr>
<td>PowerMeterGPIBAddress</td>
<td>ISPC</td>
<td>Specifies the GPIB address of the power meter.</td>
</tr>
<tr>
<td>PowerMeterInterfaces</td>
<td>ISPC6</td>
<td>Collection for getting a handle to the available power meters.</td>
</tr>
<tr>
<td>PowerSensors (collection)</td>
<td>ISPC2</td>
<td>Collection for iterating through the PowerSensor objects which are connected to the power meter for a source power cal.</td>
</tr>
<tr>
<td>ReadingsPerPoint</td>
<td>ISPC</td>
<td>Specifies the maximum power readings for power meter settling.</td>
</tr>
<tr>
<td>ReadingsTolerance</td>
<td>ISPC3</td>
<td>Power meter settling tolerance value.</td>
</tr>
<tr>
<td>USBPowerMeterCatalog</td>
<td>ISPC6</td>
<td>Returns a list of USB power meters that are connected to the PNA.</td>
</tr>
<tr>
<td>UsePowerLossSegments</td>
<td>ISPC</td>
<td>Specifies if subsequent calls to the AcquirePowerReadings method will make use of the loss table (PowerLossSegments).</td>
</tr>
</tbody>
</table>
UsePowerSensorFrequencyLimits ISPC Specifies if subsequent calls to the AcquirePowerReadings method will make use of power sensor frequency checking capability.

**ISourcePowerCalibrator History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISourcePowerCalibrator</td>
<td>2.0</td>
</tr>
<tr>
<td>ISourcePowerCalibrator2</td>
<td>3.5</td>
</tr>
<tr>
<td>ISourcePowerCalibrator3</td>
<td>4.0</td>
</tr>
<tr>
<td>ISourcePowerCalibrator4</td>
<td>6.2</td>
</tr>
<tr>
<td>ISourcePowerCalibrator5</td>
<td>7.2</td>
</tr>
<tr>
<td>ISourcePowerCalibrator6</td>
<td>7.5</td>
</tr>
<tr>
<td>ISourcePowerCalibrator7</td>
<td>9.2</td>
</tr>
</tbody>
</table>
SweptIMD Object

Description
Controls the Swept IMD Application settings.
See Properties to set for each sweep type.

Accessing the SweptIMD object

```vba
Dim app as AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
app.CreateCustomMeasurementEx(1, "SweptIMD", "PwrMain", 1)
Dim IMD
Set IMD = app.ActiveChannel.CustomChannelConfiguration
```

See Also:

- Example Program Create and Cal a Swept IMD Measurement
- SweptIMDCal Object
- About the Swept IMD Application
- PNA Automation Interfaces
- The PNA Object Model

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>See History</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CompositeNormalizationMode</td>
<td>ISweptIMD</td>
<td>Sets and returns the method by which CTB and CSO calculations are performed.</td>
</tr>
<tr>
<td>CompositeNormalizedCSOPower</td>
<td>ISweptIMD</td>
<td>Sets and returns the CSO Power.</td>
</tr>
<tr>
<td>CompositeNormalizedCTBPower</td>
<td>ISweptIMD</td>
<td>Sets and returns the CTB Power.</td>
</tr>
<tr>
<td>CoupleTonePower</td>
<td>ISweptIMD</td>
<td>ON</td>
</tr>
<tr>
<td>Property</td>
<td>Module</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>CSO\text{NumDistortionProducts}</td>
<td>ISweptIMD</td>
<td>Sets the “N = number of distortion products” value for the calculation of the CSO parameter.</td>
</tr>
<tr>
<td>CSO\text{Offset Property}</td>
<td>ISweptIMD</td>
<td>Sets and returns the offset that is applied to CSO measurements.</td>
</tr>
<tr>
<td>CTB\text{Offset Property}</td>
<td>ISweptIMD</td>
<td>Sets and returns the offset that is applied to CTB measurements.</td>
</tr>
<tr>
<td>CTB\text{XMODNumCarriers}</td>
<td>ISweptIMD</td>
<td>Sets the “N = Total number of carriers” value used in the calculation of the XMOD and CTB parameter.</td>
</tr>
<tr>
<td>\text{DeltaFrequency}</td>
<td>ISweptIMD</td>
<td>Fixed tone spacing value.</td>
</tr>
<tr>
<td>\text{DeltaFrequencyStart}</td>
<td>ISweptIMD</td>
<td>Start spacing of the main tones.</td>
</tr>
<tr>
<td>\text{DeltaFrequencyStop}</td>
<td>ISweptIMD</td>
<td>Stop spacing of the main tones.</td>
</tr>
<tr>
<td>\text{DeviceInputPort}</td>
<td>ISweptIMD</td>
<td>Reads input port map</td>
</tr>
<tr>
<td>\text{DeviceOutputPort}</td>
<td>ISweptIMD</td>
<td>Reads output port map</td>
</tr>
<tr>
<td>\text{EqualTonePower}</td>
<td>ISweptIMD2</td>
<td>Set Equal Tone Power state.</td>
</tr>
<tr>
<td>\text{F1Frequency}</td>
<td>ISweptIMD</td>
<td>Frequency of the F1 tone.</td>
</tr>
<tr>
<td>\text{F2Frequency}</td>
<td>ISweptIMD</td>
<td>Frequency of the F1 tone.</td>
</tr>
<tr>
<td>\text{FrequencyCenter}</td>
<td>ISweptIMD</td>
<td>Center frequency of the main tones.</td>
</tr>
<tr>
<td>\text{FrequencyCenterCenter}</td>
<td>ISweptIMD</td>
<td>Sweep center frequency when sweeping the main tones.</td>
</tr>
<tr>
<td>\text{FrequencyCenterSpan}</td>
<td>ISweptIMD</td>
<td>Frequency span when sweeping the main tones.</td>
</tr>
<tr>
<td>\text{FrequencyCenterStart}</td>
<td>ISweptIMD</td>
<td>Start frequency when sweeping the main tones.</td>
</tr>
<tr>
<td>\text{FrequencyCenterStop}</td>
<td>ISweptIMD</td>
<td>Stop frequency when sweeping the main tones.</td>
</tr>
<tr>
<td>\text{HighestOrderProduct}</td>
<td>ISweptIMD</td>
<td>Reads the highest product that can be measured by SweptIMD.</td>
</tr>
<tr>
<td>\text{IMToneIFBandwidth}</td>
<td>ISweptIMD</td>
<td>IF Bandwidth for measurement of the intermodulation products.</td>
</tr>
</tbody>
</table>
MainToneIFBandwidth ISweptIMD IF Bandwidth for measurement of the Main tones.

SweepType ISweptIMD Type of sweep for a Swept IMD measurement.

TonePower ISweptIMD Power level of the Main Tones.

TonePowerSetAt ISweptIMD2 Set power level at DUT Input or Output

TonePowerStart ISweptIMD Start power level of the Main tones.

TonePowerStop ISweptIMD Stop power level of the Main tones.

The following commands are relevant for each IMD sweep type:

0 - naIMDToneCWSweep

- F1Frequency Property
- F2Frequency Property
- FrequencyCenter Property
- DeltaFrequency Property
- TonePower (1,2) Property

1 - naIMDTonePowerSweep

- F1Frequency Property
- F2Frequency Property
- FrequencyCenter Property
- DeltaFrequency Property
- TonePowerStart Property
- TonePowerStop Property

2 - naIMDToneCenterFreqSweep

- FrequencyCenterStart Property
- FrequencyCenterStop Property
- FrequencyCenterCenter Property
- **FrequencyCenterSpan Property**
- **DeltaFrequency Property**
- **TonePower (1,2) Property**

3 - **naIMDDeltaFrequencySweep**

- **DeltaFrequencyStart Property**
- **DeltaFrequencyStop Property**
- **FrequencyCenter Property**
- **TonePower (1,2) Property**

4 - **naIMDToneSegmentSweep** (Not available for IMDx)

- **DeltaFrequency Property**
- **TonePower (1,2) Property**
- Also use standard segment sweep commands

5 - **naLOPowerSweep** (IMDx ONLY)

- converter.**LOStartPower Property**
- converter.**LOStopPower Property**

**ISweptIMD History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISweptIMD</td>
<td>8.33</td>
</tr>
<tr>
<td>ISweptIMD2</td>
<td>9.40</td>
</tr>
</tbody>
</table>

Last Modified:

- 5-May-2011 Added tone power settings
- 11-Aug-2009 Added limited port map
- 25-Mar-2009 Fixed Access example
- 19-Aug-2008 MX New topic
**SweptIMDCal Object**

**Description**
Sets properties that are unique to a Swept IMD Cal (opt 087).
Use the [Guided Calibration commands](#) for the remaining commands to perform a Swept IMD Cal

**Accessing the SweptIMDCal object**

```vbnet
Dim app as AgilentPNA835x.Application
Set IMDcal = pna.GetCalmanager.CreateCustomCalEx(channelNum)
Set IMDCalExtension = IMDcal.CustomCalConfiguration
IMDCalExtension.PowerLevel = 5
```

**See Also:**
- Example Program [Create and Cal a Swept IMD Measurement](#)
- SweptIMD Object
- About Swept IMD measurements
- The PNA Object Model
- PNA Automation Interfaces

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalibrationFrequencies</td>
<td>ISweptIMDCal</td>
<td>Perform the source power cal at the center frequencies or at all main tone frequencies.</td>
</tr>
<tr>
<td>CalMethod</td>
<td>ISweptIMDCal</td>
<td>Sets the method by which the match-correction portion of an IMD cal is performed</td>
</tr>
<tr>
<td>EnableLOPowerCal</td>
<td>ISweptIMDCal2</td>
<td>Enable or disable an LO power cal with an IMDx calibration.</td>
</tr>
<tr>
<td>Include2ndOrderProduct</td>
<td>ISweptIMDCal</td>
<td>Include the second order products in the calibration.</td>
</tr>
</tbody>
</table>
MaxProduct ISweptIMDCal Sets and returns the maximum intermod product frequencies to be calibrated.

PowerLevel ISweptIMDCal Set and read the power level of the source power cal.

PowerSensorCalKitType ISweptIMDCal Set and read the cal kit to be used for port 1 adapter compensation.

<table>
<thead>
<tr>
<th>PowerSensorConnectorType</th>
<th>ISweptIMDCal</th>
<th>Superseded</th>
<th>Use GuidedCal PowerSensorConnectorType</th>
</tr>
</thead>
</table>

ISweptIMD History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISweptIMDCal</td>
<td>8.33</td>
</tr>
<tr>
<td>ISweptIMDCal2</td>
<td>8.55</td>
</tr>
</tbody>
</table>

Last Modified:

9-Sep-2008    MX New topic
TestsetControl Object

Description
A TestsetControl object is used to control one of the supported test sets. Only one external test set can be controlled by the PNA at any time. The Testset Control object appears as an item in the ExternalTestsets collection, which in turn is a property of the main application object.

If the specified test set is not connected to the PNA or is not ON, then setting Enabled = True will return an error. All other properties can be set even if the test set is not connected.

**Note:** The ONLY way to load a test set configuration file is by sending the testsets.Add method. There is no method to query the test set type. See an example program.

Accessing a TestsetControl object
The ExternalTestsets collection is a property of the main Application Object. You can obtain a handle to a testset object by specifying an item in the collection.

Visual Basic Example

```vbnet
Dim pna
Dim testsets As ExternalTestsets
Dim tset1 As TestsetControl
Set pna = CreateObject("AgilentPNA835x.Application")
Set testsets = pna.ExternalTestsets
Set tset1 = testsets(1)
' make COM calls on tset1 object
End Sub
```

See Also:
- E5091A Testset Object
- About External Testset Control
- ExternalTestset Control Example
- ExternalTestsets Collection
- The PNA Object Model
## Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(See history)</td>
<td>None</td>
</tr>
</tbody>
</table>

## Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ControlLines</td>
<td>IExternalTestset</td>
<td>Sets the control lines of the specified Test set.</td>
</tr>
<tr>
<td>Enabled</td>
<td>IExternalTestset</td>
<td>Enables and disables (ON/OFF) the port mapping and control line output of the specified test set.</td>
</tr>
<tr>
<td>ID</td>
<td>IExternalTestset</td>
<td>Returns the test set ID number.</td>
</tr>
<tr>
<td>Label</td>
<td>IExternalTestset</td>
<td>Returns the label on a given channel for the specified test set.</td>
</tr>
<tr>
<td>NumberOfPorts</td>
<td>IExternalTestset</td>
<td>Reads the number of ports that are on the specified test set.</td>
</tr>
<tr>
<td>OutputPorts</td>
<td>IExternalTestset</td>
<td>Sets or returns the port mappings for ALL ports.</td>
</tr>
<tr>
<td>PortCatalog</td>
<td>IExternalTestset</td>
<td>Returns the selections available for a given logical port.</td>
</tr>
<tr>
<td>SelectPort</td>
<td>IExternalTestset</td>
<td>Sets and returns the logical port value.</td>
</tr>
<tr>
<td>ShowProperties</td>
<td>IExternalTestset</td>
<td>Turns status bar display of test set properties on or off.</td>
</tr>
<tr>
<td>Type</td>
<td>IExternalTestset</td>
<td>Returns the test set model.</td>
</tr>
</tbody>
</table>

## ExternalTestset History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IExternalTestset</td>
<td>6.0</td>
</tr>
<tr>
<td>IExternalTestset</td>
<td>6.0</td>
</tr>
</tbody>
</table>
Trace Object

Description
The Trace object controls how the measurement data is displayed. You can control scale, reference position, and value from the Trace Object.

Accessing a Trace object
There are several ways to get a handle to a trace.

```vbscript
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim trace As Trace

Then you can do any of the following:

Set trace = app.NAWindows(1).traces(1)
set trace = app.NAWindows.item(1).ActiveTrace
set trace = app.ActiveNAWindow.traces.item(1)
set trace = app.ActiveNAWindow.ActiveTrace
Set trace = app.Measurements(1).trace
Set trace = app.ActiveMeasurement.trace
```

See Also:

- PNA Automation Interfaces
- The PNA Object Model
- Traces, Channels, and Windows on the PNA
- Example Programs
## Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autoscale</td>
<td>Autoscales the trace or all of the traces in the selected window.</td>
</tr>
<tr>
<td></td>
<td>Shared with the NAWindow Object</td>
</tr>
</tbody>
</table>

## Property

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measurement</td>
<td>Returns a measurement handle from trace object.</td>
</tr>
<tr>
<td>Name</td>
<td>Sets or returns the trace name</td>
</tr>
<tr>
<td>ReferencePosition</td>
<td>Sets or returns the Reference Position of the active trace.</td>
</tr>
<tr>
<td>ReferenceValue</td>
<td>Sets or returns the value of the Y-axis Reference Level of the active trace.</td>
</tr>
<tr>
<td>YScale</td>
<td>Sets or returns the Y-axis Per-Division value of the active trace.</td>
</tr>
</tbody>
</table>

## ITrace History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ITrace</td>
<td>1.0</td>
</tr>
<tr>
<td>ITrace2</td>
<td>9.40</td>
</tr>
</tbody>
</table>
Traces Collection

Description
Child of the Application Object. A collection that provides a mechanism for getting a handle to a trace or iterating through the traces in a window.

Accessing the Traces collection
Get a handle to the traces collection through the NaWindows collection. The following example sets the variable trcs to the collection of traces in window 1 of the NaWindows collection.

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim trcs As traces
Set trcs = app.NAWindows(1).traces
```

See Also:
- Trace Object
- Collections in the Analyzer
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item</td>
<td>Use to get a handle to a trace</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Count</td>
<td>Returns the number of traces in the collection.</td>
</tr>
<tr>
<td>Parent</td>
<td>Returns a handle to the current Application.</td>
</tr>
</tbody>
</table>
Transform Object

Description
Contains the methods and properties that control Time Domain transforms.

Accessing the Transform Object

```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)

Dim trans As Transform
Set trans = app.ActiveMeasurement.Transform
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Time Domain Topics
- Example Programs

Note: Sweep Type must be set to Linear before setting Time Domain Transform (state) ON.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SetFrequencyLowPass</td>
<td>ITransform</td>
<td>Sets low frequencies for low pass.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center</td>
<td>Sets or returns the Center time.</td>
</tr>
<tr>
<td></td>
<td>Shared with the Gating Object</td>
</tr>
<tr>
<td>CoupledParameters</td>
<td>Select Transform parameters to couple</td>
</tr>
<tr>
<td>DistanceMarkerMode</td>
<td>Sets the measurement type in order to determine the correct marker distance.</td>
</tr>
<tr>
<td>DistanceMarkerUnit</td>
<td>Sets the unit of measure for the display of marker distance values.</td>
</tr>
<tr>
<td>ImpulseWidth</td>
<td>Sets or returns the Impulse Width of Time Domain transform windows.</td>
</tr>
<tr>
<td>KaiserBeta</td>
<td>Sets or returns the Kaiser Beta of Time Domain transform windows.</td>
</tr>
<tr>
<td>Mode</td>
<td>Sets the type of transform.</td>
</tr>
</tbody>
</table>
Span ITransform Sets or returns the Span time.
Shared with the Gating Object

Start ITransform Sets or returns the Start time.
Shared with the Gating Object

State ITransform Turns an Object ON and OFF.

StepRiseTime ITransform Sets or returns the Rise time of the stimulus in Low Pass Step Mode.

Stop ITransform Sets or returns the Stop time.
Shared with the Gating Object

**ITransform History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ITransform</td>
<td>1.0</td>
</tr>
<tr>
<td>ITransform2</td>
<td>4.2</td>
</tr>
</tbody>
</table>
**TriggerSetup Object**

**Description**
These properties setup Global triggering that effects the entire PNA application.

**Accessing the TriggerSetup object**

```vba
Dim app as AgilentPNA835x.Application
Dim trigSetup as ITriggerSetup
Set trigSetup = app.TriggerSetup
```

**See Also:**

- PNA Automation Interfaces
- The PNA Object Model
- Triggering in the PNA
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>See History (below)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcceptTriggerBeforeArmed</td>
<td>ITriggerSetup2</td>
</tr>
<tr>
<td>ExternalTriggerConnectionBehavior</td>
<td>ITriggerSetup</td>
</tr>
<tr>
<td>ReadyForTriggerPolarity</td>
<td>ITriggerSetup3</td>
</tr>
<tr>
<td>Scope</td>
<td>ITriggerSetup</td>
</tr>
<tr>
<td>Source</td>
<td>ITriggerSetup</td>
</tr>
</tbody>
</table>
TriggerOutputEnabled

ITriggerSetup2 Enables the PNA to send trigger signals out the rear-panel TRIGGER OUT connector.

ITriggerSetup History

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ITriggerSetup</td>
<td>4.0</td>
</tr>
<tr>
<td>ITriggerSetup2</td>
<td>4.2</td>
</tr>
<tr>
<td>ITriggerSetup3</td>
<td>7.50.2 and 8.2</td>
</tr>
</tbody>
</table>
VMC Type Object

Description
Contains the methods and properties to perform a Vector Measurement Calibration for the Frequency Converter Application (option 083).

Accessing the VMCType object
See an example which creates and calibrates a VMC measurement.
You can also do the following:

```vbs
Set app = CreateObject("AgilentPNA835x.Application")
Set CalMgr = app.GetCalManager
Set guidedCal = CalMgr.CreateCustomCalEx(1)
Set VMC = guidedCal.CustomCalConfiguration
VMC.ConnectorType(1) = "APC 3.5 male"
```

See Also:
- PNA Automation Interfaces
- The PNA Object Model
- Example Programs

<table>
<thead>
<tr>
<th>Methods</th>
<th>Interface</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AcquireStep</td>
<td>IVMCType</td>
<td>Acquire the measurement data for the specified step in the calibration process.</td>
</tr>
<tr>
<td>GenerateErrorTerms</td>
<td>IVMCType</td>
<td>Generates the error terms for the calibration.</td>
</tr>
<tr>
<td>GenerateSteps</td>
<td>IVMCType</td>
<td>Returns the number of steps required to complete the calibration.</td>
</tr>
<tr>
<td>GetStepDescription</td>
<td>IVMCType</td>
<td>Returns the description of the specified step in the calibration process.</td>
</tr>
<tr>
<td>Initialize</td>
<td>IVMCType</td>
<td>Begins a calibration.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Properties</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AutoOrient</td>
<td>Sets ECAL module automatic orientation ON or OFF.</td>
</tr>
<tr>
<td>Function</td>
<td>Type</td>
</tr>
<tr>
<td>---------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>CalKitType</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>CharacterizeMixerOnly</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>CharFileName</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>CharMixerReverse</td>
<td>IVMCTYPE2</td>
</tr>
<tr>
<td>CompatibleCalKits</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>ConnectorType</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>Do1PortEcal</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>Do2PortEcal</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>EcalCharacterization</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>EcalOrientation1Port</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>EcalOrientation2Port</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>LoadCharFromFile</td>
<td>IVMCTYPE</td>
</tr>
<tr>
<td>NetworkFilename</td>
<td>IVMCTYPE3</td>
</tr>
<tr>
<td>NetworkMode</td>
<td>IVMCTYPE3</td>
</tr>
<tr>
<td>OmitIsolation</td>
<td>IVMCTYPE</td>
</tr>
</tbody>
</table>
**ThruCalMethod**  IVMCType  **Superseded** - Replaced by PathThruMethod Property

Sets and returns the method for performing the thru portion of the calibration.

**ValidConnectorTypes**  IVMCType

Returns a list of connector types for which there are calibration kits.

---

**IVMCType History**

<table>
<thead>
<tr>
<th>Interface</th>
<th>Introduced with PNA Rev:</th>
</tr>
</thead>
<tbody>
<tr>
<td>IVMCType</td>
<td>3.5</td>
</tr>
<tr>
<td>IVMCType2</td>
<td>3.53</td>
</tr>
<tr>
<td>IVMCType3</td>
<td>6.0</td>
</tr>
</tbody>
</table>
AcceptTriggerBeforeArmed Property

**Description**
Determines what happens to an EDGE trigger signal if it occurs before the PNA is ready to be triggered. (LEVEL trigger signals are always ignored.) For more information, see [External triggering](#).

**VB Syntax**
```vbnet
trigsetup.AcceptTriggerBeforeArmed = boolean
```

**Variable**
- **trigsetup**
  - A `TriggerSetup2` object
- **boolean**
  - Choose from:
    - **False** - A trigger signal is ignored if it occurs before the PNA is ready to be triggered.
    - **True** - A trigger signal is remembered and then used when the PNA becomes armed (ready to be triggered). The PNA remembers only one trigger signal.

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
trigsetup.AcceptTriggerBeforeArmed = True 'Write

atba = trigsetup.AcceptTriggerBeforeArmed 'Read
```

**C++ Syntax**
```cpp
HRESULT get_AcceptTriggerBeforeArmed( BOOL *pVal);
HRESULT put_AcceptTriggerBeforeArmed( BOOL newVal);
```

**Interface**
`ITriggerSetup2`
Read / Write

About Performing a Calibration

AcquisitionDirection Property

Description Specifies the direction of each part of a 2-port calibration.

VB Syntax `cal.AcquisitionDirection = value`

Variable (Type) - Description
- `cal` A Calibrator (object)
- `value` (enum NADirection) - Choose from:
  - 0 - naForward - measures the forward direction
  - 1 - naReverse - measures the reverse direction

Return Type Long Integer

Default naForward

Examples `cal.AcquisitionDirection = naForward`

C++ Syntax `HRESULT AcquisitionDirection(tagNADirection dir);`

Interface ICalibrator
AcquisitionMode Property

**Description**
Set and read the method by which gain compression data is acquired.

**VB Syntax**
gca.AquisitionMode = value

**Variable**
(Variable) - Description

- **gca** A GainCompression (object)

- **value** (NAGCAAcquisitionMode) Choose from:
  - naSmartSweep (0)  Iterate quickly to find compression point
  - naSweepPowerAtEachFreq2D (1)  Sweep power at each frequency
  - naSweepFreqAtEachPower2D (2)  Sweep frequency at each power level

**Return Type**
Enum

**Default**
naSmartSweep

**Examples**
gca.AquisitionMode = naSmartSweep 'Write
acqMode = gca.AquisitionMode 'Read

**C++ Syntax**
HRESULT get_AcquisitionMode(tagNAGCAAcquisitionMode* mode)
HRESULT put_AcquisitionMode(tagNAGCAAcquisitionMode mode)

**Interface**
IGainCompression

Last Modified:
11-Sep-2007   MX New topic
**ActiveCalKit Property**

**Description**

Returns a handle to the Active CalKit object. The active cal kit is the kit selected for use in Unguided calibrations.

You can either (1) use the handle directly to access CalKit properties and methods, or (2) set a variable to the CalKit object. The variable retains a handle to the original object if another CalKit becomes active.

**VB Syntax**

1) `app.ActiveCalKit.<setting>`

or

2) `Set cKit = app.ActiveCalKit`

**Variable (Type)** - Description

- `app` - An Application (object)
- `<setting>` - A CalKit property (or method) and arguments
- `cKit` - A CalKit object

**Return Type**

CalKit object

**Default**

None

**Examples**

```vbnet
Public cKit as CalKit
Set cKit = app.ActiveCalKit 'read
```

**C++ Syntax**

```cpp
HRESULT get_ActiveCalKit (ICalkit * kit)
```

**Interface**

IApplication

---

**Last Modified:**

3-Jun-2008   Added Unguided
ActiveChannel Property

Description
Returns a handle to the Active Channel object. You can either (1) use the handle directly to access channel properties and methods, or (2) set a variable to the channel object. The variable retains a handle to the original channel if another channel becomes active.

VB Syntax
(1) `app.ActiveChannel.<setting>`
or
(2) Set `chan = app.ActiveChannel`

Variable (Type) - Description
- `chan`: A Channel (object)
- `app`: An Application (object)
- `<setting>`: A channel property (or method) and arguments

Return Type
Channel object

Default
Not applicable

Examples
1) `app.ActiveChannel.Averaging = 1`
2) `Public chan as Channel
   Set chan = app.ActiveChannel`

C++ Syntax
`HRESULT get_ActiveChannel(IChannel** pVal)`

Interface
`IApplication`
Active (ExtDev) Property

**Description**
Sets and returns the state of activation for an external device. When true, device is available for use in measurements.

**Note:** Send this command AFTER sending other External Device settings to avoid communicating with the device before it has been fully configured.

See Also **ExternalDeviceDeActivatePolicy Property** - Determines whether External Devices remain activated or are de-activated when the PNA is Preset or when a Instrument State is recalled.

**VB Syntax**
```
extDevices.Active = value
```

**Variable**
*(Type)* - Description

- **extDevices** An **ExternalDevice** *(object)*
- **value** (Boolean) Choose from:
  - **True** - Device is active.
  - **False** - Device is NOT active.

**Return Type**
Boolean

**Default**
False - When configured using the front panel user interface, the device is ON (activated) by default.

**Examples**
```
extDevices.Active = True 'Write
bool = extDevices.Active 'Read
See example program to configure PMAR device
See example program to configure External Source
```

**C++ Syntax**
```
HRESULT get_Active( VARIANT_BOOL* value);
HRESULT put_Active( VARIANT_BOOL newVal);
```

**Interface**
IEternalDevices

---

Last Modified:
31-Jul-2009  MX New topic
### ActiveMarker Property

**Description**
Returns a handle to the Active Marker object. You can either (1) use the handle directly to access Marker properties and methods, or (2) set a variable to the Marker object. The variable retains a handle to the original object if another Marker becomes active.

**VB Syntax**
1) `meas.ActiveMarker.<setting>`
   
   or
   
   2) Set `mark = meas.ActiveMarker`

**Variable (Type) - Description**

- **meas (object)** - An Measurement object
- **<setting>** - A marker property (or method) and arguments
- **mark (object)** - A marker object

**Return Type**
marker object

**Default**
None

**Examples**
```vbnet
Public mark as marker
Set mark = meas.ActiveMarker
```

**C++ Syntax**
`HRESULT get_ActiveMarker(IMarker** marker)`

**Interface**
IMeasurement
### ActiveMeasurement Property

**Description**
Returns a handle to the Active Measurement object. You can either (1) use the handle directly to access measurement properties and methods, or (2) set a variable to the measurement object. The variable retains a handle to the original measurement.

**VB Syntax**
1) `app.ActiveMeasurement.<setting>`
   or
2) `Set meas = app.ActiveMeasurement`

**Variable ** *(Type) - Description*

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>meas</td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td>app</td>
<td>An Application <em>(object)</em></td>
</tr>
<tr>
<td>&lt;setting&gt;</td>
<td>A measurement property (or method) and arguments</td>
</tr>
</tbody>
</table>

**Return Type**
Measurement object

**Default**
None

**Examples**
1) `app.ActiveMeasurement.Averaging = 1`
2) `Public meas as Measurement
   Set meas = app.ActiveMeasurement`

**C++ Syntax**
`HRESULT get_ActiveMeasurement(IMeasurement **ppMeas)`

**Interface**
IApplication
ActiveNAWindow Property

**Description**
Returns a handle to the Active Window object. You can either (1) use the handle directly to access window properties and methods, or (2) set a variable to the window object. The variable retains a handle to the original window if another window becomes active.

**VB Syntax**
1) `app.ActiveNAWindow.<setting>`
   or
2) Set `win = app.ActiveNAWindow`

**Variable**
- **(Type) - Description**
  - `win` A NAWindow *(object)*
  - `app` An Application *(object)*
  - `<setting>` A NAWindow property (or method) and arguments

**Return Type**
A NAWindow object

**Default**
Not applicable

**Examples**
- Public `win as NAWindow`
- `Set win = app.ActiveWindow`

**C++ Syntax**
```
HRESULT get_ActiveNAWindow(INAWindow **ppWindow)
```

**Interface**
IApplication
**About Traces**

**ActiveTrace Property**

**Description**
Returns a handle to the Active Trace object. You can either (1) use the handle directly to access trace properties and methods, or (2) set a variable to the trace object. The variable retains a handle to the original trace if another trace becomes active.

**VB Syntax**
1) `win.ActiveTrace.<setting>`  
   or  
   2) Set `trce = win.ActiveTrace`

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>trce</code></td>
<td>A Trace <em>(object)</em></td>
</tr>
<tr>
<td><code>win</code></td>
<td>An NAWindow <em>(object)</em></td>
</tr>
<tr>
<td><code>&lt;setting&gt;</code></td>
<td>A trace property (or method) and arguments</td>
</tr>
</tbody>
</table>

**Return Type**
An NAWindow object

**Default**
None

**Examples**
1) `win.ActiveTrace.Autoscale`
2) `Public trce as Trace`  
   `Set trce = Application.ActiveNAWindow.ActiveTrace`

**C++ Syntax**
`HRESULT get_ActiveTrace(ITrace* *pVal)`

**Interface**
INAWindow
ActiveBackground Property

**Description**
Set and return the background color of the active window for the PNA display or hardcopy print.

**VB Syntax**
```vbnet
colors.ActiveBackground = value
```

**Variable (Type) - Description**
- `colors` A `ComColors` (object)
- `value` (Long Integer) - RGB color of the ActiveBackground pen.
  
  Convert the three RGB colors to an integer as follows:
  ```
  RGB = R + (G*2^8) + (B*2^16)
  ```
  To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

**Return Type**
Long

**Default**
Display = 0,0,24

**Examples**
```vbnet
R = 10
G = 10
B = 10
RGB = R + (G*2^8) + (B*2^16)
colors.ActiveBackground = RGB 'Write
color = colors.ActiveBackground 'Read
```

**C++ Syntax**
```csharp
HRESULT get_ActiveBackground(long* pVal);
HRESULT put_ActiveBackground(long newVal);
```

**Interface**
IComColors2

---

Last Modified:
22-Feb-2010   MX New topic
## ActiveLabels Property

**Description**
Set and return the labels and grid frame colors in the active window for the PNA display or hardcopy print. (Active labels, Grid frame)

**VB Syntax**
```vbnet
colors.ActiveLabels = value
```

**Variable**
- **Type**
  - colors: A `ComColors` (object)
- **value**: (Long Integer) - RGB color of the ActiveLabels pen.
  - Convert the three RGB colors to an integer as follows:
    ```
    RGB = R+(G*2^8)+(B*2^16)
    ```
  - To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

**Return Type**
Long

**Default**
- Display = 175,175,175
- Print = 0,0,0 (Black)

**Examples**
```vbnet
R = 10
G = 10
B = 10
RGB = R+(G*2^8)+(B*2^16)
colors.ActiveLabels = RGB 'Write
color = colors.ActiveLabels 'Read
```

**C++ Syntax**

```
HRESULT get_ActiveLabels(long* pVal);
HRESULT put_ActiveLabels(long newVal);
```

**Interface**
IComColors

---

Last Modified:
7-Aug-2009   MX New topic
**ActiveXAxisRange Property**

**Description**  
For FCA and GCX measurements, sets the swept parameter to display on the X-axis. This command does not change the default setting for new traces.

Use `Converter.ActiveXAxisRange` to change all existing traces and make the setting the default setting for new traces.

This command is NOT used for NFX, IMDX, and IMSX measurements. Use `Converter.ActiveXAxisRange`.

**VB Syntax**  
`mixer.ActiveXAxisRange = value`

**Variable**  
*(Type)* - Description

- `mixer`  
  A Mixer *(object)*

- `value`  
  *(Enum as MixerStimulusRange)* - Parameter to display on the X-axis. Choose from:
  - 0 - `mixINPUT` - Input frequency span
  - 1 - `mixLO_1` - First LO frequency span
  - 2 - `mixLO_2` - Second LO frequency span
  - 3 - `mixOUTPUT` - Output frequency span

**Return Type**  
Enum

**Default**  
OUTPUT

**Examples**  
`mixer.ActiveXAxisRange = 1`  
`variable = mixer.ActiveXAxisRange`

**C++ Syntax**  
`HRESULT get_ActiveXAxisRange(tagMixerStimulusRange *Val)`  
`HRESULT put_ActiveXAxisRange(tagMixerStimulusRange newVal)`

**Interface**  
IMixer3

Last Modified:

- 27-Sep-2010  
  Clarification

- 26-Jan-2009  
  Corrected enums
ActiveXAxisRange Property

**Description**
For all converter applications, sets the swept frequency range to display on the X-axis for all existing traces and sets the default for all future traces.

**VB Syntax**
```
obj.ActiveXAxisRange = value
```

**Variable**
- **obj** (Type) - Description
  - A Converter Object
- **value** (Enum as ConverterStimulusRange)
  - Swept stimulus range to display on the X-axis. Choose from:
    - 0 - naInputRange - Input frequency range
    - 1 - naLO1Range - LO 1 frequency range
    - 2 - naLO2Range - LO 2 frequency range
    - 3 - naOutputRange - Output frequency range
    - 4 - naPerMeasurementRange - reserved for future use.
  - If the specified frequency range is not swept, the default swept range is used.

**Return Type**
Enum

**Default**
Search is performed in the following order until a swept range is found:

1. OUTPUT
2. INPUT (If the OUTPUT is fixed)
3. Number of Points (If ALL ranges are fixed)

**Examples**
```
conv.ActiveXAxisRange = naInputRange 'Write
variable = conv.ActiveXAxisRange 'Read
```

**C++ Syntax**
```
HRESULT get_ActiveXAxisRange(tagConverterStimulusRange range *Val)
HRESULT put_ActiveXAxisRange(tagConverterStimulusRange range newVal)
```

**Interface**
IConverter

Last Modified:
- 9-Sep-2010  Added LO2 range and Modified for all converter apps.
- 2-Feb-2009  Added converter
- 26-Jan-2009  Corrected enums
**ADCCaptureMode Property**

**Description**
Sets and returns the ADC capture mode modeled as a 2-pole switch in the diagram on the SignalProcessingModuleFour page. The switch either bypasses or routes the IF through the 3-stage digital filter.

**VB Syntax**

```
spm4.ADCCaptureMode = value
```

**Variable** *(Typing)* - *Description*

- `spm4` - A SignalProcessingModuleFour (object)
- `value` - (Enum as NAStates) Capture mode.
  - **naOFF (0)** - The digital filters are used to process IF information. The filters can be configured automatically or manually using FilterMode Property.
  - **naON (1)** - The digital filters are bypassed and the raw ADC readings are taken directly. With DSP 4 versions, a maximum of 4096 data points per sweep can be acquired.
  - With DSP 5 versions, the PNA maximum data points per sweep can be acquired. Learn more about DSP Versions.

**Return Type**
Enum

**Default**
OFF

**Examples**

```
spm4.ADCCaptureMode = 0 'Write
mode = spm4.ADCCaptureMode 'Read
```

**C++ Syntax**

```
HRESULT get_ADCCaptureMode(tagNAStates* pCaptureMode);
HRESULT put_ADCCaptureMode(tagNAStates pCaptureMode);
```

**Interface**
ISignalProcessingModuleFour

---

**Last Modified:**

- 26-Aug-2010  Updated for DSP 5 (A.09.30)
- 18-Jun-2007  MX New topic
Sets and returns the ALC mode for the specified channel and port. Use `GetSupportedALCModes` to return a list of valid ALC modes for the PNA.

Learn more about ALC mode.

VB Syntax

```
chan.ALCLevelingMode(sourcePort) = value
```

Variable (Type) - Description

- `chan` (object) - A `Channel` object
- `sourcePort` (long integer) - The source port for which to make this setting. If ports are remapped, specify the logical port number.
  
  **Note**: If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

- `value` (enum as `naALCLevelingMode`) - Choose from:
  
  - 0 `naALCInternal`
  - 1 `naALCExternal` (E835x Only)
  - 2 `naALCOpenLoop` (PNA-X only)

Return Type

Enum

Default

`naALCInternal`

Examples

```
chan.ALCLevelingMode(1) = 'Write
state = chan.ALCLevelingMode(4) 'Read
```

C++ Syntax

```
HRESULT get_ALCLevelingMode(long port, tagNAALCLevelingMode* pVal);
HRESULT put_ALCLevelingMode(long port,tagNAALCLevelingMode newVal);
```

Interface

`IChannel9`
3-Mar-2009  Removed 3
24-Apr-2008  Added note for string names
30-Apr-2007  Edited for src strings
10/18/06     MX New topic
About Segment Sweep

AllowArbitrarySegments Property

Description
Enables you to setup a segment sweep with arbitrary frequencies. The start and stop frequencies of each segment can overlap other segments. Also, each segment can have a start frequency that is greater than its stop frequency which causes a reverse sweep over that segment. Learn more about Arbitrary Segment Sweep.

VB Syntax
`segs.AllowArbitrarySegments = value`

Variable (Type) - Description
- `segs` A `Segments` collection (object)
- `value` (boolean)
  - `True` - Allows the setup of arbitrary segment sweep.
  - `False` - Prevents the setup of arbitrary segment sweep.

Return Type
Boolean

Default
False

Examples
`segs.AllowArbitrarySegments = True 'Write`

`AllowArbSegs = AllowArbitrarySegments 'Read`

C++ Syntax
- `HRESULT get_AllowArbitrarySegments(VARIANT_BOOL *pVal)`
- `HRESULT put_AllowArbitrarySegments(VARIANT_BOOL newVal)`

Interface
ISegments3
AlternateSweep Property

Description  
Sets sweeps to either alternate or chopped.

VB Syntax  
`object.AlternateSweep = value`

Variable (Type) - Description

- **object**  
  Channel (object)
  
  or
  
  CalSet (object) - Read-only property

- **value** (boolean) - Choose either:
  
  - **False** - Sweep mode set to Chopped - reflection and transmission are measured on the same sweep.
  
  - **True** - Sweep mode set to Alternate - reflection and transmission measured on separate sweeps. Improves Mixer bounce and Isolation measurements. Increases cycle time.

Return Type  
boolean

Default  
False (0)

Examples

`chan.AlternateSweep = True 'Write`

`altSwp = chan.AlternateSweep 'Read`

C++ Syntax

`HRESULT AlternateSweep(VARIANT_BOOL *pVal)`

`HRESULT AlternateSweep(VARIANT_BOOL newVal)`

Interface  
IChannel

ICalSet3
AmbientTemperature Property

**Description**
Sets and returns the temperature at which the current noise measurement is occurring. [Learn more.](#)

**VB Syntax**
noiseCal.AmbientTemperature = value

**Variable**
*noiseCal* (Type) - Description

噪声Cal (object)

*value* (double) Ambient temperature in Kelvin.

**Return Type**
Double

**Default**
295

**Examples**
noise.AmbientTemperature = 289 'Write

temp = noise.AmbientTemperature 'Read

**C++ Syntax**
HRESULT get_AmbientTemperature(Double* pValue)

HRESULT put_AmbientTemperature(Double pNewValue)

**Interface**
INoiseCal

Last Modified:
6-Sep-2007   MX New topic
### AnalysisCWFreq Property

**Description**  
Set and return the CW frequency for a compression analysis trace.

**VB Syntax**  
gcaMeas.AnalysisCWFreq = value

**Variable**  
(� Type) - Description

gcaMeas A GainCompressionMeas (object)

value (Double) CW frequency in Hz. Choose a frequency within the range of the gain compression channel.

**Return Type**  
Double

**Default**  
Not Applicable

**Examples**  
gcaMeas.AnalysisCWFreq = 1e9  'Write

cwfreq = gca.AnalysisCWFreq  'Read

**C++ Syntax**  
HRESULT get_AnalysisCWFreq(Double* value)  
HRESULT put_AnalysisCWFreq(Double value)

**Interface**  
IGainCompressionMeas

---

Last Modified:

3-Sep-2009   MX New topic
**AnalysisEnable Property**

**Description**  
Set and read the (ON | OFF) state of Gain Compression Analysis.

**VB Syntax**  
gcaMeas.AnalysisEnable = value

**Variable**  
*Type* - Description

gcaMeas  
A GainCompressionMeas (object)

value  
(Boolean)  
Choose from:

- **False** - Disable GCA analysis trace.
- **True** - Enable GCA analysis trace.

**Return Type**  
Boolean

**Default**  
False

**Examples**  
gcaMeas.AnalysisEnable = True 'Write

analysis = gca.AnalysisEnable 'Read

**C++ Syntax**  
HRESULT get_AnalysisEnable(VARIANT_BOOL* value)

HRESULT put_AnalysisEnable(VARIANT_BOOL value)

**Interface**  
IGainCompressionMeas

---

Last Modified:

3-Sep-2009  
MX New topic
### AnalysisIsDiscreteFreq Property

**Description**
Sets and returns whether the CW frequency for the compression analysis trace can be set to only the discrete frequencies or provides interpolation.

**VB Syntax**
`gcaMeas.AnalysisIsDiscreteFreq = value`

**Variable**
**Type** - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>gcaMeas</code></td>
<td>A <code>GainCompressionMeas</code> object</td>
</tr>
</tbody>
</table>

**value**
**Type** - Description

<table>
<thead>
<tr>
<th><code>value</code></th>
<th>Choose from:</th>
</tr>
</thead>
<tbody>
<tr>
<td>False</td>
<td>Interpolated data points.</td>
</tr>
<tr>
<td>True</td>
<td>Discrete data points only.</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
'Write
gcaMeas.AnalysisIsDiscreteFreq = True

isDisc = gca.AnalysisIsDiscreteFreq 'Read
```

**C++ Syntax**

```cpp
HRESULT get_AnalysisIsDiscreteFreq(VARIANT_BOOL* value)
HRESULT put_AnalysisIsDiscreteFreq(VARIANT_BOOL value)
```

**Interface**
`IGainCompressionMeas`

---

**Last Modified:**
3-Sep-2009

MX New topic
Write/Read

AnalysisXAxis Property

Description
Sets and returns the type of data to display on the x-axis of a compression analysis trace.

VB Syntax
```
gcaMeas.AnalysisXAxis = value
```

Variable 
**gcaMeas** A GainCompressionMeas (object)

**value** (Enum as NAGCAAnalysisXAxis) Choose from:
- naPsourceAsXAxis (0) - Power from the source.
- naPinAsXAxis (1) - Input power to the DUT.

Return Type
Enum

Default
naPinAsXAxis (1)

Examples
```
gcaMeas.AnalysisXAxis = naPinAsXAxis  'Write

xAxis = gca.AnalysisXAxis  'Read
```

C++ Syntax
```
HRESULT get_AnalysisXAxis(tagNAGCAAnalysisXAxis* value)
HRESULT put_AnalysisXAxis(tagNAGCAAnalysisXAxis  value)
```

Interface
IGainCompressionMeas

Last Modified:
3-Sep-2009   MX New topic
<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the name of the Analyzer making measurements on the channel.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>chan.Application</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>chan</code></td>
<td>A <code>Channel</code> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>object</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>None</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>rfna = chan.Application</code> 'returns the Analyzer name'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_Application(IApplication** Application)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IChannel</td>
</tr>
</tbody>
</table>
ArrangeWindows Property

**Description**  
Sets the arrangement of all the windows. Overlay, Stack2, Split3 and Quad4 will create windows.

To control the state of one window, use `app.WindowState`.

**VB Syntax**  
`app.ArrangeWindows = value`

**Variable**  
(Type) - Description

- `app` An `Application` (object)
- `value` (enum NAWindowModes) - Choose from:
  - 0 - naTile
  - 1 - naCascade
  - 2 - naOverlay
  - 3 - naStack2
  - 4 - naSplit3
  - 5 - naQuad4

**Return Type**  
Not Applicable

**Default**  
naTile

**Examples**  
`app.ArrangeWindow = naTile`  
`app.ArrangeWindow = naTile` *Write*

**C++ Syntax**  
`HRESULT put_ArrangeWindows(tagNAWindowModes newVal)`

**Interface**  
IApplication
**AttenuatorMode Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the mode of operation of the attenuator control for the specified port number. This command is automatically set to Manual when an Attenuator value is set.</th>
</tr>
</thead>
</table>

**VB Syntax**

```vbnet
object.AttenuatorMode(portNum) = value
```

**Variable** *(Type) - Description*

- **object** *(Channel (object))
  - or
  - **CalSet** *(object) - Read-only property

- **portNum** *(long) - Port number of attenuator control to be changed.

- **value** *(enum NAModes) - Choose from:
  - 0 - **naAuto** - Attenuator control set to automatic. The analyzer will set the attenuator control appropriately to deliver the specified power at the source.

**Return Type**

NAModes

**Default**

0 - Auto

**Examples**

```vbnet
chan.AttenuatorMode(1) = naAuto 'Write
attn = chan.AttenuatorMode(1) 'Read
```

**C++ Syntax**

```csharp
HRESULT get_AttenuatorMode(long port, tagNAModes* pVal)
HRESULT put_AttenuatorMode(long port, tagNAModes newVal)
```

**Interface**

IChannel
ICalSet3
Attenuator Property

Description
Sets or returns the value of the source attenuator for the specified port number. Sending this command automatically sets AttenuatorMode to Manual.

VB Syntax
object.Attenuator(portNum) = value

Variable
(Type) - Description
object Channel (object)
or
CalSet (object) - Read-only property

portNum (long integer) - Port number of source attenuator to be changed.

value (double) - Attenuation value. The range of settable values depends on the PNA model. To determine the valid settings, do one of the following:

- See PNA models and options to see the range and step size for each model / option.
- To determine the maximum attenuator value use MaximumSourceStepAttenuator.

If an invalid attenuation setting is entered, the PNA will select the next lower valid value. For example, if 19 is entered, then for an E8361A, 10 dB attenuation will be selected.

Return Type
Double

Default
20 dB

Examples
chan.Attenuator(1) = 20 'Write

attn = chan.Attenuator(cnum) 'Read

C++ Syntax
HRESULT get_Attenuator(long port, double *pVal)
HRESULT put_Attenuator(long port, double newVal)

Interface
IChannel
ICalSet3

Last Modified:
28-Mar-2011 Fixed typo
25-Oct-2007 Edit value text
30-Apr-2007 Minor edits
# AutoDetection Property

**Description**  
Choose to automatically or manually set pulse mode (Narrowband or Wideband) for the channel.

**VB Syntax**  
```
pulseMeas.AutoDetection = bool
```

**Variable** *(Type)* - Description

- **pulseMeas**  
  A [PulseMeasurementControl](#) *(object)*

- **bool**  
  - **False** - Manually set the pulse mode. Use [WideBandDectionState](#) to set the pulse mode.
  - **True** - Automatically set the pulse mode.

**Return Type**  
Boolean

**Default**  
True

**Examples**  
```
pulse.AutoDetection = True 'Write
value = pulse.AutoDetection 'Read
```

**C++ Syntax**  
```
HRESULT get_AutoDetection(VARIANT_BOOL *pVal);
HRESULT put_AutoDetection(VARIANT_BOOL newVal);
```

**Interface**  
IPulseMeasurementControl

---

**Last Modified:**  
11-Mar-2010   New topic
**AutoIFBandWidth Property - Superseded**

**Description**
This command is replaced by: [AutoCWSweepTime Property](#).
In Wideband pulse mode, choose to set the IF bandwidth automatically or manually.

**VB Syntax**

```vb
pulseMeas.AutoIFBandWidth = bool
```

**Variable (Type) - Description**

- **pulseMeas** A [PulseMeasurementControl](#) (object)
- **bool**
  - **False** - Manually set the IFBW for the measurement.
  - **True** - Automatically set the IFBW for the measurement.

**Return Type**
Boolean

**Default**
True

**Examples**

```vbnet
pulse.AutoIFBandWidth = True 'Write
value = pulse.AutoIFBandWidth 'Read
```

**C++ Syntax**

```cpp
HRESULT get_AutoIFBandWidth(VARIANT_BOOL *pVal);
HRESULT put_AutoIFBandWidth(VARIANT_BOOL newVal);
```

**Interface**
IPulseMeasurementControl

---

**Last Modified:**

- May 9, 2011  Superseded
- 11-Mar-2010  New topic
### AutoIFBWAjustment Property

**Description**  
Set and read auto IFBW adjustment ON | OFF state for Gain Compression measurements.

**VB Syntax**  
\[ gca.AutoIFBWAjustment = value \]

**Variable (Type) - Description**

- **gca**  
  A GainCompression (object)

- **value**  
  (Boolean) - Auto IFBW adjustment state. Choose from:
  - False - Sets auto IFBW adjustment OFF
  - True - Sets auto IFBW adjustment ON

**Return Type**  
Boolean

**Default**  
ON

**Examples**

- `gca.AutoIFBWAjustment = True`  
  Write

- `aifbw = gca.AutoIFBWAjustment`  
  Read

**C++ Syntax**

- HRESULT get_AutoIFBWAjustment(VARIANT_BOOL* bState)
- HRESULT put_AutoIFBWAjustment(VARIANT_BOOL bState)

**Interface**  
IGainCompression

---

**Last Modified:**

8-Nov-2007  
MX New topic
AutoIFGain Property

Type topic text here.
### AutoOptimizePRF Property

**Description**
In Narrowband pulse mode, choose to set the Pulse Repetition Frequency automatically or manually. This is labeled "Optimize Pulse Frequency" on the user-interface. To make changes manually, use `MasterFrequency Property` or `MasterPeriod Property`.

**VB Syntax**

```vbnet
pulseMeas.AutoOptimizePRF = bool
```

**Variable**

- **(Type)**: Description
- **pulseMeas**
  - A `PulseMeasurementControl (object)`
  - `bool`: `False` - Manually set the PRF for the measurement.
  - `True` - Automatically set the PRF for the measurement.

**Return Type**

Boolean

**Default**

True

**Examples**

```vbnet
pulse.AutoOptimizePRF = True 'Write
value = pulse.AutoOptimizePRF 'Read
```

**C++ Syntax**

```cpp
HRESULT get_AutoOptimizePRF(VARIANT_BOOL *pVal);
HRESULT put_AutoOptimizePRF(VARIANT_BOOL newVal);
```

**Interface**

`IPulseMeasurementControl`

---

Last Modified:

11-Mar-2010   New topic
**AutoOrient Property**

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Sets ECAL module automatic orientation ON or OFF.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>obj.AutoOrient = bool</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type) - Description</em></td>
</tr>
<tr>
<td><code>obj</code></td>
<td><code>SMCTYPE</code> (object)</td>
</tr>
<tr>
<td></td>
<td>or</td>
</tr>
<tr>
<td></td>
<td><code>VMCTYPE</code> (object)</td>
</tr>
<tr>
<td><code>bool</code></td>
<td><em>(Boolean)</em></td>
</tr>
<tr>
<td><code>True</code></td>
<td>- Set AutoOrientation ON</td>
</tr>
<tr>
<td><code>False</code></td>
<td>- Set AutoOrientation OFF</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Boolean</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>True</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>Smc.AutoOrient = True</code></td>
</tr>
</tbody>
</table>

**C++ Syntax**

```cpp
HRESULT put_AutoOrient(VARIANT_BOOL bAutoOrient);
HRESULT get_AutoOrient(VARIANT_BOOL *bAutoOrient);
```

**Interface**

- `SMCTYPE`
- `VMCTYPE`
AutoOrientTunerTuner Property

Description
Sets the state of auto orientation for a noise tuner during Noise Figure for NFX.

VB Syntax
nfCal.AutoOrientTuner = bool

Variable (Type) - Description

nfCal A NoiseCal (object)

bool (Boolean)
True - Set AutoOrientTuneration ON
False - Set AutoOrientTuneration OFF

Return Type
Boolean

Default
True

Examples
nfCal.AutoOrientTuner = True

C++ Syntax
HRESULT put_AutoOrientTuner(VARIANT_BOOL bEnable);
HRESULT get_AutoOrientTuner(VARIANT_BOOL *bEnable);

Interface
INoiseCal2

Last Modified:
27-Oct-2009   MX New topic
**AutoPortExtConfig Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets the frequency span that is used to calculate Automatic Port Extension. <a href="#">Learn more about calculating Automatic Port Extension.</a></th>
</tr>
</thead>
</table>

**VB Syntax**

```
fixture.AutoPortExtConfig = value
```

**Variable (Type) - Description**

- **fixture**  
  A Fixturing (object)

- **value**  
  (ENUM as NAAutoPortExtConfig)
  - 0 naAPEC_CSPN - Use current span.
  - 1 naAPEC_AMKR - Use active marker frequency.
  - 2 naAPEC_USPN - Use custom user span. Use [AutoPortExtSearchStart Property](#) and [AutoPortExtSearchStop Property](#) to specify start and stop frequency.

**Return Type**

ENUM

**Default**

0 naAPEC_CSPN

**Examples**

```
fixture.AutoPortExtConfig = naAPEC_AMKR
value = fixture.AutoPortExtConfig 'Read
```

**C++ Syntax**

```
HRESULT get_AutoPortExtConfig(tagNAAutoPortExtConfig *pVal);
HRESULT put_AutoPortExtConfig(tagNAAutoPortExtConfig Val);
```

**Interface**

IFixturing2
### AutoPortExtDCOffset Property

**Description**
Specifies whether or not to include DC Offset as part of automatic port extension. Learn more about [Automatic DC Offset](#). Only allowed when [AutoPortExtLoss Property](#) is set to ON.

**VB Syntax**
```vbnet
fixture.AutoPortExtDCOffset = bool
```

**Variable (Type) - Description**
- `fixture` - A Fixturing (object)
- `bool` - True - Includes DC Offset correction.
- False - Does NOT include DC Offset correction.

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
fixture.AutoPortExtDCOffset = True
value = fixture.AutoPortExtDCOffset 'Read
```

**C++ Syntax**
```cpp
HRESULT get_AutoPortExtDCOffset(VARIANT_BOOL *pState);
HRESULT put_AutoPortExtDCOffset(VARIANT_BOOL bState);
```

**Interface**
IFixturing2
AutoPortExtLoss Property

Description
Specifies whether or not to include loss correction as part of automatic port extension. Learn more about Loss Compensation in port extension.

VB Syntax
fixture.AutoPortExtLoss = bool

Variable (Type) - Description
fixture A Fixturing (object)
bool True - Includes Loss correction.
False - Does NOT include Loss correction.

Return Type
Boolean

Default
False

Examples
fixture.AutoPortExtLoss = True
value = fixture.AutoPortExtLoss 'Read

C++ Syntax
HRESULT get_AutoPortExtLoss(VARIANT_BOOL *pState);
HRESULT put_AutoPortExtLoss(VARIANT_BOOL bState);

Interface IFixturing2
# AutoPortExtSearchStart Property

**Description**
Set the start frequency for custom user span. Only applies when `fixture.AutoPortExtConfig = 0 naAPEC_CSPN`.  
Learn more about User Span.

**VB Syntax**
`fixture.AutoPortExtSearchStart = value`

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A <code>Fixturing</code> (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Double) User span start value. Must be within the frequency range of the active channel and less than the value set by <code>AutoPortExtSearchStop Property</code></td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Start frequency of the current active channel.

**Examples**

```vbnet
fixture.AutoPortExtSearchStart = 1E9
value = fixture.AutoPortExtSearchStart 'Read
```

**C++ Syntax**

```cpp
HRESULT get_AutoPortExtSearchStart(double *pdVal);
HRESULT put_AutoPortExtSearchStart(double dVal);
```

**Interface**
`IFixturing2`
**AutoPortExtSearchStop Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Set the stop frequency for custom user span. Only applies when <code>fixture.AutoPortExtConfig = 0 naAPEC_CSPN</code>. Learn more about User Span. Only applies when <code>fixture.AutoPortExtConfig = 0 naAPEC_CSPN</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>fixture.AutoPortExtSearchStop = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>fixture</code></td>
<td>A <code>Fixturing</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double ) User span stop value. Must be within the frequency range of the active channel and greater than the value set by <code>AutoPortExtSearchStart Property</code></td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>Stop frequency of the current active channel.</td>
</tr>
<tr>
<td>Examples</td>
<td><code>fixture.AutoPortExtSearchStop = 1E9</code></td>
</tr>
<tr>
<td><code>value = fixture.AutoPortExtSearchStop</code></td>
<td><code>Read</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_AutoPortExtSearchStop(double *pdVal); HRESULT put_AutoPortExtSearchStop(double dVal);</td>
</tr>
<tr>
<td>Interface</td>
<td><code>IFixturing2</code></td>
</tr>
</tbody>
</table>
AutoPortExtState Property

**Description**
Enables and disables automatic port extensions on the specified port. All enabled ports will have their reference plane automatically adjusted after performing Automatic Port Extension.

**VB Syntax**
```vbnet
fixture.AutoPortExtState(port) = bool
```

**Variable (Type) - Description**
- **fixture** *(object)*
- **port** *(Integer)* Port number to enable or disable.
- **bool** *(Boolean)*
  - **True** - Enables Auto Port Extensions
  - **False** - Disables Auto Port Extensions

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
fixture.AutoPortExtState(1) = True
value = fixture.AutoPortExtState(2) 'Read
```

**C++ Syntax**
```cpp
HRESULT get_AutoPortExtState(short port, VARIANT_BOOL *pState);
HRESULT put_AutoPortExtState(short port, VARIANT_BOOL bVal);
```

**Interface**
IFixturing2
### AutoPulseTiming Property

**Description**

In Narrowband pulse mode, choose to set the delay and width automatically or manually. This is labeled "Autoselect Width and Delay" on the user-interface.

**VB Syntax**

```vbnet
pulseMeas.AutoPulseTiming = bool
```

**Variable (Type) - Description**

- `pulseMeas`:
  - A `PulseMeasurementControl (object)`
  - `bool`:
    - **False** - Manually set the delay and width for the measurement.
    - **True** - Automatically set the delay and width for the measurement.

**Return Type**

Boolean

**Default**

True

**Examples**

```vbnet
pulse.AutoPulseTiming = True 'Write
value = pulse.AutoPulseTiming 'Read
```

**C++ Syntax**

```c++
HRESULT get_AutoPulseTiming(VARIANT_BOOL *pVal);
HRESULT put_AutoPulseTiming(VARIANT_BOOL newVal);
```

**Interface**

IPulseMeasurementControl

---

Last Modified:

11-Mar-2010    New topic
**AutoSelectPulseGen Property**

**Description**  
In Narrowband pulse mode, choose to set the pulse generator used to drive the source modulation automatically or manually.

**VB Syntax**  
`pulseMeas.AutoSelectPulseGen = bool`

**Variable (Type) - Description**

- `pulseMeas`  
  A `PulseMeasurementControl` *(object)*
  
- `bool`  
  - **False** - Manually set source modulation drive for the measurement.
  - **True** - Automatically set source modulation drive for the measurement.

**Return Type**  
Boolean

**Default**  
True

**Examples**  
`pulse.AutoSelectPulseGen = True`  
`value = pulse.AutoSelectPulseGen`

**C++ Syntax**  
`HRESULT get_AutoSelectPulseGen(VARIANT_BOOL *pVal);`

`HRESULT put_AutoSelectPulseGen(VARIANT_BOOL newVal);`

**Interface**  
`IPulseMeasurementControl`

---

Last Modified:  
11-Mar-2010  
New topic
## AuxiliaryTriggerCount Property

**Description**
Returns the number of aux trigger input / output connector pairs in the PNA

**VB Syntax**
```vbnet
value = app.AuxiliaryTriggerCount
```

**Variable**
- **Value** *(Long Integer)* Variable to store the returned value.
  - 2 = PNA-X models
  - 1 = All other PNA models

- **app** An `Application (object)`

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```vbnet
ioConns = app.AuxiliaryTriggerCount
```

**C++ Syntax**
```c++
HRESULT AuxiliaryTriggerCount(long *count);
```

**Interface**
`IApplciation11`

---

**Last Modified:**
- 6-Apr-2009  Replaced N5242A with PNA-X
- 14-Dec-2006  MX New topic
**AuxTriggerScopeIsGlobal Property**

**Description**
Sets the Trigger OUT behavior to either Global or Channel. [Learn more about this setting.](#)

This command will cause the PNA to **Reset**.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

See the [AuxTrigger Object](#).

**VB Syntax**

```
pref.AuxTriggerScopeIsGlobal = value
```

**Variable** *(Type) - Description*

- **pref** - A Preferences *(object)*
- **value** - (Boolean) - Choose from:

  - **True** - Trigger properties apply to ALL channels (Global).
    - Default setting for E836x and PNA-L models.
    - Allows use of command to configure the external trigger properties.
    - "Per Point" trigger property is not settable. Use the channel's Point trigger setting.
  
  - **False** - External Trigger properties apply to each channel independently.
    - Default setting for PNA-X models.
    - Must use AuxTrigger commands to configure the external trigger properties. ExternalTriggerConnectionBehavior Property will NOT work.
    - "Per Point" trigger output property is set using the channel's Point trigger setting AND TriggerOutInterval Property.

**Return Type**

Boolean

**Default**

- **True** - E836xB and PNA-L models
- **False** - PNA-X models

**Examples**

```
pref.AuxTriggerScopeIsGlobal = 1 'Write
auxTrigPref = pref.AuxTriggerScopeIsGlobal 'Read
```

**C++ Syntax**

```
HRESULT get_AuxTriggerScopeIsGlobal(VARIANT_BOOL * pref);
HRESULT put_AuxTriggerScopeIsGlobal(VARIANT_BOOL pref);
```
Interface IPreferences5

Last modified:

25-Feb-2008 Clarification
Jan 3, 2007 MX New command
AvailableMeasurementClasses Property

**Description**
Returns a list of available measurement classes on the PNA.

**VB Syntax**

```
value = cap.AvailableMeasurementClasses
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value (Variant)</td>
<td>Variable to store the returned list of measurement classes.</td>
</tr>
<tr>
<td>cap A Capabilities (object)</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**

Variant

**Default**
Not Applicable

**Examples**

```
'Read all measurement classes
Set app = CreateObject("AgilentPNA835x.Application")
Set cap = app.Capabilities
meas=cap.AvailableMeasurementClasses
dim i
For i = 0 To UBound(meas)
    msg = msg & meas(i) & vbCrLf
Next
MsgBox msg
```

**C++ Syntax**

```
HRESULT get_AvailableMeasurementClasses(Variant *value);
```

**Interface**

ICapabilities7

---

Last Modified:

- 23-May-2011  Added example
- 4-Nov-2010  MX New topic
AverageMode Property

Description  Specifies the type of averaging to perform: Point or Sweep.

VB Syntax  \textit{chan.AverageMode} = \textit{value}

Variable  \textbf{(Type)} - Description

\textit{chan}  A \texttt{Channel} (object)

\textit{value}  \textbf{(Enum as naAverageMode)} - Average Type. Choose from:

0 - \texttt{naPoint}  Averaging measurements are made on each data point before stepping to the next data point. (Not available on 'C' models).

1 - \texttt{naSweep}  Averaging measurements are made on subsequent sweeps until the required number of averaging sweeps are performed.

Return Type  Enum

Default  1 - \texttt{naSweep}

Examples  \texttt{chan.AverageMode = naSweep 'Write}

\texttt{avgType = chan.AverageMode ' Read}

C++ Syntax  HRESULT get_AverageMode(NAAverageMode * mode);

HRESULT put_AverageMode(NAAverageMode mode);

Interface  IChannel16

Last Modified:

8-Jun-2009  Not available on 'C' models.

13-Oct-2008  MX New topic
### AveragingCount Property

**Description**
Returns the number of sweeps that have been acquired and averaged into the measurements on this channel. **AveragingFactor** specifies the number of sweeps to average. **AveragingCount** indicates the progress toward that goal.

**VB Syntax**

```vbnet
value = chan.AveragingCount
```

**Variable (Type) - Description**

- **chan**
  - A Channel (object)
- **value**
  - (Long Integer) - Variable to store the returned count

**Return Type**
Long Integer

**Default**
Not Applicable

**Example**

```
avgcount = chan.AveragingCount
```

**C++ Syntax**

```
HRESULT get_AveragingCount(long* count)
```

**Interface**
IChannel
## AveragingFactor Property

### Description
Specifies the number of measurements to combine for an average. Must also turn averaging ON by setting \texttt{chan.Averaging} = 1.

### VB Syntax
\texttt{chan.AveragingFactor = value}

### Variable (Type) - Description

- \texttt{chan}  A Channel (object)
- \texttt{value} (Long Integer)  - Number of measurement sweeps to average. Choose any number between 1 and 65536 ($2^{16}$).

### Return Type
Long Integer

### Default
1

### Examples
\texttt{chan.AveragingFactor = 5  'Write}

\texttt{avgfact = chan.AveragingFactor  ' Read}

### C++ Syntax
- HRESULT \texttt{get\_AveragingFactor(long *pVal)}
- HRESULT \texttt{put\_AveragingFactor(long newVal)}

### Interface
IChannel

---

**Last Modified:**

16-Apr-2009  Updated for Point Averaging
## Averaging Property

**Description**
Turns trace averaging ON or OFF for all measurements on the channel. Averaging is only allowed on ratioed measurements; not on single input measurements.

**VB Syntax**
```
chan.Averaging = state
```

**Variable**
- **Chan**
  - **chan** (Channel (object))

- **state** (boolean)
  - **False** - Turns averaging OFF
  - **True** - Turns averaging ON

**Return Type**
Boolean

**Default**
False

**Examples**
```
chan.Average = True 'Write

averg = chan.Averaging 'Read
```

**C++ Syntax**
```
HRESULT get_Averaging(BOOL *pVal)
HRESULT put_Averaging(BOOL newVal)
```

**Interface**
IChannel
# AvoidSpurs Property

Sets and returns the state of the avoid spurs feature.

## Description
Sets and returns the state of the avoid spurs feature.

## VB Syntax

```vbnet
mixer.AvoidSpurs = boolean
```

## Variable 

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>mixer</td>
<td>A Mixer (object)</td>
</tr>
<tr>
<td></td>
<td>A Converter (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>mixer</td>
<td>A Mixer (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Boolean) - State of avoid spurs feature. Choose from</td>
</tr>
<tr>
<td></td>
<td>False - Avoid spurs OFF</td>
</tr>
<tr>
<td></td>
<td>True - Avoid spurs ON</td>
</tr>
</tbody>
</table>

## Return Type
Boolean

## Default
False

## Examples

```vbnet
conv.AvoidSpurs = True 'Write
variable = conv.AvoidSpurs 'Read
```

## C++ Syntax

```cpp
HRESULT get_AvoidSpurs(Bool *bVal)
HRESULT put_AvoidSpurs(Bool newVal)
```

## Interface

IMixer3
IConverter5

---

**Last Modified:**

25-Jan-2011 Added converter object
**Background Property**

**Description**  Set and return the background color for the PNA display or hardcopy print.

**VB Syntax**  \texttt{colors.Background = value}

**Variable (Type) - Description**

- **colors**  A \texttt{ComColors} (object)
- **value**  (Long Integer) - RGB color of the Background pen. Convert the three RGB colors to an integer as follows:

\[
\text{RGB} = R + (G*2^8) + (B*2^{16})
\]

To find the three RGB values from the Display Colors dialog, click \texttt{Change Color}, then \texttt{Define Custom Color}.

**Return Type**  Long

**Default**  Display = 0,0,0 (Black)

Print = 255,255,255 (White)

**Examples**

\[
\begin{align*}
R &= 10 \\
G &= 10 \\
B &= 10 \\
\text{RGB} &= R + (G*2^8) + (B*2^{16}) \\
\text{colors.Background} &= \text{RGB} \quad \text{'Write} \\
\text{color} &= \text{colors.Background} \quad \text{'Read}
\end{align*}
\]

**C++ Syntax**

\[
\text{HRESULT get_Background(long* pVal);} \\
\text{HRESULT put_Background(long newVal);} \\
\]

**Interface**  IComColors

---

Last Modified:

7-Aug-2009  MX New topic
**BackOff Property**

**Description**  
Sets and returns the backoff value used to calculate various PNOP parameters.  
Also set PinOffset Property.

A sweep must be executed (single or continuous) and  
SearchPowerNormalOperatingPoint Method must be sent before reading marker results.  
To turn off the PNOP markers, either turn them off individually or DeleteAllMarkers.

To search a User Range with the PNOP search, first activate marker 1. The user range  
used with the PNOP search only applies to marker 1 searching for the linear gain value.  
The other markers may fall outside the user range.

**VB Syntax**  
`pnop.BackOff = value`

**Variable (Type) - Description**

- **pnop** (object)
- **value** (double) - Backoff value in dB. Choose any number between -500 and 500

**Return Type**  
Double

**Default**  
0 dB

**Examples**  
`backoff = pnop.BackOff `Read

See example program

**C++ Syntax**  
`HRESULT put_BackOff(double newVal);`

`HRESULT get_BackOff(double* pNewVal)`

**Interface**  
IPNOP

---

Last Modified:

19-Feb-2010  
MX New topic
# BackOffGain Property

## Description
Returns the BackOffGain result of the PNOP marker search.

\[ \text{PBO Gain} = \text{PBO Out} - \text{PBO In} \]

## VB Syntax
\[ bOffGain = pnop.BackOffGain \]

## Variable

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>bOffGain</td>
<td>double</td>
<td>Variable to store returned value</td>
</tr>
</tbody>
</table>

\[ pnop \] A PNOP (object)

## Return Type
Double

## Default
Not applicable

## Examples
\[ bOffGain = pnop.BackOffGain \]

See example program

## C++ Syntax
\[ HRESULT get_BackOffGain(double* pNewVal) \]

## Interface
IPNOP

---

Last Modified:

19-Feb-2010  MX New topic
## BackOffPIn Property

**Description**
Returns the BackOffPIn result of the PNOP marker search.

PBO In = Marker 2 X-axis

**VB Syntax**

```vbnet
bOffPin = pnop.BackOffPIn
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>bOffPin</code></td>
<td>(double) - Variable to store returned value</td>
</tr>
<tr>
<td><code>pnop</code></td>
<td>A PNOP (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not applicable

**Examples**

```vbnet
bOffPin = pnop.BackOffPIn 'Read
See_example_program
```

**C++ Syntax**

```cpp
HRESULT get_BackOffPIn(double* pNewVal)
```

**Interface**
IPNOP

---

Last Modified:

19-Feb-2010    MX New topic
BackOffPout Property

**Description**
Returns the BackOffPout result of the PNOP marker search.

PBO Out = Marker 2 Y-axis

**VB Syntax**

```
bOffPout = pnop.BackOffPout
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>bOffPout</code></td>
<td>(double) - Variable to store returned value</td>
</tr>
<tr>
<td><code>pnop</code></td>
<td>A PNOP (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not applicable

**Examples**

```
bOffPout = pnop.BackOffPout 'Read
```

See example program

**C++ Syntax**

```
HRESULT get_BackOffPout(double* pNewVal)
```

**Interface**
IPNOP

---

Last Modified:

19-Feb-2010  MX New topic
BalancedMode Property

**Description**
Sets and returns whether the balanced transform is ON or OFF

**VB Syntax**
`balMeas.BalancedMode = value`

**Variable**
(Type) - Description

`balMeas` - A BalancedMeasurement (object)

`value` - (Boolean) - State of balanced transform. Choose from
- **False** Balanced Transform OFF
- **True** Balanced Transform ON

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
balMeas.BalancedMode = True 'Write
variable = balMeas.BalancedMode 'Read
```

**C++ Syntax**

```c++
HRESULT get_BalancedMode(VARIANT_BOOL *bVal)
HRESULT put_BalancedMode(VARIANT_BOOL newVal)
```

**Interface**
IBalancedMeasurement
BalPort1PhaseOffset Property

**Description**
Sets and returns the phase offset between the two ports that comprise Balanced port 1. `balStim.Mode` must be set to a True Stimulus mode. Applicable only with [Opt 460 - iTMSA](https://www.460.com).

**VB Syntax**
```
balStim.BalPort1PhaseOffset = value
```

**Variable** *(Type) - Description*
`balStim` A `BalancedStimulus` *(object)*

`value` *(Double) - Phase Offset in degrees. Choose a value between -360 and 360.*

**Return Type**
Double

**Default**
0

**Examples**
```
balStim.BalPort1PhaseOffset = 10 'Write
variable = balStim.BalPort1PhaseOffset 'Read
```

**C++ Syntax**
```
HRESULT get_BalPort1PhaseOffset (double *pVal)
HRESULT put_BalPort1PhaseOffset (double newVal)
```

**Interface**
`IBalancedStimulus`

---

Last Modified:
15-May-2008   MX New topic
BalPort1PowerOffset Property

Last Modified:

15-May-2008   MX New topic
**BalPort1StartPhase Property**

**Description**  
Sets and returns the start phase of a phase sweep.

**VB Syntax**  
`balStim.BalPort1StartPhase = value`

**Variable**  
(***Type*) - Description

- `balStim`  
  A [BalancedStimulus](#) (object)

- `value`  
  (Double) - Start phase in degrees. Choose a value between -360 and 360.

**Return Type**  
Double

**Default**  
0

**Examples**

- `balStim.BalPort1StartPhase = 10 'Write`
- `variable = balStim.BalPort1StartPhase 'Read`

**C++ Syntax**

- `HRESULT get_BalPort1StartPhase (double *pVal)`
- `HRESULT put_BalPort1StartPhase (double newVal)`

**Interface**  
IBalancedStimulus2

---

Last Modified:

3-Mar-2009  
MX New topic (A.08.50)
# BalPort1StopPhase Property

**Description**
Sets and returns the stop phase of a phase sweep.

**VB Syntax**
```vbnet
balStim.BalPort1StopPhase = value
```

**Variable**
- **balStim**: A `BalancedStimulus` (object)
- **value**: (Double) - Stop phase in degrees. Choose a value between -360 and 360.

**Return Type**
Double

**Default**
0

**Examples**
```vbnet
balStim.BalPort1StopPhase = 10 'Write
variable = balStim.BalPort1StopPhase 'Read
```

**C++ Syntax**
- `HRESULT get_BalPort1StopPhase (double *pVal)`
- `HRESULT put_BalPort1StopPhase (double newVal)`

**Interface**
`IBalancedStimulus2`

---

Last Modified:
3-Mar-2009   MX New topic (A.08.50)
**BalPort2PhaseOffset Property**

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Sets and returns the phase offset between the two ports that comprise Balanced port 2. <code>balStim.Mode</code> must be set to a True Stimulus mode. Applicable only with Opt 460 - iTMSA.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>balStim.BalPort2PhaseOffset = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>balStim</code></td>
<td>A <code>BalancedStimulus</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) - Phase Offset in degrees. Choose a value between -360 and 360.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Double</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>0</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>balStim.BalPort2PhaseOffset = 10 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>variable = balStim.BalPort2PhaseOffset 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_BalPort2PhaseOffset (double *pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_BalPort2PhaseOffset (double newVal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IBalancedStimulus</td>
</tr>
</tbody>
</table>

Last Modified:

15-May-2008   MX New topic
Write/Read

BalPort2PowerOffset Property

Description
Sets and returns the power offset between the two ports that comprise Balanced port 2. `balStim.Mode` must be set to a True Stimulus mode. Applicable only with Opt 460 - iTMSA.

VB Syntax

```
balStim.BalPort2PowerOffset = value
```

Variable (Type) - Description

`balStim` A `BalancedStimulus` (object)

`value` (Double) - Power Offset in dB. Choose a value between .

Return Type
Double

Default
0

Examples
```
balStim.BalPort2PowerOffset = 2 'Write
variable = balStim.BalPort2PowerOffset 'Read
```

C++ Syntax

```
HRESULT get_BalPort2PowerOffset (double *pVal)
HRESULT put_BalPort2PowerOffset (double newVal)
```

Interface
`IBalancedStimulus`

Last Modified:
15-May-2008  MX New topic
**BalPort2StartPhase Property**

**Description**
Sets and returns the start phase of a phase sweep.

**VB Syntax**
`balStim.BalPort2StartPhase = value`

**Variable (Type) - Description**
- `balStim` A `BalancedStimulus` (object)
- `value` (Double) - Start phase in degrees. Choose a value between -360 and 360.

**Return Type**
Double

**Default**
0

**Examples**
```
balStim.BalPort2StartPhase = 10 'Write
variable = balStim.BalPort2StartPhase 'Read
```

**C++ Syntax**
```
HRESULT get_BalPort2StartPhase (double *pVal)
HRESULT put_BalPort2StartPhase (double newVal)
```

**Interface**
IBalancedStimulus2

---

Last Modified:
3-Mar-2009   MX New topic (A.08.50)
**BalPort2StopPhase Property**

**Description**  
Sets and returns the stop phase of a phase sweep.

**VB Syntax**  
`balStim.BalPort2StopPhase = value`

**Variable (Type) - Description**

- `balStim`: A `BalancedStimulus (object)`
- `value`: (Double) - Stop phase in degrees. Choose a value between -360 and 360.

**Return Type**  
Double

**Default**  
0

**Examples**

- `balStim.BalPort2StopPhase = 10 'Write`
- `variable = balStim.BalPort2StopPhase 'Read`

**C++ Syntax**

- `HRESULT get_BalPort2StopPhase (double *pVal)`
- `HRESULT put_BalPort2StopPhase (double newVal)`

**Interface**  
IBalancedStimulus2

---

**Last Modified:**

3-Mar-2009    MX New topic (A.08.50)
## BandwidthTarget Property

**Description**  
Sets the insertion loss value at which the bandwidth of a filter is measured (using BandwidthTracking or SearchFilterBandwidth). For example, if you want to determine the filter bandwidth 3 db below the bandpass peak value, set BandwidthTarget to -3.

### VB Syntax

```vbnet
meas.BandwidthTarget = value
```

### Variable  
**Type** - Description

- `meas`  
  A Measurement (object)

- `value`  
  (single) - Target value. Choose any number between -500 and 500

### Return Type

Single

### Default

-3

### Examples

```vbnet
meas.BandwidthTarget = -3 'Write
```

```vbnet
fbw = meas.BandwidthTarget 'Read
```

### C++ Syntax

- HRESULT put_BandwidthTarget(float target)
- HRESULT get_BandwidthTarget(float* target)

### Interface

IMeasurement
**BandwidthTracking Property**

**Description**
Searches continually (every sweep) for the current BandwidthTarget (default is -3). To search the filter bandwidth for ONE SWEEP only (not continually), use meas.SearchFilterBandwidth.

This feature uses markers 1-4. To turn off these markers, either turn them off individually or DeleteAllMarkers.

The bandwidth statistics are displayed on the analyzer screen. To get the bandwidth statistics, use either GetFilterStatistics or FilterBW, FilterCF, FilterLoss, or FilterQ.

The analyzer screen will show either Bandwidth statistics OR Trace statistics; not both.

To restrict the search to a UserRange with the bandwidth search, first activate marker 1 and set the desired UserRange. Then send the SearchFilterBandwidth command. The user range used with bandwidth search only applies to marker 1 searching for the max value. The other markers may fall outside the user range.

**VB Syntax**

```vbnet
meas.BandwidthTracking = value
```

**Variable (Type) - Description**

- **meas**
  A Measurement (object)

- **value**
  (boolean)
  - **True** - Turns bandwidth tracking ON
  - **False** - Turns bandwidth tracking OFF

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
meas.BandwidthTracking = False 'Write
bwtrack = meas.BandwidthTracking 'Read
```

**C++ Syntax**

```cpp
HRESULT put_BandwidthTracking(VARIANT_BOOL state)
HRESULT get_BandwidthTracking(VARIANT_BOOL* state)
```

**Interface**
IMeasurement
About Balanced Measurements

**BB_BalPort1Negative Property**

**Description**
With a Balanced - Balanced topology, returns the PNA port number that is connected to the Negative side of the DUT's logical Port 1.

Use [SetBBPorts Method](#) to set the port mapping for a Balanced - Balanced topology.

**VB Syntax**

```vbnet
var = balTopology.BB_BalPort1Negative
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>var</code></td>
<td>(Long Integer) Variable to store the returned value.</td>
</tr>
</tbody>
</table>

**balTopology**
A [BalancedTopology](#) *(object)*

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**

```vbnet
variable = balTop.BB_BalPort1Negative  'Read
```

**C++ Syntax**

```cpp
HRESULT get_BB_BalPort1Negative(long *bVal)
```

**Interface**
IBalancedTopology
**BB_BalPort1Positive Property**

**Description**  
With a Balanced - Balanced topology, returns the PNA port number that is connected to the Positive side of the DUT's logical Port 1. Use [SetBBPorts Method](#) to set the port mapping for a Balanced - Balanced topology.

**VB Syntax**  
```vbnet
var = balTopology.BB_BalPort1Positive
```

**Variable**  
(Type) - Description

- var  
  (Long Integer) Variable to store the returned value.

**balTopology**  
A [BalancedTopology](#) (object)

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**  
```vbnet
variable = balTop.BB_BalPort1Positive 'Read
```

**C++ Syntax**  
```c
HRESULT get_BB_BalPort1Positive(long *bVal)
```

**Interface**  
IBalancedTopology
BB_BalPort2Negative Property

**Description**  
With a Balanced - Balanced topology, returns the PNA port number that is connected to the Negative side of the DUT’s logical Port 2.

Use [SetBBPorts Method](#) to set the port mapping for a Balanced - Balanced topology.

**VB Syntax**  
`var = balTopology.BB_BalPort2Negative`

**Variable**  
- **(Type)** - Description
  - `var` (Long Integer) Variable to store the returned value.

**balTopology**  
A [BalancedTopology](#) (object)

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**  
```
variable = balTop.BB_BalPort2Negative  'Read
```

**C++ Syntax**  
`HRESULT get_BB_BalPort2Negative(long *bVal)`

**Interface**  
IBalancedTopology
BB_BalPort2Positive Property

**Description**
With a Balanced - Balanced topology, returns the PNA port number that is connected to the Positive side of the DUT's logical Port 2.

Use **SetBBPorts Method** to set the port mapping for a Balanced - Balanced topology.

**VB Syntax**
```vbnet
var = balTopology.BB_BalPort2Positive
```

**Variable**
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>var</code></td>
<td>(Long Integer) Variable to store the returned value.</td>
</tr>
</tbody>
</table>

**balTopology**
A **BalancedTopology (object)**

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```vbnet
variable = balTop.BB_BalPort2Positive  'Read
```

**C++ Syntax**
```cpp
HRESULT get_BB_BalPort2Positive(long *bVal)
```

**Interface**
IBalancedTopology
### BBalMeasurement Property

**Description**
Sets and returns the measurement for the Balanced - Balanced topology.

**VB Syntax**

```vbnet
balMeas.BBalMeasurement = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>balMeas</code></td>
<td>A <code>BalancedMeasurement</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(String) - Balanced - Balanced Measurement parameter. Not case sensitive. Choose from:</td>
</tr>
</tbody>
</table>

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Sdd11</td>
<td>Sdd12</td>
<td>Sdc11</td>
<td>Sdc12</td>
<td></td>
</tr>
<tr>
<td>Sdd21</td>
<td>Sdd22</td>
<td>Sdc21</td>
<td>Sdc22</td>
<td></td>
</tr>
<tr>
<td>Scd11</td>
<td>Scd12</td>
<td>Scc11</td>
<td>Scc12</td>
<td></td>
</tr>
<tr>
<td>Scd21</td>
<td>Scd22</td>
<td>Scc21</td>
<td>Scc22</td>
<td></td>
</tr>
<tr>
<td>Imb1</td>
<td>Imb2</td>
<td>CMRR -(Sdd21/Scc21)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Sdd11

**Examples**

```vbnet
balMeas.BBalMeasurement = "Sdd11"  'Write
variable = balMeas.BBalMeasurement  'Read
```

**C++ Syntax**

```
HRESULT get_BBalMeasurement(BSTR *pVal)  
HRESULT put_BBalMeasurement(BSTR newVal)  
```

**Interface**
`IBalancedMeasurement`
### BeginResponse Property

**Description**
When constructing a limit line, specifies the amplitude value of the start of a limit segment.

**VB Syntax**

```vbnet
limtseg.BeginResponse = value
```

**Variable**

- **Type** - Description
  - `limtseg` A LimitSegment *(object)*
  - `value` *(double)* - Amplitude value. No units

**Return Type**

Double

**Default**

0

**Examples**

```vbnet
Set limtseg = meas.LimitTest(1)
limtseg.BeginResponse = 10 'Write
BegResp = limtseg.BeginResponse 'Read
```

**C++ Syntax**

```
HRESULT get_BeginResponse(double *pVal)
HRESULT put_BeginResponse(double newVal)
```

**Interface**

`ILimitSegment`
### BeginStimulus Property

**Description**
When constructing a limit line, specifies the beginning X-axis value.

**VB Syntax**

```vbnet
limtseg.BeginStimulus = value
```

**Variable**

- **Type**: A LimitSegment *(object)*
- **value**: *(double)* - Stimulus value. No units

**Return Type**
Double

**Default**
0

**Examples**

```vbnet
Set limtseg = meas.LimitTest(1)
limtseg.Type = naLimitSegmentType_Maximum
limtseg.BeginStimulus = 3e9
limtseg.EndStimulus = 4e9
limtseg.BeginResponse = 10
limtseg.EndResponse = 10
```

**C++ Syntax**

- HRESULT get_BeginStimulus(double *pVal)
- HRESULT put_BeginStimulus(double newVal)

**Interface**
ILimitSegment
**Read/Write**

**About Embedded LO**

## BroadbandTuningSpan Property

**Description**
Sets and returns the frequency span for the broadband tuning sweep.

**VB Syntax**

```
obj.BroadbandTuningSpan = value
```

**Variable (Type) - Description**

- **obj**
  - An [EmbeddedLO](#) (object)
  - A [ConverterEmbeddedLO](#) (object)

- **value**
  - (Double) Broadband frequency span in Hz.

**Return Type**
(Double)

**Default**
3 MHz

**Examples**

```
embedLO.BroadbandTuningSpan = 1E6 'write
```

```
value = embedLO.BroadbandTuningSpan 'read
```

**C++ Syntax**

```
HRESULT get_BroadbandTuningSpan(double* span);
HRESULT put_BroadbandTuningSpan(double span);
```

**Interface**
IEmbeddedLO

---

**Last Modified:**
- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic

840
BucketNumber Property

Description
Sets or returns the bucket number (data point) for the active marker. When the markers are interpolated (non-discrete), the returned value is the nearest marker bucket position.

VB Syntax
mark.BucketNumber = value

Variable (Type) - Description
mark A Marker (object)
value (long integer) - Data point. Choose any number between 0 and the measurement's number of data points - 1. For example, with Number of points = 201, choose between 0 and 200

Return Type
Long Integer

Default
The first marker is set to the middle of the span. Subsequent markers are set to the bucket number of the previously active marker.

Examples
mark.BucketNumber = 100 'moves the active marker to data point 100
pointNumber = mark.BucketNumber 'returns the data point number of the marker object. When the markers are interpolated (non-discrete), the returned value is the nearest marker bucket position.

C++ Syntax
HRESULT get_BucketNumber(long *pVal)
HRESULT put_BucketNumber(long newVal)

Interface
IMarker
Write/Read

C0 Property

Description
Sets and Returns the C0 (C-zero) value (the first capacitance value) for the calibration standard. To set the other capacitance values, use C1, C2, C3.

VB Syntax
\[ \text{calstd}.C0 = \text{value} \]

Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(single) - Value for C0 in femtofarads (1E-15)</td>
</tr>
</tbody>
</table>

Return Type
Single

Default
Not Applicable

Examples
\[ \text{calstd}.C0 = 15 \quad \text{Write the value of C0 to 15femtofarads} \]
\[ \text{cap0} = \text{calstd}.C0 \quad \text{Read the value of C0} \]

C++ Syntax

```c++
HRESULT get_C0(float *pVal)
HRESULT put_C0(float newVal)
```

Interface
ICalStandard
C1 Property

**Description**
Sets and Returns the C1 value (the second capacitance value) for the calibration standard. To set the other capacitance values, use **C0**, **C2**, **C3**.

**VB Syntax**
```
calstd.C1 = value
```

**Variable**

- **calstd** (A CalStandard **object**). Use calKit.GetCalStandard to get a handle to the standard.
- **value** (**single**) - Value for C1.

**Return Type**
Single

**Default**
Not Applicable

**Examples**
```
calstd.C1 = 15  'Write the value of C1.
cap1 = calstd.C1  'Read the value of C1.
```

**C++ Syntax**
```
HRESULT get_C1(float *pVal)
HRESULT put_C1(float newVal)
```

**Interface**
ICalStandard
C2 Property

Description  Sets and Returns the C2 value (the third capacitance value) for the calibration standard.
To set the other capacitance values, use C0, C1, C3.

VB Syntax  
\[
\text{calstd.C2} = \text{value}
\]

Variable  

- \text{calstd} - A CalStandard \text{(object)}. Use calKit.GetCalStandard to get a handle to the standard.
- \text{value} - (single) - Value for C2.

Return Type  Single

Default  Not Applicable

Examples  
\[
\text{calstd.C2} = 15 \quad \text{'Write the value of C2.'}
\]
\[
\text{cap2} = \text{calstd.C2} \quad \text{'Read the value of C2'}
\]

C++ Syntax  
HRESULT get_C2(float *pVal)
HRESULT put_C2(float newVal)

Interface  ICalStandard
C3 Property

Description
Sets and Returns the C3 value (the fourth capacitance value) for the calibration standard.
To set the other capacitance values, use C0, C1, C2

VB Syntax

calstd.C3 = value

Variable


(value) (single) - Value for C3.

Return Type
Single

Default
Not Applicable

Examples

calstd.C3 = 15 'Write the value of C3.

cap3 = calstd.C3 'Read the value of C3

C++ Syntax

HRESULT get_C3(float *pVal)
HRESULT put_C3(float newVal)

Interface
ICalStandard
CalFactor Property

**Description**
Sets or returns the cal factor value associated with a power sensor cal factor segment.

**VB Syntax**
`calFactSeg.CalFactor = value`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>calFactSeg</code></td>
<td>A PowerSensorCalFactorSegment (Object) or</td>
</tr>
<tr>
<td></td>
<td>A PowerSensorCalFactorSegmentPMAR (Object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) – Cal factor in percent. Choose any value between 1 and 150</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**
```
calFactSeg.CalFactor = 98 'Write
factor = calFactSeg.CalFactor 'Read
```

**C++ Syntax**
```
HRESULT put_CalFactor(Double newVal);
HRESULT get_CalFactor(Double *pVal);
```

**Interface**
IPowerSensorCalFactorSegment
IPowerSensorCalFactorSegmentPMAR
### CalibrationType Property  Superseded

**Description**

Note: This command has been replaced by `CalibrationTypeID_property`, which provides selection of Calibration Type by string.

Specifies the type of calibration to perform or apply to the active S-Parameter measurement. This command determine the ports involved in the CalType by the ports being used by the active measurement.

For example:

- If the measurement is an S23, it uses ports 2 and 3.
- If the measurement is an S22 it will use the legacy load port to figure out which two ports form the caltype. The legacy load port is set using `CreateMeasurement`.
- If `naCalType_ThreePort_SOLT` is specified on a 4-port PNA, an `E_NA_DEPRECATED_COMMAND` error is returned. There is no way to determine the intended three ports.
- If `naCalType_FourPort_SOLT` is specified on a 4-port PNA, it is obvious that the ports involved are ports 1,2,3, and 4.

Note: For FCA measurements, use `CalibrationName` and `CalibrationTypeID`.

---

**VB Syntax**

```vbnet
meas.CalibrationType = type
```

**Variable**  
**Type** - Description

- **meas** A `Measurement` (object)
- **type** (enum `NACalType`) - Calibration type. Choose from:
  - 0 - naCalType_Response_Open
  - 1 - naCalType_Response_Short
  - 2 - naCalType_Response_Thru
  - 3 - naCalType_Response_Thru_And_Isol
  - 4 - naCalType_OnePort
  - 5 - naCalType_TwoPort_SOLT
  - 6 - naCalType_TwoPortTRL
  - 7 - naCalType_None
  - 8 - naCalType_ThreePort_SOLT
  - 9 - Custom
  - 10 - naCalType_FourPort_SOLT

**Return Type**  
`NACalType`
<table>
<thead>
<tr>
<th>Default</th>
<th>naCalType_None</th>
</tr>
</thead>
<tbody>
<tr>
<td>Examples</td>
<td><code>meas.CalibrationType = naCalType_Response_Open</code> <strong>Write</strong></td>
</tr>
<tr>
<td></td>
<td><code>meascal = meas.CalibrationType</code> <strong>Read</strong></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT put_CalibrationType (tagNACalType CalType)</td>
</tr>
<tr>
<td></td>
<td>HRESULT get_CalibrationType (tagNACalType* pCalType)</td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>
**CalibrationName Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the name of the current Cal Type.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td>\texttt{value = meas.CalibrationName}</td>
</tr>
<tr>
<td>Variable (Type) - Description</td>
<td></td>
</tr>
<tr>
<td>value (string)</td>
<td>Variable to store the returned value.</td>
</tr>
<tr>
<td>meas</td>
<td>A \texttt{Measurement} (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>String</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td>\texttt{ct = meas.CalibrationName}</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_CalibrationName( BSTR* CalibrationName);</td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement2</td>
</tr>
</tbody>
</table>
Description  

Note: Beginning with Rev 6.0, this command is no longer necessary. Learn more. Because of improved calibration techniques, Both is always selected although a power meter measurement is performed only on port 1.

Specifies which SMC port to calibrate.

VB Syntax  

SMC.CalibrationPort = value

Variable  

(Type) - Description

SMC  SMCType (object)

value  (String) Port number to be calibrated. Choose from:

- 1 - SMC forward
- 2 - SMC reverse
- Both

Return Type  

String

Default  

1

Examples  

value = SMC.CalibrationPort = "Both"

C++ Syntax  

HRESULT put_CalibrationPort(BSTR port);
HRESULT get_CalibrationPort(BSTR *port);

Interface  

SMCType
VMCType
**CalibrationTypeID Property**

**Description**  
*Note: This command replaces Calibration Type Property.*  
Sets or returns the current cal type for the measurement using a Cal Type Name.  
This command is used to set the Cal Type after recalling a Cal Set. Learn more  
You can also use the CLSID or GUID associated with the Cal Type.

**VB Syntax**  
```
meas.CalibrationTypeID = id
```

**Variable**  
(Type) - Description  
meas (Object)  

(id) (String) Cal type. Case sensitive. Use one of the following:

**For Full Calibrations:**  
This command does not distinguish between TRL and SOLT. The same number of error terms is applied for both Cal Types.  
"Full n Port(x,y,z...)
where  

n = the number of ports to calibrate  
x,y,z = the port numbers to calibrate

For example:  
"Full 7 Port (2,3,4,5,6,7,8)"

**For Response Calibrations:**  
"Response(param)" OR  
"ResponseAndIsolation(param)"

Where param =

- S-parameter. For example"
  
  - "Response(S21)"
  - "ResponseAndIsolation(A/R)"

- Single or ratioed receivers using either logical receiver notation or physical receiver notation. For example:
  
  - "Response(A)"
  - "ResponseAndIsolation(a3/b4)"
For FCA Calibrations:

- "SMC_2P" (Response + Input + Output) All four sweeps required. Most accurate.
- "SMCRsp+IN" No Output match. All four sweeps required.
- "SMCRsp+OUT" No Output match. All four sweeps required.
- "SMCRsp" No Input or Output match. Saves two sweeps.

For VMC, multiple Cal types are not available.

For Gain Compression Cal
where \( r \) = receive port; \( s \) = source port

- "GCA 2P (r,s)" - full 2-port cal
- “GCA Enh Resp (r,s)” - Enhanced Response Cal

Return Type  String
Default  Not Applicable

Examples

```vbscript
Dim pna
Dim m

Set pna = CreateObject("AgilentPNA835x.Application")
Set m = pna.ActiveMeasurement
m.CalibrationTypeID = "Scalar Mixer Cal"
m.ErrorCorrection = True
MsgBox m.CalibrationName
```

C++ Syntax

```
HRESULT get_CalibrationTypeID( BSTR* CalibrationTypeID );
HRESULT put_CalibrationTypeID( BSTR CalibrationTypeID );
```

Interface  IMeasurement2

Last modified:

- 22-Sep-2009  Removed VMC strings
- 27-May-2008  Edit channel vs meas
- 11-Feb-2008  Fixed typo
- 9/12/06  MQ Added for multiport.
### CalibrationFrequencies Property

**Description**
Sets and returns the whether to perform the source power cal at the center frequencies midway between the main tones, or at all main tone frequencies.

**VB Syntax**
```vbnet
imd.CalibrationFrequencies = value
```

**Variable**
- **Type**
  - `imd` A `SweptIMD` (object)
  - `value` (Enum as NAIMDCalibrationFrequencies) Choose from:
    - **0 - naIMDCenterFrequencies** - Perform source power calibration at only the center frequencies midway between the main tones.
    - **1 - naIMDALLFrequencies** - Perform source power calibration at all main tone frequencies.

**Return Type**
Enum

**Default**
0 - naIMDCenterFrequencies

**Examples**
```vbnet
imd.CalibrationFrequencies = naIMDALLFrequencies 'Write

calFreq = imd.CalibrationFrequencies 'Read
```

**C++ Syntax**
```csharp
HRESULT get_CalibrationFrequencies(tagNAIMDCalibrationFrequencies * Val)
HRESULT put_CalibrationFrequencies(tagNAIMDCalibrationFrequencies newVal)
```

**Interface**
ISweptIMD

---

**Last Modified:**
- 9-Sep-2008  MX New topic
CalKitType Property

**Description**
Sets and returns a calibration kit type for calibration or to be used for kit modification. To get a handle to this kit, use app.ActiveCalKit.

There is also a CalKitType property for use during a Guided, SMC, and VMC Calibration.

**VB Syntax**

```
object .CalKitType = value
```

**Variable**

*(Type) - Description*

- **object**
  - calkit (object) or Application (object)

  **Note:** app.CalKitType and calkit.calKitType perform exactly the same function.

- **value** *(enum naCalKit)*
  - Calibration Kit type. Choose from:
    1 - naCalKit_User1
    2 - naCalKit_User2
    3 - naCalKit_User3
    4 - naCalKit_User4
    ...
    49 - naCalKit_User49
    50 - naCalKit_User50

  These enumerated values correspond with the calibration kit ID on the Advanced Cal Kit Modify dialog box.

  To change the cal kit name, use Name property.

**Return Type**

NACalKit

**Default**
Not Applicable

**Examples**

```
calkit.CalKitType = naCalKit_User27

kitype = app.CalKitType
```

**C++ Syntax**

```
HRESULT get_CalKitType(tagNACalKit *pVal);
HRESULT put_CalKitType(tagNACalKit newVal);
```

**Interface**

IApplication
ICalKit
CalKitType Property

Description
Sets and returns the ECal or mechanical cal kit for the specified port number to be used during the calibration.

There is also a CalKitType Property for use during an Unguided Cal.

Note: Sliding loads are not fully supported from the GuidedCalibration object. The Measure button must be pressed manually on the PNA.

Note: This command replaces Do1PortEcal Property and Do2PortEcal Property for SMC and VMC Calibrations.

VB Syntax
object.CalKitType (port) = value

Variable (Type) - Description
object Any of the following:
GuidedCalibration (object)
SMCType (object)
VMCTYPE (object)

port (Long) Port number to which the cal kit will be assigned.
For Guided Cals and SMC, select port number.
For VMC calibrations:

- 1 - Mixer Input.
- Any unused port can be used for the mixer output.
- Output port of MUT +1 - Output port of the calibration mixer. Generally this is port 3.

value (string) - Calibration Kit type. Case-sensitive.
Use GetCompatibleCalKits to return a list of valid Cal Kits.

Return Type String

Default Not Applicable

Examples
'Note: All of the following specify port 1 only
' Mechanical Cal Kit
guidedCal.CalKitType(1) = "85052C"
' Standard ECal modules
guidedCal.CalKitType(1) = "N4691-60004 ECal"
' Non-factory ECal characterizations are specified as follows:
guidedCal.CalKitType(1) = "N4691-60004 User 1 ECal"
When two or more ECAL modules with the same model number are connected, also specify the serial number as follows:

\[ \text{guidedCal.CalKitType(1) = "N4691-60004 ECAL 01234"} \]

When Disk Memory ECAL user characterizations are used, specify both the User char and the serial number as follows:

\[ \text{guidedCal.CalKitType(1) = "N4691-60004 MyDskChar ECAL 01234"} \]

Turn on auto orientation for the ECAL (default behavior).

\[ \text{value = smc.CalKitType(1)} \]

**C++ Syntax**

`HRESULT get_CalKitType( long port, BSTR *calkit)`

`HRESULT put_CalKitType( long port, BSTR calkit)`

**Interface**

`IGuidedCalibration`  
`SMCType`  
`VMCType`  

**Last Modified:**

- 6-Apr-2011  Edited Ecal Examples
- 17-Feb-2011  Replaces Do1 and 2portECal
- 13-Aug-2007  Added detail for port argument
## CalKitType Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the name of the Cal Kit to use for unguided cal.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>cal.CalKitType (port) = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>cal</code></td>
<td>Calibrator (object)</td>
</tr>
<tr>
<td><code>port</code></td>
<td>(Long) Currently unused</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(String) Cal Kit name enclosed in quotes. Use <a href="#">CalKitTypes</a> to read a list of all available Cal Kits in the PNA.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>String</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Last kit selected</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>cal.CalKitType(1) = &quot;85052B&quot;</code></td>
</tr>
</tbody>
</table>
| **C++ Syntax** | HRESULT put_CalKitType(long port, BSTR calKit);  
HRESULT get_CalKitType(long port, BSTR* pCalKit); |
| **Interface** | ICalibrator10 |

Last Modified:  
17-Mar-2010 MX New topic
**Write/Read**

**CalKitTypes Property**

**Description**

**VB Syntax**  
`cal.CalKitTypes (port) = value`

**Variable**  
*(Type) - Description*

- `cal`  
  *Calibrator* (object)

- `port`  

- `value`  
  *(String)*

**Return Type**  
Long

**Default**  

**Examples**  
`cal.CalKitTypes(4) =`

**C++ Syntax**  
HR\RESULT put_CalKitTypes(long port, BSTR calKit);
HR\RESULT get_CalKitTypes(long port, BSTR* pCalKit);

**Interface**  
ICalibrator10

---

Last Modified:  
17-Mar-2010   MX New topic
# CalMethod Property

**Description**  
Sets and returns the method for performing calibration on a noise channel.

**VB Syntax**  

```vbnet
noise.CalMethod = value
```

**Variable**  

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>noise</td>
<td>NoiseCal (object)</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>string</td>
<td>Cal Method. Choose from:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- &quot;VectorFull&quot; or &quot;Vector&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- &quot;SParameter&quot; (Not available for NFX measurements)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- &quot;ScalarFull&quot; or &quot;Scalar&quot;</td>
</tr>
</tbody>
</table>

**Return Type**  
String

**Default**  
"VectorFull"

**Examples**  

```vbnet
noise.CalMethod = "VectorFull" 'Write
```

```vbnet
calMethod = noise.CalMethod 'Read
```

**C++ Syntax**  

```csharp
HRESULT get_CalMethod(BSTR* pValue)
HRESULT put_CalMethod(BSTR pNewValue)
```

**Interface**  
INoiseCal

---

**Last Modified:**

- 27-Oct-2009  Updated for NFX
- 29-May-2007  MN New topic
### CalMethod Property

**Description**
Sets and returns the method by which the match-correction portion of an IMD calibration is performed.  [Learn more.](#)

**VB Syntax**
```
imd.CalMethod = value
```

**Variable**  
* Type: A *SweptIMDCal* (object)  
* Value (Enum as *NAIMDCalMethod*)  
  - Choose from:
    - 0 - *naIMDMatchCorrectedResponse* - Performs a full 2-port cal for full match-correction.
    - 1 - *naIMDResponseOnly* - Performs only a response cal instead of a full 2-port cal.

**Return Type**
Enum

**Default**
0 - *naIMDMatchCorrectedResponse*

**Examples**
```
imd.CalMethod = naIMDMatchCorrectedResponse 'Write

calmeth = imd.CalMethod 'Read
```

**C++ Syntax**
```
HRESULT get_CalMethod(tagNAIMDCalMethod * Val)
HRESULT put_CalMethod(tagNAIMDCalMethod newVal)
```

**Interface**
*ISweptIMD*

---

Last Modified: 18-Sep-2008  
MX New topic
Description
This command is replaced by SetCallInfoEx Method.
Specifies the power level that is expected at the desired reference plane (DUT input or output). This is not used for segment sweep with independent power levels or power sweeps.

VB Syntax
value = powerCalibrator.CalPower (chan, sourcePort)

Variable
(Type) - Description
value (double) - Variable to store the returned Cal power value in dBm.
powerCalibrator (object) - A SourcePowerCalibrator object
chan (long integer) - Channel number of the PNA.
sourcePort (long integer) - Source port number.
Use GetPortNumber to return the port number of a source that only has a string name, such as an External Source.

Return Type
None
Default
0

Examples
Set powerCalibrator = pna.SourcePowerCalibrator
powerCalibrator.CalPower = -10 'Write

power = powerCalibrator.CalPower 'Read

C++ Syntax
HRESULT get_CalPower(long channel, long sourcePort, double *pVal);

Interface
ISourcePowerCalibrator

Last Modified:
30-Apr-2007 Superseded
**Write/Read**

**Center Property**

**Description**
Sets or returns the Center time of either Gating or Time Domain transform windows

**VB Syntax**
```
object.Center = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>(object) As Gating or (object) As Transform</td>
</tr>
</tbody>
</table>

**value**

- (double) - Center time in seconds. Choose any number between:
  - ± (points-1) / frequency span

**Return Type**
Double

**Default**
0

**Examples**
```
trans.Center = 4.5e-9 'sets the Center time of a transform window
Write
gate.Center = 4.5e-9 'sets the Center time of a gating window - Write

cnt = trans.Center 'Read
```

**C++ Syntax**
```
HRESULT get_Center(double *pVal)
HRESULT put_Center(double newVal)
```

**Interface**
ITransform
IGating
Center Property

Description Returns the stimulus value of the center data point for the measurement. This function does NOT work for segment sweep measurements. To understand how this property is useful, see IMeasurement2 Interface.

VB Syntax

\[
\text{value} = \text{meas.Center}
\]

Variable (Type) - Description

- **value** (Double) - Variable to store the returned value.
- **meas** A Measurement (object)

Return Type Double

Default Not Applicable

Examples

\[
\text{Print meas.Center 'prints the center data point}
\]

C++ Syntax

HRESULT get_Center(double * Val);

Interface IMeasurement2
Write/Read

CenterFrequency Property

Description
Sets or returns the center frequency of the channel or
Sets or returns the center frequency of the segment.
See the Measurement2 Interface to learn how this method differs from meas.Center.

VB Syntax
object.centerFrequency = value

Variable
(Type) - Description
object  A Channel (object)
or
A Segment (object)

value (double) - Center frequency in Hertz. Choose any number between the minimum and maximum frequencies of the analyzer.

Return Type
Double

Default
Center of the frequency range

Examples
chan.centerFrequency = 4.5e9 'sets the center frequency of a linear sweep for the channel object -Write

centfreq = chan.centerFrequency 'Read

C++ Syntax
HRESULT get_CenterFrequency(double *pVal)
HRESULT put_CenterFrequency(double newVal)

Interface
IChannel
ISegment
**ChannelNumber Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the Channel number of the Channel or Measurement object.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>object.ChannelNumber</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>object</code></td>
<td>A Channel <em>(object)</em></td>
</tr>
<tr>
<td>or</td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Long Integer</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>chanNum = chan.ChannelNumber</code> 'returns the channel number'</td>
</tr>
<tr>
<td></td>
<td><code>chanNum = meas.ChannelNumber</code> 'returns the channel number of the measurement'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_ChannelNumber(long *pVal)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IChannel</td>
</tr>
<tr>
<td></td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>
CharacterizationNumber Property

**Description**
Sets and reads the number to which the user characterization will be stored in the ECaL module. The number must be set before sending Initialize or the default value (1) will be used.

**VB Syntax**
```vbnet
userChar.CharacterizationNumber = value
```

**Variable**
<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>userChar</td>
<td>An ECalUserCharacterizer Object</td>
</tr>
<tr>
<td>value</td>
<td>(Long) User Characterization number. Choose a value between 1 and 12.</td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```vbnet
userChar.CharacterizationNumber = 5
```

**C++ Syntax**
```c++
HRESULT put_CharacterizationNumber(long *Number);
```

**Interface**
IECalUserCharacterizer

---

Last Modified:

2-Nov-2008   New topic (8.33)
CharacterizeMixerOnly Property

Description
Sets and returns whether to perform ONLY a mixer characterization.

VB Syntax
`VMC.CharacterizeMixerOnly = bool`

Variable (Type) - Description
`VMC` (object) `bool` (Boolean)

- **True** - Perform ONLY mixer characterization.
- **False** - Perform both mixer characterization and calibration.

Return Type
Boolean

Default
False

Examples
`value = VMC.CharacterizeMixerOnly`

C++ Syntax
`HRESULT put_CharacterizeMixerOnly(VARIANT_BOOL bCharMixerOnly);`
`HRESULT get_CharacterizeMixerOnly(VARIANT_BOOL *bCharMixerOnly);`

Interface
`VMCType`
CharFileName Property

**Description**
Specifies the mixer characterization (.S2P) file and immediately loads the file. Also specify the use of a characterization file with LoadCharFromFile Property.

**VB Syntax**

```
VMC.CharFileName = value
```

**Variable**

- **(Type)** - Description
  - **VMC**
    - VMCType (object)
  - **value** (String) Full path, file name, and extension of the mixer characterization file.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```
VMC.CharFileName = "C:/Program Files/Agilent/Network Analyzer/Documents/default.S2P"
```

**C++ Syntax**

```
HRESULT put_CharFileName(BSTR filename);
HRESULT get_CharFileName(BSTR *filename);
```

**Interface**
VMCType
**CharMixerReverse Property**

**Description**
Specifies the direction in which to characterize the calibration mixer. [Learn more about the calibration mixer.](#)

**VB Syntax**

```
VMC.CharMixerReverse = bool
```

**Variable**

*(Type)* - Description

- **VMC**
  VMCTYPE (object)

- **bool** (Boolean)
  0 - Characterize the calibration mixer in the SAME direction as that specified in the mixer setup.
  1 - Characterize the calibration mixer in the REVERSE direction as that specified in the mixer setup.

**Return Type**
Boolean

**Default**
0

**Examples**

```
VMC.CharMixerReverse = 0
```

**C++ Syntax**

```
HRESULT put_CharMixerReverse(VARIANT_BOOL bcharReverse);
HRESULT get_CharMixerReverse(VARIANT_BOOL *bcharReverse);
```

**Interface**
VMCTYPE2
CitiContents Property - Superseded

Description  This command is replaced with SaveData Method

Specifies the contents of subsequent citifile saves using app. SaveCitiDataData or app. SaveCitiFormattedData

VB Syntax  \texttt{pref.CitiContents = value}

Variable (Type) - Description

\texttt{pref}  A Preferences (object)

\texttt{value}  (string) - Contents that will be saved with subsequent save commands. Choose from:

"Single" - Single trace
"Displayed" - All displayed traces
"Auto" - All displayed traces

Return Type  String

Default  "Auto"

Examples  \texttt{pref.CitiContents = "Single" 'Write}

\texttt{content = pref.CitiContents 'Read}

C++ Syntax  HRESULT get_CitiContents(BSTR *Contents)
            HRESULT put_CitiContents(BSTR Contents)

Interface  IPreferences
CitiFormat Property - Superseded

Description

This command is replaced with SaveData Method

Specifies the format of subsequent citifile saves using app.SaveCitiFormattedData

VB Syntax

`pref.CitiFormat = value`

Variable

(Type) - Description

`pref` A `Preferences` (object)

`value` (string) - Format in which the citifile will be saved with subsequent save commands. Choose from:

"MA" - Linear Magnitude / degrees

"DB" - Log Mag / degrees

"RI" - Real / Imaginary

"Auto" - Format in which the trace is already displayed. If other than Log Mag, Linear Magnitude, or Real/Imag, then the format will be in Real/Imag.

Return Type

String

Default

"Auto"

Examples

`pref.CitiFormat = "MA"` "Write"

`format = pref.CitiFormat` "Read"

C++ Syntax

HRESULT get_CitiFormat(BSTR *Format)

HRESULT put_CitiFormat(BSTR Format)

Interface

IPreferences
CmnModeZConvPortImag Property

Description
Sets the imaginary part of the impedance value for the common port impedance conversion function.

VB Syntax
```
fixture.CmnModeZConvPortImag(portNum) = value
```

Variable (Type) - Description

- **fixture**: A Fixturing (object)
- **portNum**: (Integer) Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.
- **value**: (Double) Imaginary part of the Impedance value. Choose a value between 0 and 1E18.

Return Type
Double

Default
0

Examples
```
fixture.CmnModeZConvPortImag(2) = 75 'Write
value = fixture.CmnModeZConvPortImag(1) 'Read
```

C++ Syntax
```
HRESULT get_CmnModeZConvPortImag( short portNum, double *pVal)
HRESULT put_CmnModeZConvPortImag( short portNum, double newVal)
```

Interface
IFixturing2
## CmnModeZConvPortReal Property

### Description
Sets the real part of the impedance value for the common port impedance conversion function.

### VB Syntax
```
fixture.CmnModeZConvPortReal(portNum) = value
```

### Variable (Type) - Description
- **fixture** (Type):  
  A Fixturing (object)

- **portNum** (Type):  
  Balanced (logical) port number. Choose from logical ports 1, 2, or 3. [Learn more about logical ports.](#)

- **value** (Type):  
  Real part of the Impedance value. Choose a value between 0 and 1E18.

### Return Type
Double

### Default
See [Common Mode Port Z Conversion Default](#)

### Examples
```
fixture.CmnModeZConvPortReal(2) = 75 'Write
value = fixture.CmnModeZConvPortReal(1) 'Read
```

### C++ Syntax
```
HRESULT get_CmnModeZConvPortImag( short portNum, double *pVal)
HRESULT put_CmnModeZConvPortImag( short portNum, double newVal)
```

### Interface
IFixturing2
### CmnModeZConvPortZ0 Property

**Description**
Sets the impedance value for the common port impedance conversion function. Set either this single value or set the real and imaginary parts separately. The imaginary part is set to 0.0 using this command.

**VB Syntax**

```vbnet
fixture.CmnModeZConvPortZ0(portNum) = value
```

**Variable**

- **fixture** *(Type): Fixturing (object)*
- **portNum** *(Integer): Balanced (logical) port number. Choose from logical ports 1, 2, or 3. [Learn more about logical ports.]*
- **value** *(Double): Impedance value. Choose a value between 0 and 1E7.>*

**Return Type**
Double

**Default**
See [Common Mode Port Z Conversion Default](#)

**Examples**

```vbnet
fixture.CmnModeZConvPortZ0(2) = 75 'Write
value = fixture.CmnModeZConvPortZ0(1) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_CmnModeZConvPortImag( short portNum, double *pVal)
HRESULT put_CmnModeZConvPortImag( short portNum, double newVal)
```

**Interface**
IFixturing2
CmnModeZConvState Property

**Description**
Turns ON or OFF 4-port common port impedance conversion function. Must also set the fixture simulator function to ON using FixturingState Property.

**VB Syntax**
```
fixture.CmnModeZConvState = value
```

**Variable (Type) - Description**
- `fixture` A Fixturing (object)
- `value` (Boolean)
  - **False** - Turns common port impedance conversion OFF
  - **True** - Turns common port impedance conversion ON

**Return Type**
Boolean

**Default**
False

**Examples**
```
fault.CmnModeZConvState = False 'Write

value = fault.CmnModeZConvState 'Read
```

**C++ Syntax**
```
HRESULT get_CmnModeZConvState( VARIANT_BOOL *pVal)
HRESULT put_CmnModeZConvState( VARIANT_BOOL newVal)
```

**Interface**
IFixturing2
**CompatibleCalKits Property**

**Description**
Returns a comma-separated list of valid kits that use the specified connector type. This includes mechanical cal kits, applicable characterizations found within ECal modules currently connected to the PNA, and all user characterizations stored in PNA disk memory.

**Note:** Beginning with PNA Rev 9.1, the serial number is returned for ALL ECal modules that are connected with the connector type of the specified port. Previously, the returned list would include the serial numbers to distinguish the ECal modules only when two or more identical ECal models were connected to the PNA.

**VB Syntax**
\[
\text{value} = \text{obj}.\text{CompatibleCalKits}\ (\text{port})
\]

**Variable (Type) - Description**
- **value** (Variant) Variable to store the returned list of Cal Kits.
- **obj** Any of the following:
  - GuidedCalibration (object) Superseded Replaced with GetCompatibleCalKits Method
  - SMCTYPE (object)
  - VMCTYPE (object)
- **port** (Long) Port number for which you want compatible kits.
  First set the **ConnectorType** for the port.

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
```vbnet
Dim kits As Variant
kits = MySMC.CompatibleCalKits(1)
```

**C++ Syntax**
```
HRESULT get_CompatibleCalKits(long port, VARIANT* Kits);
```

**Interface**
IGuidedCalibration
- SMCTYPE
- VMCTYPE

---

**Last Modified:**
- 16-Nov-2009 Added Note (9.1)
- 16-Nov-2009 Superseded
CompositeNormalizationMode Property

Description
Sets and returns the method by which CTB and CSO calculations are performed.

VB Syntax
```vbnet
imd.CompositeNormalizationMode = value
```

Variable (Type) - Description
- **imd**: A SweptIMD Object
- **value**: (Enum)
  - 0 - **naNone**: the normalized power is not used in calculation
  - 1 - **naNumberOfCarriers**: CTB and CSO is corrected by subtracting 10*log(N/2), where
    - N = # of carriers for CTB
    - N = # of distortion products for CSO
  - 2 - **naPdBm**: the composited normalized power for CTB or CSO is treated as a dBm value
  - 3 - **naPdBmV**: the composited normalized power for CTB or CSO is treated as a dBmV value.

Note: Power values are stored using the currently-set units. Therefore, first set units with this command, then set power values using:
- CompositeNormalizedCSOPower
- CompositeNormalizedCTBPower

Return Type
Enum

Default
**naNumberOfCarriers**

Examples
```vbnet
imd.CompositeNormalizationMode = naNone 'Write
value = imd.CompositeNormalizationMode 'Read
```

C++ Syntax
```cpp
HRESULT get_CompositeNormalizationMode(tagNAIMDCompositeNormalizationMode *pVal)
HRESULT put_CompositeNormalizationMode(tagNAIMDCompositeNormalizationMode pVal)
```

Interface
ISweptIMD

Last Modified:
16-Sep-2008  MX New topic
CompositeNormalizedCSOPower Property

Sets and returns the CSO Power for POWER normalization mode. Valid only with measurement parameters: CSO2Lo and CSO2Hi and for Normalization Modes dBm and dBmV.

VB Syntax

\[ \text{imd.CompositeNormalizedCSOPower} = \text{value} \]

Variable (Type) - Description

*imd* A SweptIMD Object

*value* (Double) Power level. The units are determined by CompositeNormalizationMode Property, which must be set first.

Return Type

Double

Default

0

Examples

\[ \text{imd.CompositeNormalizedCSOPower} = -5 \]  'Write

value = imd.CompositeNormalizedCSOPower  'Read

C++ Syntax

HRESULT get_CompositeNormalizedCSOPower(double *pVal)
HRESULT put_CompositeNormalizedCSOPower(double  pVal)

Interface

ISweptIMD

Last Modified:

16-Sep-2008  MX New topic
**CompositeNormalizedCTBPower Property**

**Description**
Sets and returns the CSO Power. Valid only with measurement parameters: CTBLo and CTBHi and for Normalization Modes $\text{dBm}$ and $\text{dBmV}$.

**VB Syntax**

```vbnet
imd.CompositeNormalizedCTBPower = value
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>imd</td>
<td>A SweptIMD Object</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>(Double)</td>
<td>Power level. The units are determined by CompositeNormalizationMode Property, which must be set first.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**

```vbnet
imd.CompositeNormalizedCTBPower = -5 'Write
value = imd.CompositeNormalizedCTBPower 'Read
```

**C++ Syntax**

```cpp
HRESULT get_CompositeNormalizedCTBPower(double *pVal)
HRESULT put_CompositeNormalizedCTBPower(double  pVal)
```

**Interface**
ISweptIMD

---

**Last Modified:**

- 7-Apr-2009 Corrected parameters
- 16-Sep-2008 MX New topic
### Compression Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the Compression result of the PNOP marker search. Pnop Comp = Pnop Gain - Linear Gain (not shown on marker readout).</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>comp = pnop.Compression</code></td>
</tr>
<tr>
<td>Variable (Type) - Description</td>
<td><code>comp</code> (double) - Variable to store returned value</td>
</tr>
<tr>
<td></td>
<td><code>pnop</code> A PNOP (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>Not applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>comp = pnop.Compression</code> 'Read'</td>
</tr>
<tr>
<td></td>
<td>See example program</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_Compression(double* pNewVal)</td>
</tr>
<tr>
<td>Interface</td>
<td>IPNOP</td>
</tr>
</tbody>
</table>

Last Modified:

19-Feb-2010 MX New topic
CompressionAlgorithm Property

**Description**  
Set and read the algorithm method used to compute gain compression.

**VB Syntax**  
gca.CompressionAlgorithm = value

**Variable**  
*(Type)* - Description

- **gca**  
  A GainCompression (object)

- **value**  
  (tagNAGCACompressionAlgorithm) - Algorithm method. Choose from:
  - naCompressionFromLinearGain (0)
  - naCompressionFromMaximumGain (1)
  - naBackoffCompression (2)
  - naXYCompression (3)
  - naSaturation (4)

**Return Type**  
Enum

**Default**  
naCompressionFromLinearGain (0)

**Examples**  

```vbnet
gca.CompressionAlgorithm = naXYCompression 'Write
compAlg = gca.CompressionAlgorithm  'Read
```

**C++ Syntax**  
HRESULT get_CompressionAlgorithm(tagNAGCACompressionAlgorithm* pVal)
HRESULT put_CompressionAlgorithm(tagNAGCACompressionAlgorithm newVal)

**Interface**  
IGainCompression

---

Last Modified:

11-Sep-2007  MX New topic
### CompressionBackoff Property

**Description**  
Set and read value for the BackOff compression algorithm.

**VB Syntax**  
gca.CompressionBackoff = value

**Variable**  
(Variable) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gca</td>
<td>GainCompression (object)</td>
<td>A GainCompression (object)</td>
</tr>
<tr>
<td>value</td>
<td>Double</td>
<td>Backoff value in dB. Choose from 30 to (-30)</td>
</tr>
</tbody>
</table>

**Return Type**  
Double

**Default**  
10

**Examples**  
gca.CompressionBackoff = 7 'Write

acqMode = gca.CompressionBackoff 'Read

**C++ Syntax**  
HRESULT get_CompressionBackoff(double* pValue)

HRESULT put_CompressionBackoff(double newValue)

**Interface**  
IGainCompression

---

Last Modified:

21-Nov-2007  MX New topic
CompressionDeltaX Property

Description
Set and read the ‘X’ value in the delta X/Y compression algorithm.

VB Syntax

```vbnet
(gca.CompressionDeltaX = value
```

Variable (Type) - Description

- **gca**: A `GainCompression` (object)
- **value**: (double) X value in dB. Choose from 30 to (-30)

Return Type
Double

Default
10

Examples
```
gca.CompressionDeltaX = 'Write

xDelta = gca.CompressionDeltaX 'Read
```

C++ Syntax

```cpp
HRESULT get_CompressionDeltaX(double* pVal)
HRESULT put_CompressionDeltaX(double newVal)
```

Interface
IGainCompression

Last Modified:
11-Sep-2007   MX New topic
### CompressionDeltaY

**Description**  
Set and read the “Y” value in the delta X/Y compression algorithm.

**VB Syntax**  

gca.CompressionDeltaY = value

**Variable**  

- **Type**
- **Description**
  - *gca*  
    - A [GainCompression](#) 
  - *value*  
    - (double)

**Return Type**  
Double

**Default**  
9

**Examples**  

gca.CompressionDeltaY = 7 'Write

xDelta = gca.CompressionDeltaY 'Read

**C++ Syntax**  

- HRESULT get_CompressionDeltaY(double* pVal)
- HRESULT put_CompressionDeltaY(double newVal)

**Interface**  
IGainCompression

---

Last Modified:  
11-Sep-2007  
MX New topic
CompressionInterpolation Property

**Description**
Sets whether or not interpolation should be performed on 2D measured compression data. Applies ONLY to 2D acquisition modes.

**VB Syntax**
```
gca.CompressionInterpolation = value
```

**Variable** *(Type)* - Description
- **gca** A GainCompression *(object)*
- **value** *(boolean)* - Choose from:
  - **True** Interpolate the results
  - **False** Do NOT interpolate the results but return the value closest to compression.

**Return Type**
Boolean

**Default**
False

**Examples**
```
gca.CompressionInterpolation = True 'Write
compInt = gca.CompressionInterpolation 'Read
```

**C++ Syntax**
```
HRESULT get_CompressionInterpolation(VARIANT_BOOL* pVal)
HRESULT put_CompressionInterpolation(VARIANT_BOOL newVal)
```

**Interface**
IGainCompression

---

Last Modified:

11-Sep-2007 MX New topic
### CompressionLevel Property

**Description** Set and read the desired gain reduction (from reference gain). This value is used for Compression Methods: Compression from Linear Gain and Compression from Maximum Gain.

**VB Syntax**

```vbnet
gca.CompressionLevel = value
```

**Variable**

- **gca**: A `GainCompression` object
- **value**: (Double) - Compression level in dB. Choose a value greater than 0.1 dB.

**Return Type** Double

**Default** 1

**Examples**

```vbnet
gca.CompressionLevel = 1.5 'Write
compLevel = gca.CompressionLevel 'Read
```

**C++ Syntax**

- `HRESULT get_CompressionLevel(double* pVal)`
- `HRESULT put_CompressionLevel(double newVal)`

**Interface** `IGainCompression`

---

**Last Modified:**

11-Sep-2007   MX New topic
## CompressionMax Property

**Description**
Returns the Compression Max result of a PSat or PNOP marker search.

\[ \text{Comp Max} = \text{Gain Max} - \text{Linear Gain} \text{ (not shown on PNOP marker readout).} \]

**VB Syntax**
```vbnet
compMax = pMarker.CompressionMax
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>compMax</td>
<td>(double)</td>
<td>Variable to store returned value</td>
</tr>
<tr>
<td>pMarker</td>
<td>A PNOP</td>
<td>(object) or a PSaturation (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not applicable

**Examples**
```vbnet
compMax = pMark.CompressionMax 'Read
```

See example program

**C++ Syntax**
```c++
HRESULT get_CompressionMax(double* pNewVal)
```

**Interface**
IPNOP or IPSaturation

---

Last Modified:

19-Feb-2010    MX New topic
Write/Read

CompressionLevel (Marker) Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Set and read the marker compression level. First use SearchCompressionPoint to create the compression marker.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td>mkr.CompressionLevel = value</td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td>mkr</td>
<td>A Marker (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Double) - Compression level in dB. Choose any number between: -500 dB to 500 dB Standard gain compression values are positive.</td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>1</td>
</tr>
<tr>
<td>Examples</td>
<td>See example program</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_CompressionLevel(double* pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_CompressionLevel(double newVal)</td>
</tr>
<tr>
<td>Interface</td>
<td>IMarker4</td>
</tr>
</tbody>
</table>

Last Modified:

8-Feb-2009  MX New topic
## CompressionPin Property

**Description**
Reads the input power at the marker compression level. First issue `SearchCompressionPoint Method` or `Tracking Property`.

**VB Syntax**
```vbnet
value = mkr.CompressionPin
```

**Variable (Type) - Description**

- `mkr`: A Marker (object)
- `value`: (Double) - Variable to store the returned input power value.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```vbnet
compLevel=mkr.CompressionPin 'Read
```

**C++ Syntax**
`HRESULT get_CompressionPin(double* pVal)`

**Interface**
IMarker4
### Read-only

**CompressionPout Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Reads the output power at the marker compression level. First issue <a href="#">SearchCompressionPoint Method</a> or <a href="#">Tracking Property</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>value = mkr.CompressionPout</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>mkr</code></td>
<td>A <a href="#">Marker</a> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(Double)</strong> - Variable to store the returned output power value.</td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td>Use <code>compLevel=mkr.CompressionPout</code> to read the output power at the marker compression level. See example program.</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT get_CompressionPout(double* pVal)</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMarker4</td>
</tr>
</tbody>
</table>

*Last Modified: 8-Feb-2009*  
MX New topic
CompressionSaturation Property

**Description**
Returns the Compression Saturation result of a PSat marker search.
Comp Sat = Gain Sat - Gain Linear

**VB Syntax**
`compSat = pSat.CompressionSaturation`

**Variable**
- **(Type)**: (double)
- **Description**: Variable to store returned value

**Return Type**
Double

**Default**
Not applicable

**Examples**
`compSat = pSat.CompressionSaturation` 'Read

See example program

**C++ Syntax**
`HRESULT get_CompressionSaturation(double* pNewVal)`

**Interface**
IPSaturation

Last Modified:
22-Feb-2010  MX New topic
**Read / Write**

**ConnectorType Property**

**Description**
Sets or queries the connector type for the specified port.

**VB Syntax**
\[ obj.ConnectorType (port) = value \]

**Variable**
**Type** - Description
- **obj** - Any of the following:
  - GuidedCalibration (object)
  - SMCType (object)
  - VMCType (object)
- **port** - (Long) Port number of the connector type.
- **value** - (String) - Connector type. **Case-sensitive.**
  Use [ValidConnectorType Property](#) to list connector types.

**Return Type**
String

**Default**
None

**Examples**
- `SMC.ConnectorType(1) = "APC 3.5 male"`
- `value = SMC.ConnectorType(1)`

**C++ Syntax**
- `HRESULT get_ConnectorType(long port, BSTR *connector)`
- `HRESULT put_ConnectorType(long port, BSTR connector)`

**Interface**
- IGuidedCalibration
- SMCType
- VMCType

---

**Last Modified:**
- 13-Aug-2007  Added detail to port argument
ConnectorType Property

Description  
Sets or queries the connector type for the specified port.

VB Syntax  
`ecalUser.ConnectorType (port) = value`

Variable  
(Type) - Description

`ecalUser` An ECalUserCharacterizer (object)

`port` (Enum) ECal port for which connector type is to be set. Choose from:

1 or `naECalPort_A`
2 or `naECalPort_B`
3 or `naECalPort_C`
4 or `naECalPort_D`

`value` (String) - Connector type.

- When the User Characterization is to be stored in the ECal module, then the connector type is limited to a Factory-defined connector type. See the list.
- When the User Characterization is to be stored in PNA disk memory, then the connector type can also be a User-defined connector type.

Return Type  
String

Default  
"" (Empty String)

Examples

`ecalUser.ConnectorType(naECalPort_B) = "APC 3.5 male" ' Write`

`Value = ecalUser.ConnectorType(naECalPort_B)`

C++ Syntax

HRESULT get_ConnectorType(NAECalPort port, BSTR *connector);
HRESULT put_ConnectorType(NAECalPort port, BSTR connector);

Interface  
IECalUserCharacterizer

Last Modified:

18-Sep-2009  Modified for factory connectors (A.09.00)
5-Nov-2008  New topic (8.33)
ControlLines Property

**Description**
Sets the control lines of the specified test set. Control lines, provided through the front panel connector of a test set, are used to control external equipment such as a part handler. See your test set documentation to learn more about control lines.

**VB Syntax**
`tset.ControlLines (chNum) = value`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>tset</code></td>
<td>A <code>TestsetControl</code> object.</td>
</tr>
<tr>
<td><code>OR</code></td>
<td>An <code>E5091Testset</code> object.</td>
</tr>
<tr>
<td><code>chNum</code></td>
<td>(Integer) Channel number of the measurement.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Data value used to set control lines. Values are obtained by adding weights from the following table that correspond to individual lines.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Line</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
</tr>
<tr>
<td>5</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>32</td>
</tr>
<tr>
<td>7</td>
<td>64</td>
</tr>
<tr>
<td>8</td>
<td>128</td>
</tr>
</tbody>
</table>

- The E5091A interprets SENS:MULT1:OUTP 0 as all lines LOW.
- All "Z" and "H" series test sets interpret SENS:MULT1:OUTP 0 as all lines HIGH.

Refer to your test set documentation for setting control line values.

**Return Type**
Variant

**Default**
0
Examples

For a Z5623A K64 test set, the following sets line 3 and 4 OFF; all other lines ON.

testset1.ControlLines(2) = 12

See E5091A Example Program

See External Testset Program

C++ Syntax

HRESULT get_ControlLines(long channelNum, VARIANT *stateByte);
HRESULT put_ControlLines(long channelNum, VARIANT stateByte);

Interface

ITestsetControl

IE5091Testset

Last Modified:

17-Aug-2007 Added different test sets active High and low
Read-only

**Count Property**

**Description**
Returns the number of items in a collection of objects.

**VB Syntax**
`object.Count`

**Variable**
(Type) - Description

- `object` Any of the following (objects):
  - CalFactorSegments Collection
  - CalFactorSegmentsPMAR Collection
  - Cal Sets Collection
  - Channels Collection
  - E5091Testset Collection
  - ExternalDevices Collection
  - ExternalTestsets Collection
  - GuidedCalibrationPowerSensors Collection
  - LimitTest Collection
  - Measurements Collection
  - NaWindows Collection
  - PowerLossSegments Collection
  - PowerLossSegmentsPMAR_Collection.htm
  - PowerSensors Collection
  - Segments Collection
  - Traces Collection
  - PowerMeterInterfaces Collection

**Return Type**
Long Integer

**Default**
Not applicable

**Examples**
```vbnet
numofchans = chans.Count 'return the number of channels
```

**C++ Syntax**
```cpp
HRESULT get_Count(long *pInterface)
```

**Interface**
All listed above
8-Feb-2011  Added GuidedCalibratioPowerSensors (9.33)
31-Jul-2009  Added ExternalDevices (9.0)
6-Mar-2009  PowerMeterInterfaces
### CouplePorts Property

**Description**
Turns ON and OFF port power coupling. ON means the power level is the same for both ports. OFF means the power level may be set independently for each port.

**VB Syntax**

\[ \text{object}.\text{CouplePorts} = \text{value} \]

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>Channel (object) or CalSet (object) - Read-only property</td>
</tr>
</tbody>
</table>

**Value**

(enum NAStates) Choose from:

- 0 - NaOff - Turns coupling OFF
- 1 - NaOn - Turns coupling ON

**Return Type**

Long Integer

1 - ON
0 - OFF

**Default**

NaON (1)

**Examples**

\[ \text{chan.CouplePorts} = \text{NaOff} \] `Write`

\[ \text{couplport} = \text{chan.CouplePorts} \] `Read`

**C++ Syntax**

HRESULT get_CouplePorts(tagNAStates *pState)
HRESULT put_CouplePorts(tagNAStates newState)

**Interface**

IChannel | CalSet3
CoupleChannelParams Property

Description
Turns ON and OFF Time Domain Trace Coupling. All of the measurements in the specified channel are coupled.

- To select Transform parameters to couple, use Trans.CoupledParameters Property
- To select Gating parameters to couple, use Gate.CoupledParameters Property

VB Syntax

```vbnet
chan.CoupleChannelParams = state
```

**Variable** *(Type) - Description*

- `chan` A [Channel](object) (object)
- `state` (boolean)
  - `False` - Turns Trace Coupling OFF
  - `True` - Turns Trace Coupling ON

**Return Type**
Boolean

**Default**
True

**Examples**

```vbnet
chan.CoupleChannelParams = False 'Write
```

```vbnet
couple = chan.CoupleChannelParams 'Read
```

**C++ Syntax**

```c
HRESULT get_CoupleChannelParams(VARIANT_BOOL *isCoupled);
HRESULT put_CoupleChannelParams(VARIANT_BOOL isCoupled);
```

**Interface**
IChannel5
Coupled Property

**Description**
Sets and returns the state of coupling (ON or OFF) of this range to the primary range.

**VB Syntax**

```
FOMRange.Coupled = value
```

**Variable** (Type) - Description

- **object**
  - An `FOMRange` object

- **value** (boolean) - State of coupling.
  - `True` - Couple range to primary range.
  - `False` - Do NOT couple to primary range.

**Return Type**
Boolean

**Default**
True

**Examples**

```
fomRange.Coupled = False 'this range is NOT coupled to the primary range.
coupl = fomRange.Coupled 'Read
```

**C++ Syntax**

```
HRESULT get_Coupled(VARIANT_BOOL *pVal)
HRESULT put_Coupled(VARIANT_BOOL pVal)
```

**Interface**
IFOMRange
**CoupledMarkers Property**

**Description**
Sets and Reads the state of Coupled Markers (ON and OFF)

**VB Syntax**
```vbnet
app.CoupledMarkers = state
```

**Variable**
- **app** (Type) - An Application (object)
- **state** (boolean)
  - False (0) - Turns Coupled Markers OFF
  - True (1) - Turns Coupled Markers ON

**Return Type**
Boolean
- False - OFF
- True - ON

**Default**
False

**Examples**
```vbnet
app.CoupledMarkers = True 'Write
coupl = app.CoupledMarkers 'Read
```

**C++ Syntax**
```cpp
HRESULT put_CoupledMarkers(VARIANT_BOOL bState)
HRESULT get_CoupledMarkers(VARIANT_BOOL *bState)
```

**Interface**
IApplication
**CoupledParameters Property (Gating)**

**Description**
Specifies the time domain gating parameters to be coupled. The settings for those parameters will be copied from the active measurement to all other measurements on the channel.

To turn coupling ON and OFF, use `CoupleChannelParams Property`

To specify Transform parameters to couple, use `Transform.CoupledParameters Property`

**VB Syntax**
```
gate.CoupledParameters = value
```

**Variable**
```
trans A Gating (object)
value (Enum As NAGatingCoupledParams) - Parameters to couple. To specify more than one parameter, add the numbers. Choose from:
1 - naGatingStimulusCoupled (Start, Stop, Center, and Span TIME settings.)
2 - naGateStateCoupled (ON / OFF)
4 - naGatingShapeCoupled (Minimum, Normal, Wide, and Maximum)
8 - naGatingTypeCoupled (Bandpass and Notch)
```

**Return Type**
Enum

**Default**
29

**Examples**
```
gate.CoupledParameters = 15 'Couple all parameters
```

```
CP = gate.CoupledParameters 'Read
```

**C++ Syntax**
```
HRESULT get_CoupledParameters(long *lParams);
HRESULT put_CoupledParameters(long lParams);
```

**Interface**
IGating2
**CoupledParameters Property (Transform)**

**Description**
Specifies the time domain transform parameters to be coupled. The settings for those parameters will be copied from the active measurement to all other measurements on the channel.

To turn coupling ON and OFF, use `CoupleChannelParams Property`

To specify Gating parameters to couple, use `Gate.CoupledParameters Property`

**VB Syntax**
```
trans.CoupledParameters = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>trans</code></td>
<td>Transform (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>(Enum As NATransformCoupledParams)</th>
<th>Parameters to couple. To specify more than one parameter, add the numbers. Choose from:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 - naTransformStimulusCoupled</td>
<td>(Start, Stop, Center, and Span TIME settings.)</td>
</tr>
<tr>
<td>2 - naTransformStateCoupled</td>
<td>(ON / OFF)</td>
</tr>
<tr>
<td>4 - naTransformWindowCoupled</td>
<td>(Kaiser Beta / Impulse Width)</td>
</tr>
<tr>
<td>8 - naTransformModeCoupled</td>
<td>(Low Pass Impulse, Low Pass Step, Band Pass)</td>
</tr>
<tr>
<td>16 - naTransformDistMkrUnitCoupled</td>
<td>(Distance maker Units)</td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
29

**Examples**
```
trans.CoupledParameters = 31 'Couple all parameters
```
```
CP = trans.CoupledParameters 'Read
```

**C++ Syntax**
```
HRESULT get_CoupledParameters(long *lParams);
HRESULT put_CoupledParameters(long lParams);
```

**Interface**
ITransform2
### CouplePhasePortSettings Property

**Description**
Sets and returns whether to couple phase control settings (IFBW, Tolerance, Max Iterations).

**VB Syntax**

```vbnet
phase.CouplePhasePortSettings(srcPort) = value
```

**Variable**

- **phase** (Type) - Description
  
  A [PhaseControl](#) Object

- **srcPort** (Long Integer) Source port for which to make phase control settings.
  
  **Note**: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value** (Boolean) Coupling state. Choose from:
  
  - **True** - Couple phase control settings. The phase control settings from <port> are copied to the other phase-controlled ports.
  
  - **False** - Do NOT couple phase control settings. The phase control settings for each phase-controlled port are made independently.

**Return Type**

Boolean

**Default**

False

**Examples**

```vbnet
phase.CouplePhasePortSettings 1 = True ' Write
value = phase.CouplePhasePortSettings 2 ' Read
```

**C++ Syntax**

```c++
HRESULT get_CouplePhasePortSettings(long port, VARIANT_BOOL* pVal);
HRESULT put_CouplePhasePortSettings(long port, VARIANT_BOOL newVal);
```

**Interface**

IPhaseControl

---

**Last Modified:**

8-Dec-2010  MX New topic
**CoupleTonePower Property**

**Description**  Sets and returns the ON | OFF state of power coupling for F1 and F2.

**VB Syntax**  
`object.CoupleTonePower = value`

**Variable**  
*object* - A SweptIMD, or IMSpectrum Object

*value* - (Boolean) - Choose from:

- **True** - F1 and F2 power is coupled.
- **False** - F1 and F2 power is NOT coupled. Set power levels individually.

**Return Type**  Boolean

**Default**  True

**Examples**  
`ims.CoupleTonePower = true`  'Write

`value = ims.CoupleTonePower`  'Read

**C++ Syntax**  
- HRESULT get_CoupleTonePower(VARIANT_BOOL* val)
- HRESULT put_CoupleTonePower(VARIANT_BOOL val)

**Interface**  ISweptIMD

IIMSMpectrum

---

Last Modified:

19-Aug-2008  MX New topic
**CpuRevision Property**

**Description**

Returns a number that corresponds to the CPU speed of the PNA.

**VB Syntax**

```vbnet
value = cap.CpuRevision
```

**Variable**

**Type** - Description

- `value` (String) - Variable to store the returned number. Use the following table to learn the CPU speed.

<table>
<thead>
<tr>
<th>Reported CPU version</th>
<th>Clock speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>266 MHz</td>
</tr>
<tr>
<td>2.0</td>
<td>500 MHz</td>
</tr>
<tr>
<td>3.0</td>
<td>1100 MHz</td>
</tr>
<tr>
<td>4.0</td>
<td>1600 MHz</td>
</tr>
<tr>
<td>5.0</td>
<td>2000 MHz</td>
</tr>
</tbody>
</table>

- `cap` A [Capabilities](#) (object)

**Return Type**

String

**Default**

Not Applicable

**Examples**

```vbnet
value = cap.CpuRevision 'Read
```

**C++ Syntax**

```cpp
HRESULT get_CpuRevision(BSTR *value);
```

**Interface**

ICapabilities6

---

Last Modified:

30-Nov-2009  MX New topic
**CSONumDistortionProducts Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the &quot;N = number of distortion products&quot; value for the calculation of the CSO parameter. <a href="#">Learn more.</a></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>imd.CSONumDistortionProducts = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td>imd</td>
<td>A <a href="#">SweptIMD</a> Object</td>
</tr>
<tr>
<td>value</td>
<td>(Long Integer) Number of distortion products</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Long Integer</td>
</tr>
<tr>
<td>Default</td>
<td>40</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>imd.CSONumDistortionProducts = True 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>value = imd.CSONumDistortionProducts 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_CSONumDistortionProducts(long *pVal)</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_CSONumDistortionProducts(long *pVal)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ISweptIMD</td>
</tr>
</tbody>
</table>

Last Modified:

- 9-Jun-2010 Fixed type
- 25-Aug-2008 MX New topic
CSOOffset Property

Description
Sets and returns the offset that is applied to CSO measurements. Valid only with measurement parameters: CSO2Lo and CSO2Hi.

VB Syntax
`imd.CSOOffset = value`

Variable (Type) - Description
`imd` A SweptIMD Object
`value` (Double) Offset value in dBm.

Return Type
Double

Default
0

Examples
`imd.CSOOffset = 2 'Write`
`value = imd.CSOOffset 'Read`

C++ Syntax
HRESULT get_CSOOffset(double *pVal)
HRESULT put_CSOOffset(double pVal)

Interface
ISweptIMD

Last Modified:
16-Sep-2008 MX New topic
**CTBOffset Property**

**Description**
Sets and returns the offset that is applied to CTB measurements. Valid only with measurement parameters: CTB, CTBLo, CTBHi, CTBE, CTBELo, and CTBEHi.

**VB Syntax**

```
imd.CTBOffset = value
```

**Variable - Description**

- **imd**
  A SweptIMD Object

- **value**
  (Double) Offset value in dBm.

**Return Type**
Double

**Default**
0

**Examples**

```
imd.CTBOffset = 2 'Write

value = imd.CTBOffset 'Read
```

**C++ Syntax**

```
HRESULT get_CTBOffset(double *pVal)
HRESULT put_CTBOffset(double pVal)
```

**Interface**
ISweptIMD

---

Last Modified:

- 7-Apr-2009  Added valid parameters
- 16-Sep-2008  MX New topic
### CTBXMODNumCarriers Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets the “N = Total number of carriers” value used in the calculation of the XMOD and CTB parameter. <a href="#">Learn more.</a></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>imd.CTBXMODNumCarriers = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>Type</strong> - Description</td>
</tr>
<tr>
<td><code>imd</code></td>
<td>A <a href="#">SweptIMD</a> Object</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Long Integer) Number of carriers.</td>
</tr>
<tr>
<td>Return Type</td>
<td>Long Integer</td>
</tr>
<tr>
<td>Default</td>
<td>40</td>
</tr>
</tbody>
</table>

#### Examples

- `imd.CTBXMODNumCarriers = 15` 'Write
- `value = imd.CTBXMODNumCarriers` 'Read

#### C++ Syntax

- `HRESULT get_CTBXMODNumCarriers(double *pVal)`
- `HRESULT put_CTBXMODNumCarriers(double *pVal)`

#### Interface

- ISweptIMD

---

**Last Modified:**

- 25-Aug-2008 MX New topic
Read-only

CustomCalConfiguration Property

**Description** Calibration for the following PNA Applications is performed using the GuidedCalibration Object. This command provides access to additional Properties and Methods which extends the GuidedCal Object.

<table>
<thead>
<tr>
<th>Meas Type</th>
<th>Custom Cal Interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gain Compression</td>
<td>IGainCompressionCal Object</td>
</tr>
<tr>
<td>Noise Figure</td>
<td>INoiseCal Object</td>
</tr>
<tr>
<td>Swept IMD</td>
<td>ISweptIMD Cal Object</td>
</tr>
</tbody>
</table>

**VB Syntax**

```vbscript
set custCal = cal.CustomCalConfiguration()
```

**Variable** *(Type) - Description*

- `custCal` *(object)* The handle to an interface that provides application-specific calibration properties.

  - `cal` GuidedCalibration Object.

**Return Type** Depends on the MeasType. See above table.

**Default** None

**Examples** See examples: NoiseFigure GainCompression SweptIMD

**C++ Syntax**

```cpp
HRESULT CustomCalConfiguration(IDispatch** value);
```

**Interface** IGuidedCalibration4

---

Last Modified:

- 22-Jul-2009 Updated description
- 8-Sep-2008 MX New topic
CustomChannelConfiguration Property

**Description**
Returns a handle to the custom application object on the active channel. You can either (1) use the handle directly to access measurement properties and methods, or (2) set a variable to the measurement object. The variable retains a handle to the original measurement.

Currently, the custom application objects to which this property provides access are:

- **NoiseFigure Object**
- **GainCompression Object**
- **SweptIMD Object**

**VB Syntax**
1) `set custChan = chan.CustomChannelConfiguration. <setting>`
or
2) `set custChan = app.ActiveChannel.CustomChannelConfiguration`

**Variable (Type) - Description**

- **custChan**
  A variable in which the handle to a custom application is returned. *(object)*

- **chan**
  A Channel *(object)*

- **<setting>**
  A property or method setting on the custom application object.

**Return Type**
Custom application object

**Default**
None

**Examples**
See examples: NoiseFigure GainCompression SweptIMD

**C++ Syntax**
HRESULT CustomChannelConfiguration(IDispatch** value);

**Interface**
IChannel12

---

Last Modified:

27-Aug-2008  Added Swept IMD
16-Oct-2007  MX New topic
**CustomMeasurementConfiguration Property**

**Description**
Returns a handle to a custom measurement object on the active channel. You can use the handle to access custom measurement properties and methods. Currently, the custom measurement objects to which this property provides access is:

- **GainCompressionMeas Object**

**VB Syntax**

```vbnet
Set custMeas = meas.CustomMeasurementConfiguration
```

**Variable (Type) - Description**

- **custMeas**
  A variable in which the handle to a custom measurement is returned. *(object)*

- **chan**
  A *Measurement* *(object)*

**Return Type**
Custom Measurement object

**Default**
None

**Examples**
See examples: [GainCompression](#)

**C++ Syntax**

```cpp
HRESULT CustomMeasurementConfiguration(IDispatch** value);
```

**Interface**
IMeasurement12

---

Last Modified:

3-Sep-2009   MX New topic
Write/Read

CWFrequency Property

Description
Set the Continuous Wave (CW) frequency. Must first send chan.SweepType = naCWTimeSweep.
See Also: calset.CWFrequency Property

VB Syntax
object.CWFrequency = value

Variable (Type) - Description

object One of the following:

- Channel (object)
- FOMRange (object) Range must be UNCOUPLED.

See also Measurement2 interface.

double value CW frequency. Choose any number between:
the minimum and maximum frequency limits of the analyzer
Units are Hz

Return Type Double

Default 1e9

Examples
chan.CWFrequency = 5e9 'Write
cwfreq = chan.CWFrequency 'Read

C++ Syntax
HRESULT put_CWFrequency(double newVal)
HRESULT get_CWFrequency(double *pVal)

Interface IChannel
IFOMRange
Read-only

Data Property

Description
Reads the next specified number of data points from the FIFO buffer. The data is returned in 32 bit real/imaginary pair. Data is cleared as it is read.

Note: This method is the slowest way to transfer data using COM. However, it is supported by all COM client programming languages. For better performance, try using DataInCompactForm.

VB Syntax
array = fifo.Data (count)

Variable
- Description

array (Variant) Variable to store the returned data.

tifo A FIFO Object

count (Long Integer) Number of data points to read.

Return Type
Variant array. Each VARIANT is typed as a 4-byte floating point number.

Default
Not Applicable

Examples
value = fifo.Data(500) 'Read

C++ Syntax
HRESULT get_Data(long count,VARIANT * data);

Interface
IFIFO

Last Modified:
7-Oct-2008    MX New topic
### DataAndLimits Property

**Description**  
Set and return the color of Data and Limit Lines for nth trace in a window.

**VB Syntax**  
\[ trace(n).DataAndLimits = value \]

**Variable**  
**(Type)** - **Description**

- **trace(n)**: One of the 8 ComTraceColors objects
- **value** *(Long Integer)*: RGB color of the DataAndLimits pen.

Convert the three RGB colors to an integer as follows:

\[ \text{RGB} = R + (G \times 2^8) + (B \times 2^{16}) \]

To find the three RGB values from the Display Colors dialog, click **Change Color**, then **Define Custom Color**.

**Return Type**  
Long

**Default**  
Varies for each trace.

**Examples**

- \[ R = 10 \]
- \[ G = 10 \]
- \[ B = 10 \]
- \[ \text{RGB} = 10 + (10 \times 2^8) + (10 \times 2^{16}) \]

\[ \text{trace1.DataAndLimits} = \text{RGB} \ 'Write\]

\[ \text{color} = \text{trace1.DataAndLimits} \ 'Read\]

**C++ Syntax**

```cpp
HRESULT get_DataAndLimits(long* pVal);
HRESULT put_DataAndLimits(long newVal);
```

**Interface**  
IComTraceColors

---

Last Modified:

7-Aug-2009   MX New topic
DataCount Property

Description  Returns the total number of data points in the FIFO buffer.

VB Syntax  

\[ value = \text{fifo}.\text{DataCount} \]

Variable (Type) - Description

\begin{itemize}
  \item \textit{value} (Long Integer) Variable to store the returned number of data points.
  \item \textit{fifo} A FIFO Object
\end{itemize}

Return Type  Long Integer

Default  Not Applicable

Examples  

\[ value = \text{fifo}.\text{DataCount} \]

C++ Syntax  

\begin{verbatim}
HRESULT get_DataCount(long *value)
\end{verbatim}

Interface  IFIFO

Last Modified:  

7-Oct-2008  MX New topic
**DataInCompactForm Property**

**Description**  
Reads FIFO data the same as Data Property but returns the data in a more compact form of SAFEARRAY. This is significantly faster but it is not supported in all client environments.

**VB Syntax**  
`array = fifo.DataInCompactForm(count)`

**Variable (Type) - Description**

- **array**  (Variant) Variable to store the returned data
- **fifo**  A FIFO Object
- **count**  (Long Integer) Number of data points to read.

**Return Type**  
Returns an array of 4 byte floating point numbers.

**Default**  
Not Applicable

**Examples**  
`value = fifo.DataInCompactForm(500)`  

**C++ Syntax**  
`HRESULT get_DataInCompactForm(long count, VARIANT * data);`

**Interface**  
IFIFO
**Delay Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and Returns the electrical delay value for the calibration standard.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td></td>
</tr>
</tbody>
</table>

```
calstd.Delay (n) = value
```

<table>
<thead>
<tr>
<th>Variable</th>
<th><strong>(Type) - Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>n</code></td>
<td>Pulse Generator number.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(single) - Electrical delay in picoseconds</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>C++ Syntax</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>HRESULT get_Delay(float *pVal)</td>
<td></td>
</tr>
<tr>
<td>HRESULT put_Delay(float newVal)</td>
<td></td>
</tr>
</tbody>
</table>

| Interface | ICalStandard |

---

**Examples**

```
calstd.Delay = 12  'Write 12ps Delay
```

```
stdDelay = calstd.Delay  'Read the value of Delay
```

**Last Modified:** 5-Jan-2007  MX New topic
Delay (pulse) Property

**Description**
Sets the pulse delay - the amount of time before a new pulse begins.

**VB Syntax**
`pulse.Delay(n) = value`

**Variable (Type) - Description**
- `pulse` (PulseGenerator object)
  - n (Integer) Pulse generator number. Choose from 0 to 4. 0 is the generator that pulses the ADC.
  - value (Double) Delay value in seconds. Choose a value from about 33ns to about 70 seconds.

**Return Type**
Double

**Default**
0

**Examples**
```
pulse.Delay(1) = 1ms 'Write
```
```
value = pulse.Delay(4) 'Read
```

**C++ Syntax**
```
HRESULT get_Delay(integer pulse, double* delay);
HRESULT put_Delay(integer pulse, double delay);
```

**Interface**
IPulseGenerator

---

Last Modified:
1-Jan-2007  MX New topic
### Delay Property

**Description**
Specifies the delay that should be applied by the PNA after the aux trigger input is received and before the acquisition is made.

**Note:** Use on PNA-X ONLY. Other models do NOT have an Aux Input.

**VB Syntax**
```vb
auxTrig.Delay = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>auxTrig</td>
<td>An <strong>AuxTrigger</strong> (object)</td>
</tr>
<tr>
<td>value</td>
<td>(double) - Delay value in seconds. Choose a value between 0 and 3.0 seconds.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```vb
auxTrig.Delay = 1.2 'Write 1.2s Delay
```

```vb
value = auxTrig.Delay 'Read the value
```

**C++ Syntax**
```cpp
HRESULT get_Delay(double *val);
HRESULT put_Delay(double val);
```

**Interface**
IAuxTrigger

---

Last Modified:
- 5-Sep-2008    Added Note
- 14-Dec-2006    MX New topic
DelayCalculationMethod Property

**Description** Set and return the method of setting the delay through the calibration mixer.

**VB Syntax**
```
smc.DelayCalculationMethod = value
```

**Variable (Type) - Description**
- `smc`: An SMCType (object)
- `value`: (Enum as NADelayCalculationMethod)

  0 - `naDelayCalculationMethod_FixedDelay` - use a known delay value set with `FixedDelay Property`
  1 - `naDelayCalculationMethod_MixerCharacterizationFile` - use the S2P file set with `MixerCharacterizationFile Property`

**Return Type** Enum

**Default** 0 - `naDelayCalculationMethod_FixedDelay`

**Example**
```
SMC.DelayCalculationMethod = naDelayCalculationMethod_FixedDelay
```

**C++ Syntax**
```
HRESULT put_DelayCalculationMethod(tagNADelayCalculationMethod Value);
HRESULT get_DelayCalculationMethod(tagNADelayCalculationMethod* Value);
```

**Interface** SMCType5

---

Last Modified:
25-Mar-2010  MX New topic
DelayIncrement Property

Description: Sets the pulse delay increment. The delay increments with each pulse by the <value> amount. For example, in this diagram the delay starts as 1. On the second pulse, delay=2. On the third pulse, delay=3.

Important: If D + W is greater than P, then undefined PNA behavior results. There is NO error message or warning. Delay includes the incremented value.

This is useful for pulse profiling.

VB Syntax: pulse.DelayIncrement(n) = value

Variable: (Type) - Description

pulse: A PulseGenerator (object)

n: (Integer) Pulse generator number. Choose from 0 to 4. 0 is the generator that pulses the ADC.

value: (Double) Delay increment value in seconds.

Return Type: Double

Default: 0

Examples: pulse.DelayIncrement(1) = 1ms 'Write

value = pulse.DelayIncrement(4) 'Read

C++ Syntax: HRESULT get_DelayIncrement(integer pulse, double* dIncr);
HRESULT put_DelayIncrement(integer pulse, double dIncr);

Interface: IPulseGenerator

Last Modified:
18-Jun-2007  MX New topic
Write/Read

DeltaFrequency Property

**Description**
Sets and returns the fixed tone spacing value. Use with IMD sweep types:

- `naIMDToneCWSweep`
- `naIMDTonePowerSweep`
- `naIMDToneCenterFreqSweep`

**VB Syntax**
```
object.DeltaFrequency = value
```

**Variable**
*(Type) - Description*

- **object**
  A SweptIMD or IMSpectrum Object

- **value**
  (Double) - Tone spacing frequency in Hz. Both the F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**
Double

**Default**
1 MHz

**Examples**
```
imd.DeltaFrequency = 1e6 'Write
value = imd.DeltaFrequency 'Read
```

**C++ Syntax**
```
HRESULT get_DeltaFrequency(double *pVal)
HRESULT put_DeltaFrequency(double newVal)
```

**Interface**
ISweptIMD
IMSpectrum

---

Last Modified:

19-Aug-2008  MX New topic
**DeltaFrequencyStart Property**

**Description**  
Sets and returns the starting main tone separation for \(\text{sweep type} = \text{naIMDDeltaFrequencySweep}\)

**VB Syntax**  
\(\text{imd}.\text{DeltaFrequencyStart} = \text{value}\)

**Variable**  
**Type** - Description

- **imd**  
  A **SweptIMD** Object

- **value**  
  (Double) - Starting tone separation between F1 and F2 in Hz. Both F1 and F2 tones MUST be within the frequency range of the PNA where:
  
  \[
  \begin{align*}
  \text{F1 (start)} &= \text{imd.FrequencyCenter} - \text{imd.DeltaFrequencyStart} / 2 \\
  \text{F2 (start)} &= \text{imd.FrequencyCenter} + \text{imd.DeltaFrequencyStart} / 2
  \end{align*}
  \]

**Return Type**  
Double

**Default**  
1 MHz

**Examples**

\[
\begin{align*}
\text{imd.DeltaFrequencyStart} &= 5e6 \quad \text{'Write} \\
\text{value} &= \text{imd.DeltaFrequencyStart} \quad \text{'Read}
\end{align*}
\]

**C++ Syntax**

- HRESULT get_DeltaFrequencyStart(double *pVal))
- HRESULT put_DeltaFrequencyStart(double newVal)

**Interface**  
ISweptIMD

---

Last Modified:

19-Aug-2008  MX New topic
**DeltaFrequencyStop Property**

**Description**
Sets and returns the stop spacing of the main tones. Use with `sweep type = naIMDDeltaFrequencySweep`

**VB Syntax**
```
imd.DeltaFrequencyStop = value
```

**Variable (Type) - Description**

`imd` A SweptIMD Object

`value` (Double) - Stopping tone separation between F1 and F2 in Hz. Both F1 and F2 tones MUST be within the frequency range of the PNA where:

- F1 (stop) = `imd.FrequencyCenter – imd.DeltaFrequencyStop / 2`
- F2 (stop) = `imd.FrequencyCenter + imd.DeltaFrequencyStop / 2`

**Return Type**
Double

**Default**
10 MHz

**Examples**
```
imd.DeltaFrequencyStop = 20e6  'Write

value = imd.DeltaFrequencyStop  'Read
```

**C++ Syntax**
```
HRESULT get_DeltaFrequencyStop(double *pVal))
HRESULT put_DeltaFrequencyStop(double newVal)
```

**Interface**
ISweptIMD

---

Last Modified:

19-Aug-2008   MX New topic
**Description**
Sets a marker as a delta marker. The reference marker must already be turned ON. See `meas.ReferenceMarkerState`.

**VB Syntax**
`mark.DeltaMarker = state`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>mark</code></td>
<td>A Marker (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean) -</td>
</tr>
<tr>
<td><strong>True</strong></td>
<td>marker is a delta marker</td>
</tr>
<tr>
<td><strong>False</strong></td>
<td>marker is NOT a delta marker</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**

- `mark.DeltaMarker = True` 'Write
- `delta = mark.DeltaMarker` 'Read

**C++ Syntax**

```
HRESULT get_DeltaMarker(VARIANT_BOOL bState)
HRESULT put_DeltaMarker(VARIANT_BOOL *bState)
```

**Interface**
IMarker
Description Property

Sets or returns the descriptive string assigned to the Cal Set. Change this string so that you can easily identify each Cal Set constructed.

**Description**

VB Syntax:  

```vbnet
CalSet.Description = value
```

Variable (Type) - Description

- **CalSet** (object) - A **Cal Set** object
- **value** (string) – Description of the Cal Set

Return Type

String

Default

“CalSet_n” where n is an integer number.

Examples

- `CalSet.Description = "My Cal Set"` 'Write
- `desc = CalSet.Description` 'Read

C++ Syntax

```cpp
HRESULT get_Description(BSTR *pVal)
HRESULT put_Description(BSTR newVal);
```

Interface

ICalSet

Last modified:

Dec. 12, 2006  MX New topic
**DescriptiveText Property**

**Description**  Write and read descriptive text associated with the configuration. This text is displayed in the path configuration dialog. Text is generally used to describe external connections that must be made manually to complete the configuration setup.

**VB Syntax**  

```vbnet
pathConfig.DescriptiveText = text
```

**Variable**  

**name**  (String) Variable to store the returned configuration name.

**pathConfig**  A **PathConfiguration** (object)

**text**  (String) Descriptive text enclosed in quotes.

**Return Type**  String

**Default**  Not Applicable

**Examples**  

```vbnet
pathConf.DescriptiveText "here are the instructions for connecting the device for this configuration"
```

**C++ Syntax**  

```c++
HRESULT get_DescriptionText(BSTR* pConnectionText );
HRESULT put_DescriptionText(BSTR connectionText );
```

**Interface**  IPathConfiguration

---

Last Modified:

14-Dec-2006    MX New topic
DeviceInputPort Property

Description
Returns the PNA port number to be used for the DUT input.

VB Syntax
$port = obj.DeviceInputPort$

Variable
$(Type) - Description$

$port$ (long) Variable to store the PNA port number of the DUT input.

$mixer$ A Mixer (object)

Return Type
Long

Default
1

Examples
$value = mixer.DeviceInputPort$

C++ Syntax
HRESULT get_DeviceInputPort(long *pVal)

Interface
IMixer8

Last Modified:
23-Apr-2008   New topic.
### DeviceInputPort Property

**Description**  
Read the PNA port number which is connected to the DUT input. Use `SetPortMap Method` to change the port mapping.

**VB Syntax**  
`obj.DeviceInputPort`

**Variable**  
**(Type) - Description**

- `obj`: A `Converter (object)` or
  - A `GainCompression (object)` or
  - A `SweptIMD (object)` or
  - An `IMSpectrum (object)`
  - A `NoiseFigure (object)`

**Return Type**  
Integer

**Default**  
1

**Examples**  
`inPort = gca.DeviceInputPort` 'Read

**C++ Syntax**  
`HRESULT get_DeviceInputPort(int* pVal)`

**Interface**  
`IGainCompression`
- `ISweptIMD`
- `IMSpectrum`
- `INoiseFigure6`
- `IConverter6`

---

**Last Modified:**

- 23-May-2011  Added converter
- 30-Apr-2010  Added NF
- 11-Aug-2009  Added IMD and IMS (9.0)
- 11-Sep-2007  MX New topic
DeviceOutputPort Property

Description: Returns the PNA port number to be used for the DUT output.

VB Syntax:  

```
port = mixer.DeviceOutputPort
```

Variable:  

**Type** - Description

- `port` (long): Variable to store the PNA port number of the DUT output.
- `mixer` A **Mixer** (object)

Return Type: Long

Default: 2

Examples:  

```
value = mixer.DeviceOutputPort
```

C++ Syntax:  

```
HRESULT get_DeviceOutputPort(long *pVal)
```

Interface: IMixer8

---

Last Modified:  

23-Apr-2008   New topic.
**Write/Read**

**InputLinearPowerLevel Property**

**Description**
Set and read the input power at which Linear Gain and all S-parameters are measured.

**VB Syntax**
`gca.InputLinearPowerLevel = value`

**Variable**
- **(Type)** - Description
  - `gca` A GainCompression (object)
  - `value` (double) Linear input power level in dBm. Choose a value from +30 to (-30).

**Return Type**
Double

**Default**
-25 dBm

**Examples**
- `gca.InputLinearPowerLevel = -10` 'Write
- `LinPwr = gca.InputLinearPowerLevel` 'Read

**C++ Syntax**
- `HRESULT get_InputLinearPowerLevel(double* pVal)`
- `HRESULT put_InputLinearPowerLevel(double newVal)`

**Interface**
IGainCompression

---

**Last Modified:**
11-Sep-2007  MX New topic
DeviceOutputPort Property

**Description**
Read the PNA port number which is connected to the DUT Output.
Use [SetPortMap Method](#) to change the port mapping.

**VB Syntax**
```
obj.DeviceOutputPort
```

**Variable**
(Type) - Description

- **obj**
  - A [Converter](#) (object) or
  - A [GainCompression](#) (object) or
  - A [SweptIMD](#) (object) or
  - An [IM Spectrum](#) (object)
  - A [NoiseFigure](#) (object)

**Return Type**
Integer

**Default**
2

**Examples**
```
outPort = gca.DeviceOutputPort 'Read
```

**C++ Syntax**
```
HRESULT get_DeviceOutputPort(int* pVal)
```

**Interface**
IGainCompression
ISweptIMD
IMSpectrum
INoiseFigure6
IConverter6

---

**Last Modified:**
- 23-May-2011  Added Converter
- 30-Apr-2010  Added NF
- 11-Aug-2009  Added IMD and IMS
- 11-Sep-2007  MX New topic
# DeviceType Property

**Description**
Sets and returns the DeviceType for the external device.

**VB Syntax**
```vbnet
extDevices.DeviceType = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>extDevices</td>
<td>An <code>ExternalDevice</code> (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
</table>
| value    | Device Type. Choose from:
  - "Source" for external source.
  - "Power Meter" for power meter. |

**Return Type**
String

**Default**
"Unknown"

**Examples**
```
extDevices.DeviceType = "source"  'Write
value = extDevices.DeviceType   'Read
```

See example program to configure [PMAR device](#)

See example program to configure [External Source](#)

**C++ Syntax**

```c++
HRESULT get_DeviceType( BSTR* value);
HRESULT put_DeviceType( BSTR newVal);
```

**Interface**
`IExternalDevices`

---

Last Modified:

31-Jul-2009   MX New topic
### DiffPortMatch_C Property

**Description**  
Sets the Capacitance value of the differential matching circuit.

**VB Syntax**  
`fixture.DiffPortMatch_C(portNum) = value`

**Variable**  
- **Type**: Description  
  - **fixture**: A `Fixturing` object  
  - **portNum**: Integer  
    - Balanced (logical) port number. Choose from logical ports 1, 2, 3. [Learn more about logical ports](#)  
  - **value**: Double  
    - Capacitance value in farads. Choose a value between `-1E18` to `1E18`.

**Return Type**  
Double

**Default**  
0

**Examples**  
- `fixture.DiffPortMatch_C(2) = 1e-6` 'Write
- `value = fixture.DiffPortMatch_C(1)` 'Read

**C++ Syntax**  
- `HRESULT get_DiffPortMatch_C( short portNum, double *pVal)`  
- `HRESULT put_DiffPortMatch_C( short portNum, double newVal)`

**Interface**  
`IFixturing2`
## DiffPortMatch_G Property

**Description**  Sets the Conductance value of the differential matching circuit.

**VB Syntax**  
```vbnet
fixture.DiffPortMatch_G(portNum) = value
```

**Variable**  (Type) - Description

- **fixture**  A Fixturing (object)
- **portNum**  (Integer)  Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.
- **value**  (Double)  Conductance value in siemens. Choose a value between -1E18 to 1E18.

**Return Type**  Double

**Default**  0

**Examples**  
```vbnet
fixture.DiffPortMatch_G(2) = 1e-3  'Write
value = fixture.DiffPortMatch_G(1)  'Read
```

**C++ Syntax**  
```cpp
HRESULT get_DiffPortMatch_G( short portNum, double *pVal)
HRESULT put_DiffPortMatch_G( short portNum, double newVal)
```

**Interface**  IFixturing2
**DiffPortMatch_L Property**

**Description**
Sets the Inductance value of the differential matching circuit.

**VB Syntax**
```vbnet
fixture.DiffPortMatch_L(portNum) = value
```

**Variable**
- **Type** - Description
- **fixture** (A Fixturing object)
- **portNum** (Integer) Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.
- **value** (Double) Inductance value in henries. Choose a value between -1E18 to 1E18.

**Return Type**
Double

**Default**
0

**Examples**
```vbnet
fixture.DiffPortMatch_L(2) = 1e-3 'Write
value = fixture.DiffPortMatch_L(1) 'Read
```

**C++ Syntax**
```c++
HRESULT get_DiffPortMatch_L( short portNum, double *pVal)
HRESULT put_DiffPortMatch_L( short portNum, double newVal)
```

**Interface**
IFixturing2
# DiffPortMatch_R Property

**Description**
Sets the Resistance value of the differential matching circuit.

**VB Syntax**
```
fixture.DiffPortMatch_R(portNum) = value
```

**Variable**
- **(Type)**: A Fixturing object
- **portNum** (Integer): Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.
- **value** (Double): Resistance value in ohms. Choose a value between $-1E18$ to $1E18$.

**Return Type**
Double

**Default**
0

**Examples**
```
fixture.DiffPortMatch_R(2) = 1e3 'Write
value = fixture.DiffPortMatch_R(1) 'Read
```

**C++ Syntax**
```
HRESULT get_DiffPortMatch_R(short portNum, double *pVal)
HRESULT put_DiffPortMatch_R(short portNum, double newVal)
```

**Interface**
IFixturing2
DiffPortMatchMode Property

Description
Sets the differential matching circuit type. To select a user-defined circuit, specify IN ADVANCE the 2-port touchstone filename with DiffPortMatch_UserFilename Property. If you do not specify the appropriate file and you select USER, an error occurs and naNO_CIRCUIT is automatically selected.

VB Syntax
```
fixture.DiffPortMatchMode(pNum) = value
```

Variable (Type) - Description

**fixture**
A Fixturing (object)

**pNum**
(Integer) Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.

**value**
(Enum as NADiffPortMatchCircuitMode) Choose from:
- 0 or naSHUNT_L_SHUNT_C_CIRCUIT - Specifies the circuit that consists of shunt L and shunt C.
- 1 or naUSER_FILE_CIRCUIT - Specifies the user-defined circuit.
- 2 or naNO_CIRCUIT - Specifies no-circuit.

Return Type
Enum

Default
naSHUNT_L_SHUNT_C_CIRCUIT

Examples
```
fixture.DiffPortMatchMode(2) = naNO_CIRCUIT 'Write
value = fixture.DiffPortMatchMode(1) 'Read
```

C++ Syntax
```
HRESULT get_DiffPortMatchMode( short port, tagNADiffPortMatchCircuitMode *eVal)
HRESULT put_DiffPortMatchMode( short port, tagNADiffPortMatchCircuitMode eVal)
```

Interface
IFixturing2
### DiffPortMatchState Property

**Description**
Turns ON or OFF 4-port differential port matching function. Must also set the fixture simulator function to ON using **FixturingState Property**.

**VB Syntax**
```vbnet
fixture.DiffPortMatchState = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td><strong>A Fixturing</strong> (object)</td>
</tr>
<tr>
<td>value</td>
<td><strong>Boolean</strong></td>
</tr>
<tr>
<td></td>
<td><em>False</em> - Turns differential port matching OFF</td>
</tr>
<tr>
<td></td>
<td><em>True</em> - Turns differential port matching ON</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
fixture.DiffPortMatchState = False 'Write
value = fixture.DiffPortMatchState 'Read
```

**C++ Syntax**
```cpp
HRESULT get_DiffPortMatchState( VARIANT_BOOL *pVal)
HRESULT put_DiffPortMatchState( VARIANT_BOOL newVal)
```

**Interface**
IFixturing2
DiffPortMatchUserFilename Property

**Description**
Specifies the 2-port touchstone file in which the information on the user-defined differential matching circuit is saved. Following this command, send DiffPortMatchCircuit Property. If the specified file does not exist, an error occurs when you set the type of differential matching circuit to USER.

**VB Syntax**
```
fixture.DiffPortMatchUserFilename(pNum) = value
```

**Variable (Type) - Description**
- **fixture** (Fixturing object)
- **pNum** (Integer) Balanced (logical) port number. Choose from logical ports 1, 2 or 3. Learn more about logical ports.
- **value** (String) Full path, file name, and extension (.s2P) of the de-embedding circuit. Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

**Return Type**
String

**Default**
Not Applicable

**Examples**
```vbnet
fixture.DiffPortMatchUserFilename(2) = "C:/Program Files/Agilent/Network Analyzer/Documents/myFile.s4p" 'Write

value = fixture.DiffPortMatchUserFilename(1) 'Read
```

**C++ Syntax**
```
HRESULT get_DiffPortMatchUserFilename( short port, BSTR *bstrFile)
HRESULT put_DiffPortMatchUserFilename( short port, BSTR bstrFile)
```

**Interface**
IFixturing2
**DiffZConvPortImag Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets the imaginary part of the impedance value for the differential port impedance conversion function.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>fixture.DiffZConvPortImag(portNum) = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>fixture</code></td>
<td>A <code>Fixturing (object)</code></td>
</tr>
<tr>
<td><code>portNum</code></td>
<td>(Integer) Balanced (logical) port number. Choose from logical ports 1, 2, or 3. <a href="#">Learn more about logical ports.</a></td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Imaginary part of the Impedance value. Choose a value between 0 and 1E18</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Double</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>0</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>fixture.DiffZConvPortImag(2) = 75 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>value = fixture.DiffZConvPortImag(1) 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_DiffZConvPortImag( short portNum, double *pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_DiffZConvPortImag( short portNum, double newVal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IFixturing2</td>
</tr>
</tbody>
</table>
## DiffZConvPortReal Property

**Description**
Sets the imaginary part of the impedance value for the differential port impedance conversion function.

**VB Syntax**
```vbnet
fixture.DiffZConvPortReal(portNum) = value
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing</td>
<td>object</td>
</tr>
<tr>
<td>portNum</td>
<td>(Integer)</td>
<td>Balanced (logical) port number. Choose from logical ports 1, 2, or 3. <a href="#">Learn more about logical ports.</a></td>
</tr>
<tr>
<td>value</td>
<td>(Double)</td>
<td>Real part of the Impedance value. Choose a value between 0 and 1E18</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
See [Differential Port Z Conversion Default](#)

**Examples**
```vbnet
fixture.DiffZConvPortReal(2) = 75 'Write
value = fixture.DiffZConvPortReal(1) 'Read
```

**C++ Syntax**
```c++
HRESULT get_DiffZConvPortReal( short portNum, double *pVal)
HRESULT put_DiffZConvPortReal( short portNum, double newVal)
```

**Interface**
IFixturing2
Write/Read

DiffZConvPortZ0 Property

Description
Sets the impedance value for the differential port impedance conversion function. Set either this single value or set the real and imaginary parts separately. The imaginary part is set to 0.0 using this command.

VB Syntax
`fixture.DiffZConvPortZ0(portNum) = value`

Variable (Type) - Description

- fixture (A Fixturing object)
- portNum (Integer) Balanced (logical) port number. Choose from logical ports 1, 2, or 3. Learn more about logical ports.
- value (Double) Impedance value. Choose a value between 0 and 1E18

Return Type
Double

Default
See Differential Port Z Conversion Default

Examples
```
fixture.DiffZConvPortZ0(2) = 75 'Write
value = fixture.DiffZConvPortZ0(1) 'Read
```

C++ Syntax
```
HRESULT get_DiffZConvPortZ0( short portNum, double *pVal)
HRESULT put_DiffZConvPortZ0( short portNum, double newVal)
```

Interface IFixturing2
**DiffZConvState Property**

**Description**
Turns ON or OFF 4-port differential impedance conversion function. Must also set the fixture simulator function to ON using `FixturingState Property`.

**VB Syntax**
```vbnet
fixture.DiffZConvState = value
```

**Variable**

- `fixture` *(Type)* - Fixturing *(object)*
- `value` *(Boolean)*
  - `False` - Turns differential impedance conversion OFF.
  - `True` - Turns differential impedance conversion ON.

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
fixture.DiffZConvState = False 'Write
value = fixture.DiffZConvState 'Read
```

**C++ Syntax**
```cpp
HRESULT get_DiffZConvState( VARIANT_BOOL *pVal)
HRESULT put_DiffZConvState( VARIANT_BOOL newVal)
```

**Interface**
IFixturing2
### Format Property

**Description**
Sets or returns the display format of the measurement.

**VB Syntax**
`meas.Format = value`

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>meas</th>
<th>A Measurement (object)</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(enum NADataFormat) - Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - naDataFormat_LinMag</td>
</tr>
<tr>
<td></td>
<td>1 - naDataFormat_LogMag</td>
</tr>
<tr>
<td></td>
<td>2 - naDataFormat_Phase</td>
</tr>
<tr>
<td></td>
<td>3 - naDataFormat_Polar</td>
</tr>
<tr>
<td></td>
<td>4 - naDataFormat_Smith</td>
</tr>
<tr>
<td></td>
<td>5 - naDataFormat_Delay</td>
</tr>
<tr>
<td></td>
<td>6 - naDataFormat_Real</td>
</tr>
<tr>
<td></td>
<td>7 - naDataFormat_Imaginary</td>
</tr>
<tr>
<td></td>
<td>8 - naDataFormat_SWR</td>
</tr>
<tr>
<td></td>
<td>9 - naDataFormat_PhaseUnwrapped</td>
</tr>
<tr>
<td></td>
<td>10 - naDataFormat_InverseSmith</td>
</tr>
<tr>
<td></td>
<td>11 - naDataFormat_Kelvin</td>
</tr>
<tr>
<td></td>
<td>12 - naDataFormat_Fahrenheit</td>
</tr>
<tr>
<td></td>
<td>13 - naDataFormat_Celsius</td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
1 - naDataFormat_LogMag

**Examples**

```vbnet
meas.Format = naDataFormat_Real 'Write
fmt = meas.Format 'Read
```

**C++ Syntax**

- `HRESULT get_Format(tagDataFormat *pVal)`
- `HRESULT put_Format(tagDataFormat newVal)`

**Interface**
IMeasurement

---

**Last Modified:**
1-Oct-2007 Added temperature formats
Write-Read  
DisplayAutomationErrors Property

**Description**  Enables or disables automation error messages from being displayed on the screen.

**VB Syntax**  
app.DisplayAutomationErrors = value

**Variable**  
* (Type) - Description

- **app**  An Application (object)
- **value**  (Boolean)
  - **True** allows error to show on display,
  - **False** turns error off from display.

**Return Type**  Boolean

**Default**  True

**Examples**

```vbnet
Dim app As Application
Set app = New Application
app.DisplayAutomationErrors = False 'Turns off display
print app.DisplayAutomationErrors 'prints False
```

**C++ Syntax**  
HRESULT get_DisplayAutomationErrors(VARIANT_BOOL * Val);
HRESULT put_DisplayAutomationErrors(VARIANT_BOOL Val);

**Interface**  IApplication2
### DisplayGlobalPassFail Property

**Description**
Shows or hides the dialog which displays global pass/fail results. [Learn more about Global Pass/Fail.](#)

**VB Syntax**
`app.DisplayGlobalPassFail = value`

**Variable (Type) - Description**
- **app** An **Application** *(object)*
- **value** *(Boolean)*
  - **True** - displays the pass/fail dialog.
  - **False** - hides the pass/fail dialog.

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
Dim app As Application
Set app = New Application
app.DisplayGlobalPassFail = true 'shows dialog
```

**C++ Syntax**
```cpp
HRESULT get_DisplayGlobalPassFail(VARIANT_BOOL * Val);
HRESULT put_DisplayGlobalPassFail(VARIANT_BOOL Val);
```

**Interface**
IApplication6
DisplayRange Property

Description: Sets or returns the range to be displayed on the PNA x-axis. All traces in the channel have this same x-axis scaling.

VB Syntax: `FOM.DisplayRange = value`

Variable (Type) - Description

- object: An FOM (object)
- value: (string) - Range to be displayed on the PNA x-axis. Case insensitive.

Return Type: String

Default: "Receivers"

Examples:

```
fom.DisplayRange = "Source" 'sets the x-axis to the frequency range of "source"
```

```
disprange = fom.DisplayRange 'Read
```

C++ Syntax:

```
HRESULT get_DisplayRange(BSTR *pDspRange)
HRESULT put_DisplayRange(BSTR pDspRange)
```

Interface: IFOM

Last Modified:

7-Mar-2007 Changed to receivers
**Write/Read**

**Distance Property**

**Description**
Set or query marker distance on a time domain trace.

The Write command moves the marker to the specified distance value. Once moved, you can read the **Y axis** value or **read the X-axis time** value. (Distance is calculated from the X-axis time value.)

The Read command reads the distance of the marker.

If the marker is set as delta, the WRITE and READ data is relative to the reference marker.

**VB Syntax**

```
mark.Distance = value
```

**Variable**

*Type* - Description

- **mark**
  - A **Marker** *(object)*

- **value**
  - **(double)** - Marker distance in the unit of measure specified with **DistanceMarkerUnit Property**

**Return Type**

Double

**Default**

Not Applicable

**Examples**

```
mark.Distance = 3e9 'Write
XVal = mark.Distance 'Read
```

**C++ Syntax**

```
HRESULT get_Distance(double *pVal);
HRESULT put_Distance(double newVal);
```

**Interface**

IMarker2
## DistanceMarkerMode Property

### Description
Specifies the measurement type in order to determine the correct marker distance.

- Select Auto for S-Parameter measurements.
- Select Reflection or Transmission for arbitrary ratio or unratioed measurements.

This setting affects the display of all markers for only the active measurement.

### VB Syntax
```
trans.DistanceMarkerMode = value
```

### Variable *(Type)* - Description

- **trans** *(Transform)* *(object)*
- **value** *(enum As NADistanceMarkerMode)* - Choose from:
  0 - naDistanceMarkerModeAuto
  1 - naDistanceMarkerModeReflection
  2 - naDistanceMarkerModeTransmission

### Return Type
Enum

### Default
0 - naDistanceMarkerModeAuto

### Examples
```
trans.DistanceMarkerMode = naDistanceMarkerModeReflection 'Write

DMM = trans.DistanceMarkerMode 'Read
```

### C++ Syntax
```
HRESULT get_DistanceMarkerMode(tagNADistanceMarkerMode *pVal);
HRESULT put_DistanceMarkerMode(tagNADistanceMarkerMode newVal);
```

### Interface
ITransform2
DistanceMarkerUnit Property

Description: Specifies the unit of measure for the display of marker distance values. This setting affects the display of ALL markers for only the ACTIVE measurement (unless Distance Marker Units are coupled using CoupledParameters Property).

VB Syntax: `trans.DistanceMarkerUnit = value`

Variable: (Type) - Description
- `trans`: A Transform (object)
- `value`: (Enum As NADistanceMarkerUnit) - Distance Marker Units. Choose from:
  - 0 - naDistanceMarkerUnitMeter
  - 1 - naDistanceMarkerUnitFeet
  - 2 - naDistanceMarkerUnitInch

Return Type: Enum

Default: 0 - naDistanceMarkerUnitMeter

Examples: `trans.DistanceMarkerUnit = naDistanceMarkerUnitFeet 'sets the...`
```
U = trans.DistanceMarkerUnit 'Read
```

C++ Syntax: HRESULT get_DistanceMarkerUnit(tagNADistanceMarkerUnit *pVal);
HRESULT put_DistanceMarkerUnit(tagNADistanceMarkerUnit newVal);

Interface: ITransform2
**Divisor Property**

**Description**
Sets and returns the Divisor value to be used when coupling this range to the primary range.
This setting is valid only if the specified range is coupled to the primary range.

**VB Syntax**
```vbnet
FOMRange.Divisor = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>An FOMRange (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Double) - Divisor value - (Unitless)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**
```vbnet
fomRange.Divisor = .5  'Write
Div = fomRange.Divisor  'Read
```

**C++ Syntax**
```cpp
HRESULT get_Divisor(double *pVal)
HRESULT put_Divisor(double *pVal)
```

**Interface**
IFOMRange
Read-Write

Do1PortEcal Property - Superseded

Description  **Note:** This command is replaced with [CalKitType](#) which sets the ECal module or mechanical cal kit.
Specify ECAL or Mechanical calibration for the mixer characterization portion of a VMC calibration.

VB Syntax  

```
VMC.Do1PortEcal = bool
```

Variable  

**Type** - Description

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>VMC</td>
<td>VMCType (object)</td>
</tr>
<tr>
<td>bool</td>
<td>(Boolean)</td>
</tr>
<tr>
<td>True</td>
<td>- ECAL</td>
</tr>
<tr>
<td>False</td>
<td>- Mechanical</td>
</tr>
</tbody>
</table>

Return Type  Boolean

Default  False

Examples  

```
value = VMC.Do1PortEcal
```

C++ Syntax  

```
HRESULT put_Do1PortEcal(VARIANT_BOOL bDoEcal);
HRESULT get_Do1PortEcal(VARIANT_BOOL *bDoEcal);
```

Interface  VMCType
Read-Write

Do2PortEcal Property - Superseded

Description  Note: This command is replaced with CalKitType which sets the ECal module or mechanical cal kit.

Specify ECAL or Mechanical calibration. For VMC, this selection only applies to the 2-port calibration portion. For mixer characterization (VMC), use Do1PortEcal Property

VB Syntax  object.Do2PortEcal = bool

Variable  (Type) - Description

object  SMCTYPE (object) or VMCTYPE (object)

bool  (Boolean)

True - ECAL
False - Mechanical

Return Type  Boolean

Default  False

Examples  value = VMC.Do2PortEcal

C++ Syntax  HRESULT put_Do2PortEcal(VARIANT_BOOL bDoEcal);
            HRESULT get_Do2PortEcal(VARIANT_BOOL *bDoEcal);

Interface  SMCTYPE
            VMCTYPE
## Domain Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the domain (frequency, time, power, phase) of the measurement. To understand how this property is useful, see <a href="#">IMeasurement2 Interface</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>value = meas.Domain</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><em>value</em></td>
<td><em>(Enum as NADomainType)</em> - variable to store the returned value</td>
</tr>
<tr>
<td></td>
<td>0 - naDomainFrequency</td>
</tr>
<tr>
<td></td>
<td>1 - naDomainTime</td>
</tr>
<tr>
<td></td>
<td>2 - naDomainPower</td>
</tr>
<tr>
<td></td>
<td>4 - naDomainPhase</td>
</tr>
<tr>
<td><strong>meas</strong></td>
<td>A <a href="#">Measurement</a> <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Enum as NADomainType</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>Print meas.Domain</code> 'prints the value of the domain enum'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_Domain(tagNADomainType * Val);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>I_measurement2</td>
</tr>
</tbody>
</table>

Last Modified: 3-May-2011    Updated for phase
### Driver Property

**Description**
Sets and returns the external device driver (model).

**VB Syntax**
```
extDevices.Driver = value
```

**Variable**

<table>
<thead>
<tr>
<th><strong>Type</strong></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>extDevices</code></td>
<td>An <code>ExternalDevice</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(String) External device driver (model). Choose from the following:</td>
</tr>
<tr>
<td></td>
<td>&quot;AGPM&quot; for all power meters.</td>
</tr>
<tr>
<td></td>
<td>See a list of supported external source drivers.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
"AGGeneric"

**Examples**
```
extDevices.Driver = "AGESG"  'Write
value = extDevices.Driver    'Read
See example program to configure PMAR device
See example program to configure External Source
```

**C++ Syntax**
```
HRESULT get_Driver( BSTR* value);
HRESULT put_Driver( BSTR newVal);
```

**Interface**
`IExternalDevices`

---

**Last Modified:**
- 15-Jan-2010 Added 'AG" to example
- 31-Jul-2009 MX New topic
### DspFpgaRevision Property

**Description**
Returns the DSP FPGA Revision number that is visible in the Help, About Network Analyzer dialog box. [Learn more.](#)

**VB Syntax**
\[ value = cap.DspFpgaRevision \]

**Variable**
- **Type** - Description
  - value (String) - Variable to store the returned DSP revision number.
  - cap A [Capabilities](#) object

**Return Type**
String

**Default**
Not Applicable

**Examples**
\[ value = cap.DspFpgaRevision 'Read \]

**C++ Syntax**
HRESULT get_DspFpgaRevision(BSTR *value);

**Interface**
ICapabilities6

---

Last Modified:

30-Nov-2009    MX New topic
**DspRevision Property**

**Description**
Returns the DSP Revision number that is visible in the Help, About Network Analyzer dialog box. Learn more.

**VB Syntax**
```vbnet
value = cap.DspRevision
```

**Variable**

- **value** (String) - Variable to store the returned DSP revision number.
- **cap** A **Capabilities** *(object)*

**Return Type**
String

**Default**
Not Applicable

**Examples**
```vbnet
value = cap.DspRevision 'Read
```

**C++ Syntax**
```cpp
HRESULT get_DspRevision(BSTR *value);
```

**Interface**
ICapabilities6
## DUTTopology Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the device topology setting.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>balTopology.DUTTopology = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>balTopology</code></td>
<td>A <code>BalancedTopology</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(enum NADUTTopology) - Choose either:</strong></td>
</tr>
<tr>
<td></td>
<td>0 naSEBal: Single-Ended - Balanced measurement</td>
</tr>
<tr>
<td></td>
<td>1 naSESEBal: Single-Ended - Single-Ended - Balanced measurement</td>
</tr>
<tr>
<td></td>
<td>2 naBalBal: Balanced - Balanced measurement</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Enum as NADUTTopology</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>naSEBal</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>balTop.DUTTopology = naSESEBal 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>DutTop = balTop.DUTTopology 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_DUTTopology(tagNADUTTopology* pVal)</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_DUTTopology(tagNADUTTopology newVal)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IBalancedTopology</td>
</tr>
</tbody>
</table>
**DwellTime Property**

**Description**
Sets or returns the dwell time at the start of each sweep point for all measurements in a channel. Dwell time is only available with Chan. `SweepGenerationMode` = `naSteppedSweep` (not `naAnalogSweep`).

Sets or returns the dwell time of a specified sweep segment.

**VB Syntax**
```
object.DwellTime = value
```

**Variable**

<table>
<thead>
<tr>
<th><strong>(Type)</strong></th>
<th><strong>- Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>object</strong></td>
<td>A Channel (object)</td>
</tr>
<tr>
<td><strong>or</strong></td>
<td>CalSet (object) - Read-only property</td>
</tr>
<tr>
<td><strong>or</strong></td>
<td>Segment (object)</td>
</tr>
</tbody>
</table>

**value** (double) - Dwell Time in seconds. Choose any number between 0 and 86400

**Return Type**
Double

**Default**
0

**Examples**
```
chan.DwellTime = 3e-3 'sets the dwell time for the channel -Write
segs(3).DwellTime = 1e9 'sets the dwell time of segment 3 -Write
dwell = chan.DwellTime 'Read
```

**C++ Syntax**
```
HRESULT get_DwellTime(double *pVal)
HRESULT put_DwellTime(double newVal)
```

**Interface**
| IChannel |
| ISegment |
| CalSet3 |
### DwellPerPoint Property

**Description**
Sets and returns the amount of time the PNA should wait after for an external source to settle before making a measurement at each data point.

**VB Syntax**
```
extSource.DwellPerPoint = value
```

**Variable (Type) - Description**
- `chan` An `ExternalSource` (object)
- `value` (Double) Dwell time in milliseconds.

**Return Type**
Double

**Default**
3

**Examples**
```
extSource.DwellPerPoint = 10 'Write

dpp = extSource.DwellPerPoint 'Read
```

**C++ Syntax**
- HRESULT `get_DwellPerPoint (tagNAExtDevDwellPerPoint *pValue)`
- HRESULT `put_DwellPerPoint (tagNAExtDevDwellPerPoint newVal)`

**Interface**
IEternalSource
ECALCharacterization Property - Superseded

Description: This command is replaced with CalKitType which sets the ECal module and User Characterization.

Specifies the characterization data within an ECal module to be used for the SMC calibration.

Learn more about ECal User Characterization.

VB Syntax: \texttt{SMC.ECALCharacterization(mod) = value}

Variable (Type) - Description

- \texttt{SMC SMCTYPE (object)}
- \texttt{module 1 - ECAL module}
- \texttt{value (Long) – Characterization data within the ECAL module to be used for ECAL operations. Choose from:}
  - \texttt{0 – Factory Characterization}
  - \texttt{1 – UserCharacterization1}
  - \texttt{2 – UserCharacterization2}
  - \texttt{3 – UserCharacterization3}
  - \texttt{4 – UserCharacterization4}
  - \texttt{5 – UserCharacterization5}

Return Type: Long

Default: 0 - Factory Characterization

Examples: \texttt{SMC.ECALCharacterization(1) = 2}

C++ Syntax: HRESULT put_ECALCharacterization( long moduleNumber, long characterization); HRESULT get_ECALCharacterization( long moduleNumber, long* characterization);

Interface: ISMCType
**Write/Read**

**ECALCharacterization Property - Superseded**

### Description

*Note:* This command is replaced with `CalKitType` which sets the ECal module and User Characterization.

Specifies the characterization data within an ECal module to be used, and the portion of the VMC calibration. Learn more about [ECal User Characterization](#).

### VB Syntax

```vbnet
VMC.ECALCharacterization (module, port) = value
```

### Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VMC</td>
<td><code>VMCTYPE</code> (object)</td>
<td></td>
</tr>
<tr>
<td>module</td>
<td>(long integer)</td>
<td>1 - ECAL module</td>
</tr>
</tbody>
</table>
| port     | (boolean)       | **True** - 2-port calibration portion of the VMC  
**False** - 1-port (mixer characterization portion of the VMC cal) |
| value    | (Long)          | Characterization data within the ECal module to be used for ECal operations. Choose from:  
0 – Factory Characterization  
1 – UserCharacterization1  
2 – UserCharacterization2  
3 – UserCharacterization3  
4 – UserCharacterization4  
5 – UserCharacterization5 |

### Return Type

Long

### Default

0 - Factory Characterization

### Examples

`VMC.ECALCharacterization (1,True) = 4`

### C++ Syntax

```c++
HRESULT put_ECALCharacterization( long moduleNumber, long characterization);
HRESULT get_ECALCharacterization( long moduleNumber, long* characterization);
```

### Interface

`IVMCTYPE`
Write/Read

ECALCharacterizationEx Property

Description
This property replaces ECALCharacterization Property. Specifies the characterization data within an ECal module to be used for the calibration. Learn more about ECal User Characterization.

VB Syntax
```vbnet
cal.ECALCharacterizationEx (module) = value
```

Variable (Type) - Description
- **cal**: Calibrator (object)
- **module**: (long integer) Optional argument. ECal module. Choose from modules 1 through 8
  - Use IsECALModuleFoundEx to determine the number of modules connected to the PNA
  - Use GetECALModuleInfoEx to returns the model and serial number of each module.
- **value**: (Long) – Characterization data within the ECal module to be used for ECal operations. Choose from:
  - 0 – Factory Characterization
  - 1 – UserCharacterization1
  - 2 – UserCharacterization2
  - ..and so forth up to...
  - 12 – UserCharacterization12

Return Type
Long

Default
0 - Factory Characterization

Examples
```vbnet
cal.ECALCharacterizationEx (4) = 2
```

C++ Syntax
```c++
HRESULT put_ECALCharacterizationEx( long moduleNumber, long characterization);
HRESULT get_ECALCharacterizationEx( long moduleNumber, long* characterization);
```

Interface
ICalibrator4

Last Modified:
15-Jun-10 Updated for 12 User chars
### ECalID Property

**Description**
Selects the model and serial number of the ECal module to be characterized. This command does NOT set the model and serial number of the ECal module.

**VB Syntax**

```vbnet
userChar.ECalID = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>userChar</td>
<td>An <strong>ECalUserCharacterizer</strong> Object</td>
</tr>
<tr>
<td>value</td>
<td>(String) Model and serial number of the ECal module to be characterized.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
"" (Empty String)

**Examples**

```vbnet
userChar.ECalID = "N4433A,00001"
```

**C++ Syntax**

```cpp
HRESULT put_ECalID(BSTR id);
```

**Interface**
IECalUserCharacterizer

---

Last Modified:

2-Nov-2008  New topic (8.33)
ECAL Isolation Property

**Description**

*Note:* The inherent isolation of the PNA is better than that attained with this command. ONLY use this command when using an external test set, and ONLY using a 8509x ECal module.

Specifies whether the acquisition of the ECal calibration should include isolation or not.

**VB Syntax**

```
cal.ECALIsolation = value
```

**Variable**

*Type* - Description

- `cal` A Calibrator *(object)*
- `value` *(boolean)*
  - `False` - Exclude Isolation
  - `True` - Include Isolation

**Return Type**

Boolean

**Default**

False

**Examples**

```vbnet
Dim oPNA as AgilentPNA835x.Application
Dim oCal as Calibrator
Set oPNA = CreateObject("AgilentPNA835x.Application", "MachineName")
Set oCal = oPNA.ActiveChannel.Calibrator
' Uncomment the following line to have the cal include isolation
' oCal.ECALIsolation = True
' Uncomment the following line to have the cal omit isolation
' oCal.ECALIsolation = False
oCal.DoECAL2Port ' Do the cal
```

**C++ Syntax**

```cpp
HRESULT put_ECALIsolation ( VARIANT_BOOL bIsolationState );
HRESULT get_ECALIsolation ( VARIANT_BOOL *bIsolationState );
```

**Interface**

Calibrator

---

**Last Modified:**

16-Apr-2007 Un-obsoleted
Read-only

**ECALModuleNumberList Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns a list of index numbers to be used for referring to the ECal modules that are currently attached to the PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>clist = cal.ECALModuleNumberList</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>- clist</code></td>
<td>Variable to store the returned list of index numbers.</td>
</tr>
<tr>
<td><code>- cal</code></td>
<td><strong>Calibrator</strong> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Variant</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>clist = cal.ECALModuleNumberList</code></td>
</tr>
<tr>
<td></td>
<td>'If 2 modules are attached to the PNA`</td>
</tr>
<tr>
<td></td>
<td>'then the returned list will be: <code>1,2</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_ECALModuleNumberList(VARIANT *modules);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>ICalibrator6</code></td>
</tr>
</tbody>
</table>
### EcalOrientation Property

**Description**
Specifies which port of the ECAL module is connected to which port of the PNA when the `AutoOrient` property = False.

**VB Syntax**

```vb
SMC.EcalOrientation (mod) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>SMC</code></td>
<td><code>SMCType</code> (object)</td>
</tr>
<tr>
<td><code>mod</code></td>
<td>(Long)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(string) - Format this parameter in the following manner:</td>
</tr>
<tr>
<td></td>
<td>Aw,Bx,Cy,Dz</td>
</tr>
<tr>
<td></td>
<td>where</td>
</tr>
<tr>
<td></td>
<td>• A, B, C, and D are literal ports on the ECAL module</td>
</tr>
<tr>
<td></td>
<td>• w,x,y, and z are substituted for PNA port numbers to which the ECAL module port is connected.</td>
</tr>
</tbody>
</table>

Ports of the module which are not used are omitted from the string.

For example, on a 4-port ECal module with

- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1

the string would be: A2,B3,D1

**Return Type**
String

**Default**
"A1,B2"

**Examples**

```vb
SMC.EcalOrientation (1) = "A2,B1"
```

**C++ Syntax**

```cpp
HRESULT put_EcalOrientation(long lModuleNum, BSTR orientation);
HRESULT get_EcalOrientation(long lModuleNum, BSTR *orientation);
```
<table>
<thead>
<tr>
<th>Interface</th>
<th>SMCType</th>
</tr>
</thead>
</table>

975
Read/Write

EcalOrientation1Port Property

**Description**: For Mixer Characterization ONLY. Specifies which port of the ECal module is connected to which port of the PNA for the **Do1PortECAL** property when the **AutoOrient** property = False.

**VB Syntax**: `VMC.EcalOrientation1Port (mod) = value`

**Variable** *(Type)* - Description

*VMC* `VMCType (object)`

*mod* *(Long)*

1 - Use ECAL Module for the calibration.

*value* *(string)* - Choose from:

"A1" - ECAL module port A is connected to PNA port 1

"B1" - ECAL module port A is connected to PNA port 1

**Return Type**: String

**Default**: "A1"

If anything other than port 1 is specified, "B1" will be used. For example, if "A2" is specified, "B1" is used.

**Examples**: `VMC.EcalOrientation1Port(1) = "B1"`

**C++ Syntax**: `HRESULT put_EcalOrientation1Port(long lModuleNum, BSTR orientation);`  
`HRESULT get_EcalOrientation1Port(long lModuleNum, BSTR`
Read/Write

EcalOrientation2Port Property

Description
Specifies which port of the ECal module is connected to which port of the PNA for the Do2PortECAL property when the AutoOrient property = False.

VB Syntax
VMC.EcalOrientation2Port (mod) = value

Variable (Type) - Description
VMC VMCTYPE (object)

mod (Long) Module being used for the calibration.
Choose from 1 or 2.

value (string) - Format this parameter in the following manner:
Aw,Bx,Cy,Dz
where

- A, B, C, and D are literal ports on the ECAL module
- w,x,y, and z are substituted for PNA port numbers to which the ECAL module port is connected.

Ports of the module which are not used are omitted from the string.
For example, on a 4-port ECal module with

- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1

the string would be: A2,B3,D1

Return Type String

Default "A1,B2"

Examples
VMC.EcalOrientation1Port(1) = "A2,B1"

C++ Syntax
HRESULT put_EcalOrientation2Port(long lModuleNum, BSTR orientation);
HRESULT get_EcalOrientation2Port(long lModuleNum, BSTR *orientation);
| Interface | VMCType |
**ECALPortMapEx Property**

**Description**
This property replaces ECALPortMap Property. Specifies which ports of the ECal module are connected to which ports of the PNA for the DoECAL1PortEx and DoECAL2PortEx methods when the OrientECALModule property = False.

This setting remains until the PNA is restarted or this command is sent again.

**Note:** For guided calibrations where Orient is OFF and the same ECal module is used in more than one Connection Step, you are not allowed to specify how the ECal module is connected. Instead, the PNA determines the orientation. The PNA does not verify that you made the connection properly.

This command, and OrientECALModule_Property, can be used to perform ECal orientation using the Guided Calibration interface.

**VB Syntax**
```
cal.ECALPortMapEx (module) = value
```

**Variable (Type) - Description**
- **cal** A Calibrator (object)
- **module** (long integer) Optional argument. ECal module. Choose from modules 1 through 8
  - Use IsECALModuleFoundEx to determine the number of modules connected to the PNA
  - Use GetECALModuleInfoEx to return the model and serial number of each module.
- **value** (string) - Format this parameter in the following manner:
  - Aw,Bx,Cy,Dz
  - where
    - A, B, C, and D are literal ports on the ECAL module
    - w,x,y, and z are substituted for PNA port numbers to which the ECAL module port is connected.

Ports of the module which are not used are omitted from the string.

For example, on a 4-port ECal module with
- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1
the string would be: A2,B3,D1

DoECAL1PortEx or DoECAL2PortEx methods will fail if the port numbers passed to those methods are not in the string of this property and OrientECALModule property = False.

**Return Type**  String

**Default**  Not Applicable

**Examples**

```vbs
Dim cal As Calibrator
Dim sPortMap As String
Set cal = PNAapp.ActiveChannel.Calibrator
cal.ECALPortMapEx = "a2,b1" 'Write
sPortMap = cal.ECALPortMap 'Read
```

**C++ Syntax**

```cpp
HRESULT put_ECALPortMapEx( long moduleNumber, BSTR strPortMap);
HRESULT get_ECALPortMapEx( long moduleNumber, BSTR *strPortMap);
```

**Interface**  ICalibrator4

Last Modified:

7-May-2007  Added note about orient
### ElecDelayMedium Property

#### Description
Sets or returns the electrical delay medium.

#### VB Syntax
```vbnet
meas.ElecDelayMedium = value
```

#### Variable *(Type) - Description*
- **meas**: A Measurement *(object)*
- **value**: *(enum NACalStandardMedium)* choose from
  - 0 - naCoax
  - 1 - naWaveGuide

#### Return Type
NACalStandardMedium

#### Default
Not Applicable

#### Examples
```vbnet
Print meas.ElecDelayMedium 'prints the value of the electrical
delay medium
```

#### C++ Syntax
```cpp
HRESULT get_ElecDelayMedium(tagNACalStandardMedium *pVal);
HRESULT put_ElecDelayMedium(tagNACalStandardMedium newVal);
```

#### Interface
IMeasurement2
### ElecDistanceDelay Property

**Description**
Sets the electrical delay in physical length (distance) for the selected measurement.

**VB Syntax**

```
meas.ElecDistanceDelay = value
```

**Variable (Type) - Description**

- **meas**
  A Measurement (object)

- **value**
  (double) - Electrical Delay in distance. Set units using ElecDistanceDelayUnit.

**Return Type**
Double

**Default**
0

**VB Examples**
```
meas.ElecDistanceDelay = 1e-3 'Write
edelay = meas.ElecDistanceDelay 'Read
```

**C# Examples**
```
Meas.ElecDistanceDelay = 1e-3 'Write

// This property returns an object, and the object must be cast to a double to access the value.
Edelay = (double)meas.ElecDistanceDelay 'read
```

**C++ Syntax**

```
HRESULT get_ElecDistanceDelay(VARIANT *pVal)
HRESULT put_ElecDistanceDelay(VARIANT newVal)
```

**Interface**
IMeasurement11

---

Last Modified:
- 12-Nov-2010  Added C# example
- 6-Feb-2009   MX New topic
### ElecDistanceDelayUnit Property

**Description**
Sets and returns the units for specifying electrical delay in physical length (distance).

**VB Syntax**
```vbnet
meas.ElecDistanceDelayUnit = value
```

**Variable**
- **meas** (Type) - Description
  - A `Measurement` (object)
- **value** (enum `naDistanceUnit`) Choose from:
  - 0 – `naDistanceUnit_Meter`
  - 1 – `naDistanceUnit_Feet`
  - 2 – `naDistanceUnit_Inch`

**Return Type**
Enum

**Default**
0 – `naDistanceUnit_Meter`

**Examples**
```vbnet
meas.ElecDistanceDelayUnit = naDistanceUnit_Meter 'Write

edelay = meas.ElecDistanceDelayUnit 'Read
```

**C++ Syntax**
```csharp
HRESULT get_ElecDistanceDelayUnit(tagNADistanceUnit *pVal)
HRESULT put_ElecDistanceDelayUnit(tagNADistanceUnit newVal)
```

**Interface**
`IMeasurement11`

---

**Last Modified:**
6-Feb-2009   MX New topic
**ElectricalDelay Property**

**Description**
Sets the Electrical Delay for the active channel.

**VB Syntax**
```
meas.ElectricalDelay = value
```

**Variable**
- **meas** - A Measurement (object)
- **value** - (double) - Electrical Delay in seconds. Choose any number between -9.99 and 9.99

**Return Type**
Double

**Default**
0

**Examples**
```
meas.ElectricalDelay = 1e-3 'Write

edelay = meas.ElectricalDelay 'Read
```

**C++ Syntax**
```
HRESULT get_ElectricalDelay(double *pVal)
HRESULT put_ElectricalDelay(double newVal)
```

**Interface**
IMeasurement
Element Property

Description
Returns a handle to the specified PathElement object. Each element object contains a unique set of values.

The Value Property is used to set the value for each element.
See a list of configurable elements and values for various PNA models.

VB Syntax
Set elem = pathConfig.Element (element)

Variable (Type) - Description

elem (Object) IPathElement

pathConfig A PathConfiguration (object)

element (String) Configurable element. Use pathConfig.Elements to return a list of configurable elements or see a list of configurable elements for various PNA models.

Return Type Object

Default Not Applicable

Examples
Dim elem as PathElement
Set elem = app.ActiveChannel.PathConfiguration.Element("Src1")

C++ Syntax
HRESULT Element(BSTR elemName, IPathElement** ppElement);

Interface IPathConfiguration

Last Modified:
14-Dec-2006 MX New topic
**Elements Property**

**Description**
Returns an array containing the names of configurable elements.

See a list of configurable elements and settings for various PNA models.

**VB Syntax**
```vbnet
values = pathConfig.Elements
```

**Variable**

- **values** *(Variant array)* Variable to store the array of configurable elements.

- **pathConfig** A PathConfiguration *(object)*

**Return Type**
Variant array

**Default**
Not Applicable

**Examples**
```vbnet
elems = pathConfig.Elements
```

**C++ Syntax**
```cpp
HRESULT Elements( VARIANT* pElements );
```

**Interface**
IPathConfiguration

---

Last Modified:

14-Dec-2006 MX New topic
**Embed4PortA Property**

**Description**
Returns the PNA port number associated with 'a' based on the device topology.

To see 'a' for all topologies, and to specify the port connections, use [Embed4PortList Property](#).

Specify topology using [Embed4PortTopology Property](#).

**VB Syntax**
```vbnet
default_value = fixture.Embed4PortA
```

**Variable**
- **value** (Type): Short Integer
  - Description: Variable to store the returned PNA port number.
- **fixture** (Type): Fixturing (object)

**Return Type**
Integer

**Default**
Not Applicable

**Examples**
```vbnet
default_value = fixture.Embed4PortA
```

**C++ Syntax**
```cpp
HRESULT get_Embed4PortA(short *portA);
```

**Interface**
IFixturing2
## Embed4PortB Property

**Description**
Returns the PNA port number associated with 'b' based on the device topology. To see 'b' for all topologies, and to specify the port connections, use Embed4PortList Property.

Specify topology using Embed4PortTopology Property.

**VB Syntax**
```
value = fixture.Embed4PortB
```

**Variable**
- **(Type)**: Short Integer
  - **Description**: Variable to store the returned PNA port number.

**fixture**
- A Fixturing (object)

**Return Type**
Integer

**Default**
Not Applicable

**Examples**
```
value = fixture.Embed4PortB 'Read
```

**C++ Syntax**
```
HRESULT get_Embed4PortB(short *portB );
```

**Interface**
IFixturing2
**Embed4PortC Property**

**Description**
Returns the PNA port number associated with 'c' based on the device topology.

To see 'c' for all topologies, and to specify the port connections, use **Embed4PortList Property**.

Specify topology using **Embed4PortTopology Property**

**VB Syntax**

```vbnet
value = fixture.Embed4PortC
```

**Variable (Type) Description**

- **value** (Short Integer) Variable to store the returned PNA port number.

- **fixture** A **Fixturing** (object)

**Return Type**
Integer

**Default**
Not Applicable

**Examples**

```vbnet
value = fixture.Embed4PortC 'Read
```

**C++ Syntax**

```cpp
HRESULT get_Embed4PortC(short *portC);
```

**Interface**
IFixturing2
Embed4PortD Property

Description
Returns the PNA port number associated with 'd' based on the device topology.
To see 'd' for all topologies, and to specify the port connections, use Embed4PortList Property.
Specify topology using Embed4PortTopology Property.

VB Syntax
value = fixture.Embed4PortD

Variable (Type) - Description
value  (Short Integer) Variable to store the returned PNA port number.
fixture  A Fixturing (object)

Return Type
Integer

Default
Not Applicable

Examples
value = fixture.Embed4PortD  'Read

C++ Syntax
HRESULT get_Embed4PortD(short *portD );

Interface
IFixturing2
**Description**

Specifies the PNA port connections for ALL topologies. The port assignment is dependent on the DUT topology. All four port numbers are required. However, for:

- Topology A, only the first **two** arguments are valid,
- Topology B, only the first **three** arguments are valid,
- Topology C, **ALL** arguments are valid.

Specify topology using **Embed4PortTopology Property**.

Read the port assignments using the following commands. A, B, C, and D, refer to the port; NOT the topology.

- **Embed4PortA Property**
- **Embed4PortB Property**
- **Embed4PortC Property**
- **Embed4PortD Property**

![Topology A Diagram](image1)

![Topology B Diagram](image2)
### VB Syntax

```vbnet
fixture.Embed4PortList = p1, p2, p3, p4
```

### Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td><em>Fixturing</em> (object)</td>
<td></td>
</tr>
<tr>
<td>p1</td>
<td></td>
<td>PNA Port number assigned to a in above graphic.</td>
</tr>
<tr>
<td>p2</td>
<td></td>
<td>PNA Port number assigned to b in above graphic.</td>
</tr>
<tr>
<td>p3</td>
<td></td>
<td>PNA Port number assigned to c in above graphic.</td>
</tr>
<tr>
<td>p4</td>
<td></td>
<td>PNA Port number assigned to d in above graphic.</td>
</tr>
</tbody>
</table>

### Return Type

Four Integers

### Default

1, 2, 3, 4

### Examples

```vbnet
fixture.4PortNetworkTopoCPorts = 4, 3, 2, 1 'Write
```

### C++ Syntax

```cpp
HRESULT put_4PortNetworkTopoCPorts(short ChannelNum, short pPortA, short pPortB, short pPortC, short pPortD)
```

### Interface

IFxturing2
**Embed4PortNetworkFilename Property**

**Description**
Specifies the 4-port touchstone file (*.s4p) in which the network to embed or de-embed resides. If the specified file does not exist, an error occurs when type command is sent. Following this command, send **Embed4PortNetworkMode Property**.

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**
```vbnet
fixture.Embed4PortNetworkFilename(netNum) = value
```

**Variable (Type) - Description**

- `fixture` A Fixturing (object)
- `netNum` (Integer) Network position. Choose from 1 or 2. See **Embed4PortTopology Property**
- `value` (String) Full path, file name, and extension (.s4P) of the circuit.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

**Return Type**
String

**Default**
Not Applicable

**Examples**
```vbnet
fixture.Embed4PortNetworkFilename(2) = "C:/Program Files/Agilent/Network Analyzer/Documents/myFile.s4p" 'Write
```

```vbnet
value = fixture.Embed4PortNetworkFilename(1) 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Embed4PortNetworkFilename( short networkNum, BSTR *filename);
HRESULT put_Embed4PortNetworkFilename( short networkNum, BSTR filename);
```

**Interface**
IFixturing2
Write/Read

Embed4PortNetworkMode Property

Description
Specify the type of processing to take place on the specified 4-port network. First specify the network filename with FSim.Embed4PortNetworkFilename Property.

VB Syntax
fixture.Embed4PortNetworkMode(netNum) = value

Variable (Type) - Description
fixture A Fixturing (object)

netNum (Integer) Network position. Choose from 1 or 2. See Embed4PortTopology Property

value (Enum as NA4PortEmbedNetworkMode) Processing mode. Choose from:

- 0 or naNO_NETWORK - The same as disabling.
- 1 or naEMBED_NETWORK - Add Network circuit.
- 2 or naDEEMBED_NETWORK - Remove Network circuit

Return Type
Enum

Default naNO_NETWORK

Examples
fixture.Embed4PortNetworkMode(1) = naNO_NETWORK 'Write

value = fixture.Embed4PortNetworkMode(2) 'Read

C++ Syntax
HRESULT get_Embed4PortNetworkMode( short networkNum, tagNA4PortEmbedNetworkMode *eVal );
HRESULT put_Embed4PortNetworkMode( short networkNum, tagNA4PortEmbedNetworkMode eVal );

Interface IFixturing2
**Embed4PortState Property**

**Description**  
Turns ON or OFF 4-port Network embedding for all ports on the channel.

**VB Syntax**  
`fixture.Embed4PortState = value`

**Variable**  
(Type) - Description

*fixture*  
A `Fixturing` (object)

*value*  
(Boolean)

- **False** - Turns Embedding OFF
- **True** - Turns Embedding ON

**Return Type**  
Boolean

**Default**  
False (OFF)

**Examples**  
```
fixture.Embed4PortState = False  'Write
value = fixture.Embed4PortState  'Read
```

**C++ Syntax**  
```
HRESULT get_Embed4PortState(VARIANT_BOOL *pVal)
HRESULT put_Embed4PortState(VARIANT_BOOL newVal)
```

**Interface**  
IFixturing2
### Embed4PortTopology Property

**Description**
Specifies the PNA / DUT topology. [Learn more about these and other PNA/DUT configurations.](#)

**VB Syntax**
```vbnet
fixture.Embed4PortTopology = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Enum as NA4PortEmbedTopology) PNA / DUT topology. Choose from:</td>
</tr>
<tr>
<td>0</td>
<td>or naTOPOLOGY_A - 2 PNA/DUT Ports</td>
</tr>
<tr>
<td>1</td>
<td>or naTOPOLOGY_B - 3 PNA/DUT Ports</td>
</tr>
<tr>
<td>2</td>
<td>or naTOPOLOGY_C - 4 PNA/DUT Ports</td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
naTOPOLOGY_A (2 PNA/DUT Ports)

**Examples**
```vbnet
fixture.Embed4PortTopology = naTOPOLOGY_A 'Write
value = fixture.Embed4PortTopology 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Embed4PortTopology( tagNA4PortEmbedTopology *eVal );
HRESULT put_Embed4PortTopology( tagNA4PortEmbedTopology eVal );
```

**Interface**
IFixturing2
## Enable Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Turns ON / OFF the trigger output.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>auxTrig.Enable = state</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>auxTrig</code></td>
<td>An <strong>AuxTrigger (object)</strong></td>
</tr>
<tr>
<td><code>state</code></td>
<td><strong>(boolean)</strong></td>
</tr>
<tr>
<td></td>
<td><code>True</code> - Trigger Output ON</td>
</tr>
<tr>
<td></td>
<td><code>False</code> - Trigger Output OFF</td>
</tr>
<tr>
<td>Return Type</td>
<td>Boolean</td>
</tr>
<tr>
<td>Default</td>
<td>False</td>
</tr>
<tr>
<td>Examples</td>
<td><code>auxTrig.Enable = True</code> <em>'Write'</em></td>
</tr>
<tr>
<td></td>
<td><code>value = auxTrig.Enable</code> <em>'Read'</em></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_Enable(VARIANT_BOOL * enable);</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Enable(VARIANT_BOOL enable);</td>
</tr>
<tr>
<td>Interface</td>
<td>IAuxTrigger</td>
</tr>
</tbody>
</table>

Last Modified:

14-Dec-2006  MX New topic
Enabled Property

Description
Enables and disables (ON/OFF) the port mapping and control line output of the specified test set.

If the specified test set is not connected or not ON, then setting Enabled = True will report an error. All other properties can be set when the test set is not connected.

When this command is set to ON or OFF, then the display of the test set status bar (ShowProperties Property) is also set to ON or OFF.

VB Syntax
\[ tset.Enabled = value \]

Variable (Type) - Description

\[ tset \]
A TestsetControl object
OR
An E5091Testset object

\[ value \] (Boolean)
True Enables test set control.
False Disables test set control.

Return Type
Boolean

Default
False

Examples
See E5091A Example Program
See External Testset Program

C++ Syntax
HRESULT get_Enabled(VARIANT_BOOL *state);
HRESULT put_Enabled(VARIANT_BOOL state);

Interface
ITestsetControl
IE5091Testsets
EnableLOPowerCal Property

Description
Sets and returns whether or not the LO power cal step is included in the cal steps when an IMDX Cal or NFX Cal is performed.

VB Syntax

\[ \text{obj}.\text{EnableLOPowerCal}(n) = \text{value} \]

Variable (Type) - Description

\[ \text{obj} \]
A SweptIMDCal (object)
A NoiseCal (object)

\[ \text{n} \]
LO Stage. Choose 1. (Only single LO allowed)

\[ \text{value} \]
(Boolean) Choose from:

- **False** - Skips over the LO Power Cal when calibrating.
- **True** - Includes a step for LO Power Cal when calibrating.

Return Type
Boolean

Default
PNA Rev. 9.1 and above: **False**
Before Rev 9.1: **True**

Examples

\[ \text{imd}.\text{EnableLOPowerCal}(1) = \text{true} \]

\[ \text{loPwrCal} = \text{imd}.\text{EnableLOPowerCal}(1) \]

C++ Syntax

HRESULT get_EnableLOPowerCal (long stage, BOOL *enable)
HRESULT put_EnableLOPowerCal (long stage, BOOL enable)

Interface
ISweptIMD2
INoiseCal2

Last Modified:

- 21-Oct-2009   Added NoiseCal
- 27-Mar-2009   MX New topic
### EnablePhase Property

**Description**
Sets and returns the state of SMC Phase measurements.

In the User Interface, there are two "enable phase" checkboxes: in the Phase Settings dialog and in the Calibration Wizard. Checking one enables both. This single command also enables both.

**VB Syntax**
```
obj.EnablePhase = bool
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>obj</code></td>
<td>A Mixer Interface pointer to the Measurement (object)</td>
</tr>
<tr>
<td>A Converter (Object)</td>
<td></td>
</tr>
<tr>
<td><code>bool</code></td>
<td>(Boolean) -</td>
</tr>
<tr>
<td>True</td>
<td>Include phase in SMC measurements</td>
</tr>
<tr>
<td>False</td>
<td>Do NOT include phase in SMC measurements</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**
```
mixer.EnablePhase = True
```

**C++ Syntax**
```c
HRESULT get_EnablePhase(VARIANT_BOOL * val);
HRESULT put_EnablePhase(VARIANT_BOOL val);
```

**Interface**
`IMixer13`

Last Modified:
25-Mar-2010   MX New topic
EnablePowerCompensation Property

**Description**
Adjusts the source power at the specified port by the combined amount of loss through ALL enabled fixturing operations. Use this function to set the power level at the DUT input. Learn more.

**Note:** This command affects ALL measurements on the specified channel.

**VB Syntax**

```vbnet
fixture.EnablePowerCompensation (port) = bool
```

**Variable**

- `fixture` (Type) - A Fixturing (object)
- `port` (Integer) - Port number to receive power compensation.
- `bool` (Boolean)
  - `True` - Compensate source power
  - `False` - Do NOT compensate source power

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
fixture.EnablePowerCompensation(1) = True
value = fixture.EnablePowerCompensation(2) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_EnablePowerCompensation(short port, VARIANT_BOOL *pState );
HRESULT put_EnablePowerCompensation(short port, VARIANT_BOOL bVal);
```

**Interface**
IFixturing5

Last Modified:
13-Apr-2010    MX New topic
EnableSnPDataExtrapolation

**Description**
Turns ON and OFF SNP file extrapolation for both 2-port and 4-port embedding/de-embedding.

**VB Syntax**
`fixture.EnableSnPDataExtrapolation = bool`

**Variable** *(Type) - Description*

- `fixture` - A Fixturing *(object)*
- `bool` - **True** - Turns Extrapolation ON
  
  **False** - Turns Extrapolation OFF

**Return Type**
Boolean

**Default**
False

**Examples**
`fixture.EnableSnPDataExtrapolation = True`

`value = fixture.EnableSnPDataExtrapolation ` *Read*

**C++ Syntax**
```cpp
HRESULT get_EnableSnPDataExtrapolation(VARIANT_BOOL *pExtrap);
HRESULT put_EnableSnPDataExtrapolation(VARIANT_BOOL bExtrap);
```

**Interface**
IFixturing6

Last Modified:
16-Nov-2010    MX New topic
### EnableSourceUnleveledEvents Property

**Description**
Specifies whether or not to report [Source Unleveled](#) errors as system events. These events can trigger an [OnSystemEvent](#) call.

This setting will revert to the default (False) setting on Instrument Preset.

**VB Syntax**

```vb
pref.EnableSourceUnleveledEvents = bool
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pref</td>
<td>A <a href="#">Preferences</a> (object)</td>
</tr>
<tr>
<td>bool</td>
<td>(Boolean) - Choose from:</td>
</tr>
<tr>
<td>False</td>
<td>Do NOT report Source Unleveled Errors.</td>
</tr>
<tr>
<td>True</td>
<td>Report Source Unleveled Errors.</td>
</tr>
</tbody>
</table>

**Return Type**

Boolean

**Default**

False

**Examples**

```vb
pref.EnableSourceUnleveledEvents = False 'Write
prefer = pref.EnableSourceUnleveledEvents 'Read
```

**C++ Syntax**

```c++
HRESULT put_EnableSourceUnleveledEvents( VARIANT_BOOL bsourcUnlEnable)
HRESULT get_EnableSourceUnleveledEvents( VARIANT_BOOL *bsourcUnlEnable)
```

**Interface**

IPreferences3

---

**Last modified:**

- 9-Apr-2010  Added Preset note
- 15-Nov-2006  MX New command
Write/Read

EndOfSweepOperation Property

Description
Set and read the action which should be taken at the end of the last frequency or power sweep in the measurement. This setting is used to protect a sensitive device from too much power during the sweep retrace.

VB Syntax

```vbnet
(gca.EndOfSweepOperation = value
```

Variable (Type) - Description

gca A GainCompression (object)

value (NAGCAEndOfSweepOperation)

- naStandard (0) Use the default PNA method. Learn more.
- naSetToStartPower (1) Sweep Start power
- naSetToStopPower (2) Sweep Stop power.
- naSetRFOff (3) Always turn power OFF while waiting.

Return Type
Enum

Default
naStandard

Examples

```vbnet
gca.EndOfSweepOperation = naSetToStartPower 'Write

eos = gca.EndOfSweepOperation 'Read
```

C++ Syntax

```cpp
HRESULT get_EndOfSweepOperation(tagNAGCAEndOfSweepOperation* pVal)

HRESULT put_EndOfSweepOperation(tagNAGCAEndOfSweepOperation newVal)
```

Interface
IGainCompression

Last Modified:
11-Sep-2007    MX New topic
### ENRFile Property

**Description**
Sets and returns the name of the ENR file associated with the noise source.

**VB Syntax**

```vba
noise.ENRFile = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>noise</td>
<td>A NoiseCal (object)</td>
</tr>
<tr>
<td>value</td>
<td>(string) Full path and ENR filename.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Not Applicable

**Examples**

```vba
noise.ENRFile = "c:/ProgramFiles/Agilent/Network Analyzer/Documents/ENR/346C.enr"
```

ENR = noise.ENRFile  'Read

**C++ Syntax**

```cpp
HRESULT get_ENRFile(BSTR* pValue)
HRESULT put_ENRFile(BSTR pNewValue)
```

**Interface**
INoiseCal

---

**Last Modified:**
29-May-2007  MN New topic
### ENRId Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns ID of ENR table.</th>
</tr>
</thead>
</table>

#### VB Syntax

```
enr.ENRSN = ID
```

#### Variable *(Type) - Description*

<table>
<thead>
<tr>
<th>enr</th>
<th>An <strong>ENRFile</strong> <em>(object)</em></th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Identifier for the ENR table</td>
</tr>
</tbody>
</table>

#### Return Type

*String*

#### Default

*Not Applicable*

#### Examples

*See example program*

#### C++ Syntax

```c++
HRESULT get_ENRId(BSTR *Val);
HRESULT put_ENRId(BSTR Val);
```

#### Interface

*IENRFile*

---

**Last Modified:**

- 2-Aug-2007    MX New topic
ENRSN Property

Description  Sets and returns the serial number of the noise source for which the ENR file applies.

VB Syntax  
```
enr.ENRSN = serialNumber
```

Variable  
*enr* An [ENRFile](#) (object)

*serialNumber* (String) Serial number of the noise source.

Return Type  String

Default  Not Applicable

Examples See example program

C++ Syntax  
```
HRESULT get_ENRSNBSTR *Val);
HRESULT put_ENRSN(BSTR Val);
```

Interface  IENRFile

Last Modified:

2-Aug-2007  MX New topic
### ErrorCorrection Property

**Description**
Sets (or returns) error correction ON or OFF for the measurement.

**VB Syntax**
```
meas.ErrorCorrection = value
```

**Variable**
- **meas** *(Type)* - Description
  - A Measurement *(object)*
- **value** *(boolean)*
  - **False** - Turns error correction OFF
  - **True** - Turns error correction ON

**Return Type**
Boolean

**Default**
See Error Correction

**Examples**
```
meas.ErrorCorrection = True 'Write
errcorr = meas.ErrorCorrection 'Read
```

**C++ Syntax**
```
HRESULT put_ErrorCorrection (VARIANT_BOOL bState)
HRESULT get_ErrorCorrection (VARIANT_BOOL *bState)
```

**Interface**
IMeasurement
ErrorCorrection (Channel) Property

Description
Attempts to sets error correction ON or OFF for all of the measurements on the channel. This setting may not be successful for some measurements because the Cal Set currently in place may not contain the appropriate calibration data. To read the error correction state for a measurement, use Error Correction Property.

VB Syntax
chan.ErrorCorrection = value

Variable (Type) - Description
chan A Channel (object)
value (boolean)
False - Turns error correction OFF
True - Turns error correction ON

Return Type
Boolean

Default
About Error Correction

Examples
chan.ErrorCorrection = True

C++ Syntax
HRESULT put_ErrorCorrection (VARIANT_BOOL bState)

Interface
IChannel7
**ErrorCorrectionIndicator Property**

**Description**
Returns the error correction state for the measurement.

**VB Syntax**
```
value = meas.ErrorCorrectionIndicator
```

**Variable**
(Type)  - Description

- `value`  (Enum)  Error correction state.

- 0 - `naErrorCorrectionIndicator_None`  - No error correction
- 1 - `naErrorCorrectionIndicator_Master`  - Original error correction terms applied.
- 2 - `naErrorCorrectionIndicator_Interpolated`  - Error terms are interpolated.  Learn more
- 3 - `naErrorCorrectionIndicator_Delta`  - Delta Match calibration terms.  Learn more
- 4 - `naErrorCorrectionIndicator_Invalid`  Error terms are not valid.

**meas**  A Measurement (object)

**Return Type**
Enum as `NAErrorCorrectionIndicator`

**Default**
See Error Correction

**Examples**
```vbnet
errcorr = meas.ErrorCorrectionIndicator  'Read
```

**C++ Syntax**
```csharp
HRESULT get_ErrorCorrectionIndicator (enum NAErrorCorrectionIndicator *pIndicator);
```

**Interface**
`IMeasurement14`

---

Last Modified:
14-Jul-2010  MX New topic
Write-only

**ExtendedProperties Property**

**Description**
Provides access to the custom properties and methods of an external device.

**VB Syntax**
Set \( PSG = ExtDev.ExtendedProperties \)

**Variable** (Type) - Description

\( PSG \) (Object) Variable to store the returned handle to an external device.

\( ExtDev \) An [ExternalDevice](#) (object)

**Return Type**
Object

**Default**
Not Applicable

**Examples** See Example

**C++ Syntax**
HRESULT get_ExtendedProperties(IDispatch** ppObject);

**Interface** IExternalDevice

---

Last Modified:

27-Aug-2009  MX New topic
Write/Read
ExternalALC Property

Description  Sets or returns the source of the analyzer leveling control.

VB Syntax  app.ExternalALC = value

Variable  (Type) - Description
app  An Application (object)
value  (boolean) - Choose from:
   True  - Leveling control supplied through the rear panel.
   False  - Leveling control supplied inside the analyzer

Return Type  Boolean

Default  False

Examples  app.ExternalALC = True 'Write
           extALC = app.ExternalALC 'Read

C++ Syntax  HRESULT get_ExternalALC(VARIANT_BOOL *pVal)
             HRESULT put_ExternalALC(VARIANT_BOOL newVal)

Interface  IApplication
Write/Read

ExternalDeviceDeActivatePolicy Property

Description
Set and return whether External Devices remain activated or are de-activated when the PNA is Preset or when a Instrument State is recalled.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

See the ExternalDevices collection.

VB Syntax

\[
pref.ExternalDeviceDeActivatePolicy = value
\]

Variable (Type) - Description

- **pref** A Preferences (object)
- **value** (Boolean) - Choose from:

  - **True** - External device are de-activated when the PNA is Preset or when a Instrument State is recalled.
  - **False** - External devices remain active when the PNA is Preset or when a Instrument State is recalled.

Return Type
Boolean

Default
True

Examples

\[
pref.ExternalDeviceDeActivatePolicy = 1 \ 'Write
\]

\[
dDevPolicy = pref.ExternalDeviceDeActivatePolicy \ 'Read
\]

C++ Syntax

HRESULT get_ExternalDeviceDeActivatePolicy(VARIANT_BOOL * pref);
HRESULT put_ExternalDeviceDeActivatePolicy(VARIANT_BOOL pref);

Interface
IPreferences10

Last Modified:

17-Sep-2009    MX New topic
ExternalTriggerConnectionBehavior Property

Description

Configures the external triggering signal for the PNA.

- To control BNC1 and BNC2 with this command, then you MUST have AuxTriggerScopeIsGlobal = True.
- TriggerSource Property is automatically set to External when ExternalTriggerConnectionBehavior is sent.
- Edge triggering is only available on some PNA models.
- For more information, see External Triggering.

VB Syntax

trigsetup.ExternalTriggerConnectionBehavior (conn) = value

Variable (Type) - Description

trigsetup A TriggerSetup (object)

conn (enum NATriggerConnection) Rear Panel connector to send or receive trigger signals. Choose from:

Only one of the input connectors is active at a time. When a command is sent to one, the PNA automatically makes the other INACTIVE.

0 - naTriggerConnectionAUXT Trigger IN from rear-panel AUX IO connector Pin 19
1 - naTriggerConnectionBNC1 Trigger IN from rear-panel Trigger IN BNC connector
2 - naTriggerConnectionBNC2 Trigger OUT to rear-panel Trigger OUT BNC connector. Only useful in point sweep mode.
3 - naTriggerConnectionMATH Trigger IN from rear-panel Material Handler connector Pin 18

value (enum NAExternalTriggerBehavior) -

0 - naTriggerInactive - Disables the specified connector.

Choose from ONLY 1 through 4 when <conn> is set to either naTriggerConnectionBNC1 or naTriggerConnectionAUXT

1 - naTriggerInEdgeNegative - Triggers the PNA when receiving a negative going signal
2 - naTriggerInEdgePositive - Triggers the PNA when receiving a positive going signal
3 - naTriggerInLevelLow - Triggers the PNA when receiving a low level signal
4 - naTriggerInLevelHigh - Triggers the PNA when receiving a High-level signal

Choose from ONLY 5 through 8 when <conn> is set to naTriggerConnectionBNC2.

In addition to sending this command, you must also use TriggerOutputEnabled Property to enable
the BNC2 output.

5 - **naTriggerOutPulsePositiveAfter** - Sends a POSITIVE going TTL pulse at the END of each point during the sweep.

6 - **naTriggerOutPulsePositiveBefore** - Sends a POSITIVE going TTL pulse at the START of each point during the sweep.

7 - **naTriggerOutPulseNegativeAfter** - Sends a NEGATIVE going TTL pulse at the END of each point during the sweep.

8 - **naTriggerOutPulseNegativeBefore** - Sends a NEGATIVE going TTL pulse at the START of each point during the sweep.

**Return Type**: Enum as NAExternalTriggerBehavior

**Default**

BNC1 = **naTriggerInactive**

BNC2 = **naTriggerInactive**

AUXT = **naTriggerInLevelHigh**

When **Output is enabled**

BNC1 = **naTriggerInactive**

BNC2 = **naTriggerOutPulsePositiveAfter**

AUXT = **naTriggerInLevelHigh**

**Examples**

```c
trigsetup.ExternalTriggerConnectionBehavior (naTriggerConnectionAUXT) = naTriggerInLevelLow  'Write

trigBehav = trigsetup.ExternalTriggerConnectionBehavior (naTriggerConnectionAUXT)  'Read
```

**C++ Syntax**

```cpp
HRESULT get_ExternalTriggerConnectionBehavior(tagNATriggerConnection connection,tagNAExternalTriggerBehavior *trigger);

HRESULT put_ExternalTriggerConnectionBehavior(tagNATriggerConnection connection,tagNAExternalTriggerBehavior trigger);
```

**Interface**

ITriggerSetup

---

**Last Modified:**

25-Feb-2008   Added 'Global' note
ExternalTriggerDelay Property

**Description**
Sets and reads the trigger delay for all measurements in the CHANNEL. This delay is only applied while 
app.Source = naTriggerSourceExternal and 
trigsetup.Scope = naChannelTrigger. After an external trigger is applied, the start of the sweep is 
delayed for the specified delay value plus any inherent latency.

To apply a trigger delay for all channels (Global), use TriggerDelay Property.

**VB Syntax**

```vb
chan.ExternalTriggerDelay = value
```

**Variable**

- **Type** - Description

  - `chan` A Channel (object)

  - `value` Double - Trigger delay value in seconds. Range is from 0 to 107 seconds

**Return Type**

- Double

**Default**

- 0

**Examples**

```vb
chan.ExternalTriggerDelay = .003  'Write

delay = chan.ExternalTriggerDelay  'Read
```

**C++ Syntax**

- HRESULT get_ExternalTriggerDelay(double *delay);
- HRESULT put_ExternalTriggerDelay(double delay)

**Interface**

IChannel6
**F1Frequency Property**

**Description**
Sets and returns the frequency of the F1 tone. Use with IMD sweep types:

- `naIMDToneCWSweep`
- `naIMDTonePowerSweep`

**VB Syntax**

```vbnet
object.F1Frequency = value
```

**Variable (Type) - Description**

- `object` A `SweptIMD` or `IMSpectrum` Object

- `value` (Double) F1 tone frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**
Double

**Default**
.9995 GHz

**Examples**

```vbnet
imd.F1Frequency = 100e6 'Write
```

```vbnet
value = imd.F1Frequency 'Read
```

**C++ Syntax**

```csharp
HRESULT get_F1Frequency(double *pVal)
HRESULT put_F1Frequency(double newVal)
```

**Interface**
`ISweptIMD`
`IMSpectrum`

---

**Last Modified:**

19-Aug-2008   MX New topic
F2Frequency Property

**Description**
Sets and returns the frequency of the F2 tone. Use with IMD sweep types:

- naIMDToneCWSweep
- naIMDTonePowerSweep

**VB Syntax**
`object.F2Frequency = value`

**Variable**
*Type* - Description

`object` A SweptIMD or IMSpectrum Object

`value` (Double) F2 tone frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**
Double

**Default**
1.0005 GHz

**Examples**
```
imd.F2Frequency = 200e9 'Write
value = imd.F2Frequency 'Read
```

**C++ Syntax**

```
HRESULT get_F2Frequency(double *pVal)
HRESULT put_F2Frequency(double newVal)
```

**Interface**
ISweptIMD
IMSpectrum

---

_Last Modified:_
19-Aug-2008  MX New topic
FailedTraces Property

Description
Set and return the limit line color of failed traces or failure indicators (dots) and the word Fail.

VB Syntax
`colors.FailedTraces = value`

Variable (Type) - Description
- `colors` A `ComColors` (object)
- `value` (Long Integer) - RGB color of the FailedTraces pen.
  Convert the three RGB colors to an integer as follows:
  \[ \text{RGB} = R + (G \times 2^8) + (B \times 2^{16}) \]
  To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

Return Type
Long

Default
Display = 255,20,20
Print = 255,20,20

Examples
```
R = 10
G = 10
B = 10
\text{RGB} = R + (G \times 2^8) + (B \times 2^{16})
```
```
\text{colors.FailedTraces} = \text{RGB} \ 'Write
\text{color} = \text{colors.FailedTraces} \ 'Read
```

C++ Syntax
```
HRESULT get_FailedTraces(long* pVal);
HRESULT put_FailedTraces(long newVal);
```

Interface
IComColors

Last Modified:
7-Aug-2009  MX New topic
FastCWPointCount Property

**Description**

Enables Fast CW sweep and sets the number of data points for the channel. **Sweep Type** must already be set to CWTime and FIFO must already be enabled.

**See Also**

FIFO and other Antenna Features
FIFO Object
Example program
N5264A Measurement Receiver

**VB Syntax**

```
chan.FastCWPointCount = value
```

**Variable (Type) - Description**

- **chan**
  - A Channel Object

- **value**
  - (Long Integer) Number of data points to measure in Fast CW mode. This setting overwrites the standard number of points setting for the channel.
  - Set to 0 to disable Fast CW.

**Return Type**

Long Integer

**Default**

0

**Examples**

```
chan.FastCWPointCount = 1e3 'Write

value = chan.FastCWPointCount 'Read
```

**C++ Syntax**

```
HRESULT get_FastCWPointCount(long *value)
HRESULT put_FastCWPointCount(long value)
```

**Interface**

IChannel16

---

Last Modified:

- 2-Feb-2011   Fixed syntax
- 7-Oct-2008   MX New topic
FastMode Property

**Description**
Sets and returns the state of a separate IFBW setting for leveling sweeps. ON allows a higher (faster) IFBW than the measurement sweep. It also causes leveling sweeps to be noisier.

**VB Syntax**
```
RxLevel.FastMode(srcPort) = value
```

**Variable**

- **RxLevel**
  - (Type) A *ReceiverLeveling* Object

- **srcPort**
  - (Long Integer) Source port for which to set Fast Mode for Receiver Leveling.

  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**
  - (Boolean) Separate IFBW setting state. Choose from:
    - **True** - Use separate IFBW setting. Specify IFBW using *LevelingIFBW*.
    - **False** - Use same IFBW as the measurement sweep. Specify IFBW using *IF Bandwidth*.

**Return Type**
Variant Boolean

**Default**
True

**Examples**
```
rxLevel.FastMode (1) = True ' Write

value = rxLevel.FastMode 2' Read
```

**C++ Syntax**
```
HRESULT get_FastMode(long port, VARIANT_BOOL* pVal);
HRESULT put_FastMode(long port, VARIANT_BOOL newVal);
```

**Interface**
IRxLevelingConfiguration

---

Last Modified:
13-Feb-2009   MX New topic
Read-only

FilterBW Property

**Description**
Returns the results of the SearchBandwidth method.

**VB Syntax**
filtBW = meas.FilterBW

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>- Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>filtBW</td>
<td>(single) - Variable to store bandwidth data</td>
</tr>
<tr>
<td>meas</td>
<td>A Measurement (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Single

**Default**
Not applicable

**Examples**
filterBW = meas.FilterBW 'Read

**C++ Syntax**
HRESULT get_FilterBW(float* bw)

**Interface**
IMeasurement
FilterCF Property

Description  Returns the Center Frequency result of the SearchBandwidth method.

VB Syntax  \[ filtCF = meas.FilterCF \]

Variable  \( \text{(Type)} \)  -  \( \text{Description} \)

\( filtCF \)  \( \text{(double)} \) - Variable to store bandwidth CF data

\( meas \)  A Measurement \( \text{(object)} \)

Return Type  Double

Default  Not applicable

Examples  \[ filtCF = meas.FilterCF \]  'Read

C++ Syntax  HRESULT get_FilterCF(double* centerFrequency)

Interface  IMeasurement
FilterLoss Property

Description  Returns the Loss value of the SearchBandwidth method.

VB Syntax  \texttt{filtLoss = meas.FilterLoss}

Variable  \textbf{Type} - Description

\textit{filtLoss} (\textit{single}) - Variable to store bandwidth Loss data

\textit{meas} A Measurement (\textit{object})

Return Type  Single

Default  Not applicable

Examples  \texttt{filterLoss = meas.FilterLoss 'Read}

C++ Syntax  \texttt{HR\_RESULT get\_FilterLoss(float* loss)}

Interface  IMeasurement
**FilterQ Property**

**Description**
Returns the Q (quality factor) result of the SearchBandwidth method.

**VB Syntax**
\[filtQ = meas.FilterQ\]

**Variable**
- \(filtQ\) *(single)* - Variable to store bandwidth Q data
- \(meas\) A Measurement *(object)*

**Return Type**
Single

**Default**
Not applicable

**Examples**
\[filtQ = meas.FilterQ\] 'Read

**C++ Syntax**
HRESULT get_FilterQ(float* quality)

**Interface**
IMeasurement
FilterErrors Property

Description
Returns the error string associated with the digital filters. The return string has three fields separated by commas: "stage1 status, stage2 status, stage3 status"

Each of these fields can contain one or more of the following error codes:

- **NO ERROR**
- **NUMBER-OF-COEFFICIENTS** - the number of coefficients is excessive for that filter-stage
- **COEFFICIENT VALUE** - one or more coefficients are out of range for that filter-stage
- **SUM-OF-COEFFICIENTS** - the sum of all coefficients is excessive for that filter-stage,
- **FREQUENCY** - the frequency for Stage 1 is out of range (only applies stage1 field),
- **PARAMETER** - one or more parameters are out of range (only applies to stage 3 field)

VB Syntax

```vbnet
value = spm4.FilterErrors
```

Variable (Type) - Description

- **value** Variable to store the returned errors.
- **spm4** A SignalProcessingModuleFour (object)

Return Type String

Default Not Applicable

Examples

```vbnet
mode = spm4.FilterErrors 'Read
'example return strings"

NO ERROR, NO ERROR, NO ERROR
indicates no errors,

*SUM-OF-COEFFICIENTS, NO ERROR, NO ERROR
indicates that the sum of all filter coefficients exceed the maximum value for the Stage-1 filter,

*COEFFICIENT *SUM-OF-COEFFICIENTS, NO ERROR, *PARAMETER
indicates a problems with Stage 1 coefficients and a problem with one or more of the parameters associated with the Stage 3 filter.
```

C++ Syntax

```cpp
HRESULT get_FilterErrors(BSTR* dspErrors);
```
Interface  ISignalProcessingModuleFour

Last Modified:

1-Jan-2007  MX New topic
FilterMode Property

Description
Sets and returns whether the PNA configures the 3-stage digital filter settings or they will be configured manually. When making manual settings, also send ADCCaptureMode Property which routes the IF through the 3-stage filter.

VB Syntax

```
spm4.FilterMode = value
```

Variable

(Line) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>spm4</td>
<td>A SignalProcessingModuleFour (object)</td>
</tr>
<tr>
<td>value</td>
<td>(enum as NAModes) Filter mode. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naAUTO  PNA controls digital filter settings.</td>
</tr>
<tr>
<td></td>
<td>naMANUAL You control digital filter settings using other SignalProcessingModuleFour commands.</td>
</tr>
</tbody>
</table>

Return Type

Enum

Default

naAUTO

Examples

```
spm4.FilterMode = naAUTO 'Write
mode = spm4.FilterMode 'Read
```

C++ Syntax

```
HRESULT get_FilterMode(tagNAModes* dspMode);
HRESULT put_FilterMode(tagNAModes dspMode);
```

Interface

ISignalProcessingModuleFour

Last Modified:

24-Jan-2007 MX New topic
FirmwareMajorRevision Property

**Description**
Returns the major firmware revision number as an integer. For example, given a firmware revision number A.03.30, this command returns 3.

**VB Syntax**
```vbnet
value = cap.FirmwareMajorRevision
```

**Variable**
- **value** (Long) - Variable to store the returned integer value of the firmware revision number.
- **cap** A Capabilities (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**
```vbnet
value = cap.FirmwareMajorRevision 'Read
```

**C++ Syntax**
```csharp
HRESULT get_FirmwareMajorRevision(long * majorRev);
```

**Interface**
ICapabilities
FirmwareMinorRevision Property

**Description**
Returns the minor firmware revision number as an integer. For example, given a firmware revision number A.03.30, this command returns 30.

**VB Syntax**

\[
\text{value} = \text{cap}.\text{FirmwareMinorRevision}
\]

**Variable (Type) - Description**

- **value** (Long) - Variable to store the returned decimal value of the firmware revision number.
- **cap** A Capabilities (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**

value = cap.FirmwareMinorRevision 'Read

**C++ Syntax**

HRESULT get_FirmwareMinorRevision(long * minorRev );

**Interface**
ICapabilities
FirmwareSeries Property

**Description**
Returns the alpha portion of the firmware revision number. For example, given a firmware revision number A.03.30, this command returns A.

**VB Syntax**
```
value = cap.FirmwareSeries
```

**Variable (Type) - Description**
- **value** (String) - Variable to store the returned alpha value of the firmware revision number.
- **cap** A Capabilities (object)

**Return Type**
String

**Default**
Not Applicable

**Examples**
```
value = cap.FirmwareSeries 'Read
```

**C++ Syntax**
```
HRESULT get_FirmwareSeries(BSTR * series);
```

**Interface**
ICapabilities
FixedDelay Property

Description: Set and return the known delay through the calibration mixer.

VB Syntax: `smc.FixedDelay = value`

Variable (Type) - Description
- `smc` An `SMCType` (object)
- `value` (Double) Known delay through the calibration mixer in seconds.

Return Type: Double

Default: 0 seconds

Example:
```vbnet
SMC.FixedDelay = 12e-9  'Write
value = SMC.FixedDelay   'Read
```

C++ Syntax:
```
HRESULT put_FixedDelay(Double Value);
HRESULT get_FixedDelay(Double* Value);
```

Interface: `SMCType5`

Last Modified:
25-Mar-2010   MX New topic
FixedPhase Property

Description  Write and read the fixed phase value. Must not be logarithmic sweep type.

VB Syntax  
```
phase.FixedPhase(srcPort) = value
```

Variable  
**variable** (Type) - Description

- **phase**  A [PhaseControl] Object
- **srcPort**  (Long Integer) Source port for which to make phase control settings.
  
  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#)

- **value**  (Double) Phase value in degrees. Choose a value between -360 and 360.

Return Type  Double

Default  0 degrees

Examples  
```
phase.FixedPhase 1 = 15 ' Write
```

```
value = phase.FixedPhase 2' Read
```

C++ Syntax  
```
HRESULT get_FixedPhase(long port, double* pVal);
HRESULT put_FixedPhase(long port, double newVal);
```

Interface  IPhaseControl

Last Modified:  8-Dec-2010  MX New topic
### FixedRatioedPower Property

**Description**
Write and read the fixed power ratioed value. Must NOT be in power sweep to use this value during phase control.

**VB Syntax**

```vbnet
phase.FixedRatioedPower(srcPort) = value
```

**Variable (Type) - Description**

- **phase**
  A `PhaseControl` Object

- **srcPort**
  (Long Integer) Source port for which to make phase control settings.

  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**
  (Double) Fixed power ratio value in dBc within the allowable range of the PNA.

**Return Type**
Double

**Default**
0 dBc

**Examples**

```vbnet
phase.FixedRatioedPower 1 = -1 ' Write
value = phase.FixedRatioedPower 2' Read
```

**C++ Syntax**

```cpp
HRESULT get_FixedRatioedPower(long port, double* pVal);
HRESULT put_FixedRatioedPower(long port, double newVal);
```

**Interface**
`IPhaseControl`

---

**Last Modified:**
8-Dec-2010  MX New topic
**FixturingState Property**

**Description**
Turns all three fixturing functions (de-embedding, port matching, impedance conversion) ON or OFF for all ports on the specified channel. This does NOT affect port extensions.

**VB Syntax**
`fixture.FixturingState = value`

**Variable (Type) - Description**

- `fixture` A Fixturing (object)
- `value` (boolean)
  - `True` - Turns Fixturing ON
  - `False` - Turns Fixturing OFF

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
fixture.FixturingState = True 'Write
value = fixture.FixturingState 'Read
```

**C++ Syntax**

```cpp
HRESULT get_FixturingState(VARIANT_BOOL *pVal)
HRESULT put_FixturingState(VARIANT_BOOL newVal)
```

**Interface**
IFixturing
Read-only

FootSwitch Property

**Description**
Reads the Footswitch Input (pin 20 of the AUX IO connector).

**VB Syntax**

```vbnet
value = AuxIO.Footswitch
```

**Variable (Type) - Description**

- **value** (boolean) - Variable to store the returned value
  - **False** - foot switch is released
  - **True** - footswitch is depressed

- **AuxIO** (object) - A Hardware Aux I/O object

**Return Type**
Boolean

**Default**
True

**Examples**

```vbnet
fs = aux.Footswitch
```

**C++ Syntax**

```cpp
HRESULT get_FootSwitch ( VARIANT_BOOL* State );
```

**Interface**
IHWAuxIO
### About the Aux I/O Connector

#### FootswitchMode Property

**Description**
Determines what occurs when the footswitch is pressed. For more information see the FootSwitch In pin description in the Auxiliary IO connector.

**VB Syntax**

```vbnet
AuxIo.FootSwitchMode = value
```

**Variable**  
**(Type)** - Description

<table>
<thead>
<tr>
<th>value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 - naIgnoreFootswitch</td>
<td>Footswitch presses are ignored.</td>
</tr>
<tr>
<td>1 - naSweepTrigger</td>
<td>Footswitch presses trigger a sweep. The PNA must be in Manual Trigger Mode.</td>
</tr>
<tr>
<td>2 - naRecallNextState</td>
<td>Footswitch presses recall an instrument state. When more than one state is available, then each footswitch press recalls the next state, then starts over from the beginning. It is possible for a recalled state to override the current mode. If the recalled state is IGNore, then mode changes and additional footswitch presses are ignored.</td>
</tr>
<tr>
<td>3 - naRunMacro</td>
<td>Footswitch presses load and run a macro. When more than one macro is available, then each footswitch press loads and runs the next macro, then starts over from the beginning. It is possible for a Macro to override the current mode. If the macro contains a Preset, then the mode changes to the default setting IGNore and additional footswitch presses are ignored.</td>
</tr>
</tbody>
</table>

**AuxIO**  
**(object)** - A Hardware Aux I/O object

**Return Type**
NAFootSwitchMode

**Default**
0 - naIgnoreFootswitch

**Examples**

```vbnet
auxIo.FootSwitchMode = naIgnoreFootSwitch 'Write
```

**C++ Syntax**

```cpp
HRESULT get_FootSwitchMode(NAFootSwitchMode *pFootSwitchMode )
HRESULT put_FootSwitchMode(NAFootSwitchMode newFootSwitchMode)
```

**Interface**
IHWAuxIO3
### ForceDeEmbedENRAdapter Property

**Description**
Set and read the De-embedENRAdapter state. [Learn more](#).

**VB Syntax**

```vbnet
noiseCal.ForceDeEmbedENRAdapter = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>noiseCal</td>
<td>A NoiseCal (object)</td>
</tr>
<tr>
<td>value</td>
<td>(boolean) - ENR Adapter de-embed state.</td>
</tr>
<tr>
<td>False</td>
<td>False - Do not Force de-embedding.</td>
</tr>
<tr>
<td>True</td>
<td>True - Force de-embedding.</td>
</tr>
</tbody>
</table>

**Return Type**

Boolean

**Default**

False

**Examples**

```vbnet
noiseCal.ForceDeEmbedENRAdapter = False 'Write
AdapterDembed = noiseCal.ForceDeEmbedENRAdapter 'Read
```

**C++ Syntax**

```cpp
HRESULT get_ForceDeEmbedENRAdapter(VARIANT_BOOL* on);
HRESULT put_ForceDeEmbedENRAdapter(VARIANT_BOOL on);
```

**Interface**

INoiseCal2

---

Last Modified:

26-Oct-2009    MX New topic
## ForceDeEmbedSensorAdapter Property

**Description**  
Set and read the state of power sensor adapter de-embedding. [Learn more.](#)

**VB Syntax**  
```
noiseCal.ForceDeEmbedSensorAdapter = value
```

**Variable**  
*(Type)* - Description

- **noiseCal**  
  A [NoiseCal](object) (object)

- **value**  
  *(boolean)* - Power sensor adapter de-embed state.
  - **False** - Do not Force de-embedding.
  - **True** - Force de-embedding.

**Return Type**  
Boolean

**Default**  
False

**Examples**  
```
noiseCal.ForceDeEmbedSensorAdapter = False 'Write

AdapterDembed = noiseCal.ForceDeEmbedSensorAdapter 'Read
```

**C++ Syntax**  
```
HRESULT get_ForceDeEmbedSensorAdapter(VARIANT_BOOL* on);
HRESULT put_ForceDeEmbedSensorAdapter(VARIANT_BOOL on);
```

**Interface**  
INoiseCal2

---

**Last Modified:**  
26-Oct-2009  
MX New topic
Format Property (marker)

**Description**
Sets (or returns) the format of the marker.

**VB Syntax**
```
mark.Format = value
```

**Variable**
(Type) - Description

`mark` A [Marker](#) (object)

`value` (enum NAMarkerFormat) - Choose from:

- 0 - naMarkerFormat_LinMag
- 1 - naMarkerFormat_LogMag
- 2 - naMarkerFormat_Phase
- 3 - naMarkerFormat_Delay
- 4 - naMarkerFormat_Real
- 5 - naMarkerFormat_Imaginary
- 6 - naMarkerFormat_SWR
- 7 - naMarkerFormat_LinMagPhase
- 8 - naMarkerFormat_LogMagPhase
- 9 - naMarkerFormat_RealImaginary
- 10 - naMarkerFormat_ComplexImpedance
- 11 - naMarkerFormat_ComplexAdmittance
- 12 - naMarkerFormat_Kelvin
- 13 - naMarkerFormat_Fahrenheit
- 14 - naMarkerFormat_Celsius

**Return Type**
NAMarkerFormat

**Default**
1 - naMarkerFormat_LogMag

**Examples**
```
mark.Format = naMarkerFormat_SWR 'Write
fmt = mark.Format 'Read
```

**C++ Syntax**
```
HRESULT get_Format(tagNAMarkerFormat *pVal)
HRESULT put_Format(tagNAMarkerFormat newVal)
```

**Interface**
IMarker

Last Modified:
1-Oct-2007   Added temperature formats
**FormatUnit Property**

**Description**
Sets and returns the units for the specified data format. Measurements with display formats other than those specified are not affected.

**VB Syntax**
```
meas.FormatUnit (format) = value
```

**Variable**
**Type** - Description

<table>
<thead>
<tr>
<th>meas</th>
<th>A Measurement (object)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>format</th>
<th>(enum NADataFormat) - Choose:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>naDataFormat_LogMag</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>(enum naFormatUnit) - Choose from:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>naFormatUnit_dBm Units are displayed in dBm. 0 dBm = 0.001 watt</td>
</tr>
<tr>
<td>1</td>
<td>naFormatUnit_dBmV Units are displayed in dBmV. 0 dBmV = 0.001 volt</td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
0 - naFormatUnit_dBm

**Examples**
```
meas.FormatUnit(1) = naFormatUnit_dBmV 'Write
units = meas.FormatUnit(1) 'Read
```

**C++ Syntax**
```
HRESULT put_FormatUnit(tagDataFormat format, tagFormatUnit unit)
HRESULT get_FormatUnit(tagDataFormat format, tagFormatUnit* unit)
```

**Interface**
IMeasurement9

---

**Last Modified:**
26-Aug-2008  MX New topic
Write/Read

Frequency Property

**Description**
Sets group delay aperture using a fixed frequency range.

**VB Syntax**
`gdAperture.Frequency = value`

**Variable**
*(Type)* - Description

- `gdAperture` A GroupDelayAperture *(object)*
- `value` *(Double)* Frequency range (in Hz) to use for the aperture setting.

**Return Type**
Double

**Default**
Frequency range that equates to 11 points.
This can be changed to two points with a preference setting.

**Examples**

```vbnet
gdAperture.Frequency = 1e6 'Write
aperture = gdAperture.Frequency 'Read
```

**C++ Syntax**

```c++
HRESULT get_Frequency(double Frequency *pVal)
HRESULT put_Frequency(double Frequency newVal)
```

**Interface**
IGroupDelayAperture

---

Last Modified:

23-Feb-2010  MX New topic
Frequency Property

**Description**
Sets or returns the frequency associated with a Power Sensor CalFactor Segment or
Sets or returns the frequency associated with a Power Loss Segment.

**VB Syntax**

```
object.Frequency = value
```

**Variable** *(Type)* - Description

- **object**
  One of the following objects:
  - `PowerSensorCalFactorSegment`
  - `PowerSensorCalFactorSegmentPMAR`
  - `PowerLossSegment`
  - `PowerLossSegmentPMAR`

- **value** *(double)* – Frequency in units of Hz. This can be any non-negative value (limited by the maximum value of double).

**Return Type**
Double

**Default**
0

**Examples**

```
seg.Frequency = 6e9 'Write
freq = seg.Frequency 'Read
```

**C++ Syntax**

```
HRESULT put_Frequency(double newVal);
HRESULT get_Frequency(double *pVal);
```

**Interface**
One of the above objects.

---

Last Modified:

25-Aug-2009 Added PMAR
**FrequencyCenter Property**

**Description**
Sets and returns the center frequency of the main tones. Use with IMD sweep types:

- `naIMDToneCWSweep`
- `naIMDTonePowerSweep`
- `naIMDDeltaFrequencySweep`

**VB Syntax**

```vbnet
object.FrequencyCenter = value
```

**Variable (Type) - Description**

- `object` A SweptIMD or IMSpectrum Object
- `value` (Double) Tone center frequency in Hz. Both the F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**
Double

**Default**
1.0 GHz

**Examples**

```vbnet
imd.FrequencyCenter = 2e9 'Write

value = imd.FrequencyCenter 'Read
```

**C++ Syntax**

- `HRESULT get_FrequencyCenter(double *pVal)`
- `HRESULT put_FrequencyCenter(double *pVal)`

**Interface**
SweptIMD
IMSpectrum

---

Last Modified:
19-Aug-2008    MX New topic
## FrequencyCenterCenter Property

**Description**  
Sets and returns the sweep center frequency when sweeping the main tones. Use with sweep type = `naIMDToneCenterFreqSweep`.

**VB Syntax**  
```vbnet
imd.FrequencyCenterCenter = value
```

**Variable**  
(Type) - Description

- `imd`  
  A SweptIMD Object

- `value`  
  (Double) Center frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**  
Double

**Default**  
13.255 GHz

**Examples**  
```vbnet
imd.FrequencyCenterCenter = 10e9 'Write
```

```vbnet
value = imd.FrequencyCenterCenter 'Read
```

**C++ Syntax**  
```cpp
HRESULT get_FrequencyCenterCenter(double *pVal)
HRESULT put_FrequencyCenterCenter(double newVal)
```

**Interface**  
ISweptIMD

---

**Last Modified:**  
19-Aug-2008  
MX New topic
## FrequencyCenterSpan Property

**Description**
Sets and returns the frequency span when sweeping the main tones. Use with sweep type = naIMDToneCenterFreqSweep.

**VB Syntax**
```vbnet
imd.FrequencyCenterSpan = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>imd</code></td>
<td>A SweptIMD Object</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>value</code></td>
<td>(Double) Frequency span in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
26.489 GHz

**Examples**
```vbnet
imd.FrequencyCenterSpan = 10e9 'Write

value = imd.FrequencyCenterSpan 'Read
```

**C++ Syntax**
```csharp
HRESULT get_FrequencyCenterSpan(double *pVal)
HRESULT put_FrequencyCenterSpan(double newVal)
```

**Interface**
ISweptIMD

---

**Last Modified:**
19-Aug-2008   MX New topic
Write/Read

**FrequencyCenterStart Property**

**Description**
Sets and returns the start frequency when sweeping the main tones. Use with sweep type = naIMDToneCenterFreqSweep.

**VB Syntax**
```vbnet
imd.FrequencyCenterStart = value
```

**Variable** *(Type)* - Description
- **imd** A SweptIMD Object
- **value** (Double) Start frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Return Type**
Double

**Default**
10.5 MHz

**Examples**
```vbnet
imd.FrequencyCenterStart = 20e6  'Write
value = imd.FrequencyCenterStart  'Read
```

**C++ Syntax**
```cpp
HRESULT get_FrequencyCenterStart(double *pVal)
HRESULT put_FrequencyCenterStart(double newVal)
```

**Interface**
ISweptIMD

---

Last Modified:

19-Aug-2008

MX New topic
### FrequencyCenterStop Property

**Description**
Sets and returns the stop frequency when sweeping the main tones. Use with sweep type = \texttt{naIMDToneCenterFreqSweep}.

**VB Syntax**
\[
\texttt{imd.FrequencyCenterStop} = \texttt{value}
\]

**Variable**
\begin{itemize}
  \item \texttt{imd} \hspace{1cm} A SweptIMD Object
  \item \texttt{value} \hspace{1cm} (Double) Stop frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.
\end{itemize}

**Return Type**
Double

**Default**
26.4995 MHz

**Examples**
\[
\texttt{imd.FrequencyCenterStop} = 20e9 \quad \text{'Write}
\]
\[
\texttt{value} = \texttt{imd.FrequencyCenterStop} \quad \text{'Read}
\]

**C++ Syntax**
\begin{verbatim}
HRESULT get_FrequencyCenterStop(double *pVal)
HRESULT put_FrequencyCenterStop(double newVal)
\end{verbatim}

**Interface**
ISweptIMD

---

**Last Modified:**
19-Aug-2008

MX New topic
Read / Write

FrequencyList Property

Description

VB Syntax  

\[ \text{guidedCal.FrequencyList} = \text{value} \]

Variable  

(Type) - Description

*guidedCal*  

*GuidedCalibration* (object)

*value*  

(Variant) -

Return Type  

Variant

Default

Examples

```vbnet
Dim value
value = MySMC.FrequencyList
```

C++ Syntax

```cpp
HRESULT get_FrequencyList(Variant *freqList)
HRESULT put_FrequencyList(Variant freqList)
```

Interface  

IGuidedCalibration
**Write/Read**

**About Frequency Offset**

**FrequencyOffsetDivisor Property Superseded**

### Description
This method is replaced by properties on the [FOMRange Object](#).

Specifies (along with [FrequencyOffsetMultiplier](#)) the value to multiply by the stimulus.

See other Frequency Offset properties.

### VB Syntax

```vbnet
object.FrequencyOffsetDivisor = value
```

### Variable

**Type** - Description

- `object` Channel *(object)*
- `or`
- `CalSet` *(object)* - Read-only property

**value** *(Double)* - Divisor value. Range is 1 to 1000

### Return Type

Double

### Default

1

### Examples

```vbnet
chan.FrequencyOffsetDivisor = 2 'Write
fOffsetDiv = chan.FrequencyOffsetDivisor 'Read
```

### C++ Syntax

```cpp
HRESULT get_FrequencyOffsetDivisor(double*pval)
HRESULT put_FrequencyOffsetDivisor(double newVal)
```

### Interface

- [IChannel2](#)
- [CalSet3](#)
FrequencyOffsetFrequency Property  Superseded

Description  This method is replaced by properties on the FOMRange Object.

Specifies an absolute offset frequency in Hz. For mixer measurements, this would be the LO frequency. See other Frequency Offset properties.

VB Syntax  

```
object.FrequencyOffsetFrequency = value
```

Variable  (Type) - Description

- **object**  Channel *(object)*
  or  CalSet *(object)* - Read-only property

- **value**  (Double) - Offset value. Range is +/- 1000 GHz. (Offsets can be positive or negative.)

Return Type  Double

Default  0 Hz

Examples  

```
chan.FrequencyOffsetFrequency = 2 'Write

fOffsetFreq = chan.FrequencyOffsetFrequency 'Read
```

C++ Syntax  

```
HRESULT get_FrequencyOffsetFrequency(double*pval)
HRESULT put_FrequencyOffsetFrequency(double newVal)
```

Interface  IChannel2  |CalSet3
FrequencyOffsetMultiplier Property  Superseded

**Description**
This method is replaced by properties on the FOMRange Object. Specifies (along with FrequencyOffsetDivisor) the value to multiply by the stimulus. See other Frequency Offset properties.

**VB Syntax**
```
object.FrequencyOffsetMultiplier = value
```

**Variable**
- **object** - Channel (object)
- or
  - CalSet (object) - Read-only property
- **value** - (Double) - Multiplier value. Range is 1 to 1000

**Return Type**
Double

**Default**
1

**Examples**
```
chan.FrequencyOffsetMultiplier = 2  'Write
fOffsetMult = chan.FrequencyOffsetMultiplier  'Read
```

**C++ Syntax**
```
HRESULT get_FrequencyOffsetMultiplier (double*pval);
HRESULT put_FrequencyOffsetMultiplier (double newVal);
```

**Interface**
|Channel2
|CalSet3
**Write/Read**

**FrequencyOffsetRangeForCalComputations Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Specifies the FOM frequency range to use when performing calibration.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>pref.FrequencyOffsetRangeForCalComputations = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>pref</code></td>
<td>A <code>Preferences</code> object</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Enum as <code>NACalFOMRange</code>) - Choose from:</td>
</tr>
<tr>
<td>0</td>
<td><code>naCalFOMRangeAuto</code> - All other calibration situations.</td>
</tr>
<tr>
<td>1</td>
<td><code>naCalFOMRangePrimary</code> - Used for calibrating at the mmWave frequencies when NOT using a test set. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Enum</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td><code>naCalFOMRangeAuto</code></td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>pref.FrequencyOffsetRangeForCalComputations = naCalFOMRangePrimary</code></td>
</tr>
<tr>
<td></td>
<td><code>calPref = pref.FrequencyOffsetRangeForCalComputations</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_FrequencyOffsetRangeForCalComputations(tagNACalFOMRange * val);</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_FrequencyOffsetRangeForCalComputations((tagNACalFOMRange val);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IPreferences10</code></td>
</tr>
</tbody>
</table>

Last Modified:

16-Sep-2009   MX New topic
Write/Read

FrequencyOffsetCWOverride Property  Superseded

This method is replaced by properties on the FOMRange Object.

Establishes a fixed (CW) stimulus frequency while measuring the Response over a swept frequency range. For example, a fixed-frequency PNA stimulus may be applied to the RF input of a mixer whose local oscillator (LO) is being swept. Because the IF output of the mixer will be swept, the PNA receivers must also be swept.

See other Frequency Offset properties.

VB Syntax

object.FrequencyOffsetCWOverride = value

Variable  (Type) - Description

object  Channel (object)

or

CalSet (object) - Read-only property

value  (Enum as NaStates) - Choose from:

naOFF (0) - Turns CW override OFF
naON (1) - Turns CW override ON

Return Type  Enum

Default  0 Hz

Examples

chan.FrequencyOffsetCWOverride = 1 'Write

fOffsetOV = chan.FrequencyOffsetCWOverride 'Read

C++ Syntax

HRESULT get_FrequencyOffsetCWOverride (tagNAStates *pstate)
HRESULT put_FrequencyOffsetCWOverride (tagNAStates newState)

Interface

IChannel2
|CalSet3
About Frequency Offset

FrequencyOffsetState Property  Superseded

Description
This method is replaced by properties on the FOMRange Object.

Enables Frequency Offset on ALL measurements that are present on the active channel. This immediately causes the source and receiver to tune to separate frequencies. The receiver frequencies are specified with other channel and offset settings. To make the stimulus settings, use Channel Start, Stop Frequency properties. See other Frequency Offset properties.

Tip: To avoid unnecessary errors, first make other frequency offset settings. Then turn Frequency Offset ON.

VB Syntax
object.FrequencyOffsetState = value

Variable (Type) - Description

object Channel (object)

or
CalSet (object) - Read-only property

value (Enum as NaStates) - Choose from:

naOFF (0) - Turns Frequency Offset OFF

naON (1) - Turns Frequency Offset ON

Return Type
Enum

Default
naOFF (0)

Examples
chan.FrequencyOffsetState = naON  'Write

Foffset = chan.FrequencyOffsetState  'Read

C++ Syntax
HRESULT FrequencyOffsetState (tag NAStates *pState);
HRESULT FrequencyOffsetState (tag NAStates newState)

Interface
|Channel2
|CalSet3
**Gain Property**

**Description**

Returns the Gain result of the PNOP marker search.

\[ \text{Pnop Gain} = \text{Pnop Out} - \text{Pnop In.} \]

**VB Syntax**

\[ \text{gain} = \text{pnop.Gain} \]

**Variable**

- **Type** - Description
  - `gain` (double) - Variable to store returned value
  - `pnop` A PNOP (object)

**Return Type**

Double

**Default**

Not applicable

**Examples**

\[ \text{gain} = \text{pnop.Gain} \]

See example program

**C++ Syntax**

HRESULT get_Gain(double* pNewVal)

**Interface**

IPNOP

---

**Last Modified:**

19-Feb-2010  MX New topic
### GainLinear Property

**Description**  Returns the Linear Gain result of a PSat marker search.

Gain Linear = Marker 1 - Y-axis value MINUS X-axis value

**VB Syntax**  

```vb
gainLin = pSat.GainLinear
```

#### Variable (Type) - Description

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gainLin</td>
<td>double</td>
<td>Variable to store returned value</td>
</tr>
<tr>
<td>pSat</td>
<td>PSaturation (object)</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**  Double

**Default**  Not applicable

**Examples**  

```
gainLin = pSat.GainLinear 'Read
```

[See example program](#)

**C++ Syntax**  

```c++
HRESULT get_GainLinear(double* pNewVal)
```

**Interface**  IPSaturation

---

**Last Modified:**

19-Feb-2010  MX New topic
### GainMax Property

**Description**

Returns the GainMax result of the PNOP or PSAT marker search.

Gain Max = \( \text{PMax Out} - \text{PMax In} \)

**VB Syntax**

\[ \text{gainMax} = \text{pMarker}.\text{GainMax} \]

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gainMax</td>
<td>double</td>
<td>Variable to store returned value</td>
</tr>
<tr>
<td>pMarker</td>
<td>PNOP (object) or PSaturation (Object)</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**

Double

**Default**

Not applicable

**Examples**

\[ \text{gainMax} = \text{pMarker}.\text{GainMax} \quad \text{'Read} \]

See example program

**C++ Syntax**

\[ \text{HRESULT get\_GainMax(double* pNewVal)} \]

**Interface**

IPNOP or IPSaturation

---

Last Modified:

19-Feb-2010  MX New topic
### GainSaturation Property

**Description**
Returns the GainSaturation result of the PSAT marker search.

Gain Sat = Psat Out - Psat In

**VB Syntax**
\[ GainSat = pSat.GainSaturation \]

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GainSat</td>
<td>(double) - Variable to store returned value</td>
</tr>
<tr>
<td>pSat</td>
<td>A PSaturation (Object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not applicable

**Examples**

\[ GainSat = pSat.GainSaturation \]

[See example program](#)

**C++ Syntax**

```cpp
HRESULT get_GainSaturation(double* pNewVal)
```

**Interface**
IPSaturation

---

**Last Modified:**
22-Feb-2010  MX New topic
## FrequencySpan Property

**Description**
Sets or returns the frequency span of the channel.
Sets or returns the frequency span of the segment.

**VB Syntax**
```
object.FrequencySpan = value
```

**Variable**
- **object**
  - A Channel *(object)*
  - or
  - A Segment *(object)*

- **value** *(double)* - Frequency span in Hertz. Choose any number between the `minimum` and `maximum` frequencies of the analyzer.

**Return Type**
Double

**Default**
Full frequency span of the analyzer

**Examples**
```
chan.FrequencySpan = 4.5e9 'sets the frequency span of a linear sweep for the channel object -Write

freqspan = chan.FrequencySpan 'Read
```

**C++ Syntax**
```
HRESULT get_FrequencySpan(double *pVal)
HRESULT put_FrequencySpan(double newVal)
```

**Interface**
IChannel
ISegment
**Shape Property**

**Description**
Specifies the shape of the gate filter.

**VB Syntax**
`gat.Shape = value`

**Variable**
- **(Type)**: A Gating (object)
- **(Type)**: (enum NAGateShape)
  - Choose from:
    0 - naGateShapeMaximum
    1 - naGateShapeWide
    2 - naGateShapeNormal
    3 - naGateShapeMinimum

**Return Type**
NAGateShape

**Default**
2 - Normal

**Examples**
```
gat.Shape = naGateShapeMaximum 'Write
```
```
filterShape = gat.Shape 'Read
```

**C++ Syntax**
- HRESULT get_Shape(tagNAGateShape *pVal)
- HRESULT put_Shape(tagNAGateShape newVal)

**Interface**
IGating
Type (gate) Property

**Description**
Specifies the type of gate filter used.

**VB Syntax**
gat.Type = value

**Variable**
* (Type) - Description

gat - A Gating (object)

*value* (enum NAGateType) - Choose from:

0 - naGateTypeBandpass - Includes (passes) the range between the start and stop times.
1 - naGateTypeNotch - Excludes (attenuates) the range between the start and stop times.

**Return Type**
NAGateType

**Default**
Bandpass

**Examples**

<table>
<thead>
<tr>
<th>Write</th>
<th>Read</th>
</tr>
</thead>
<tbody>
<tr>
<td>gate.Type = naGateTypeNotch</td>
<td>filterType = gate.Type</td>
</tr>
</tbody>
</table>

**C++ Syntax**

HRESULT get_Type(tagNAGateType *pVal)
HRESULT put_Type(tagNAGateType newVal)

**Interface**
IGating
## GPIBAddress Property

**Description**
Sets and returns the PNA GPIB address on the talker/listener bus.

**VB Syntax**
```vbnet
app.GPIBAddress (bus) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
<td></td>
</tr>
<tr>
<td><code>bus</code></td>
<td>(Short Integer) GPIB bus. MUST be set to 0 - the talker/listener bus.</td>
<td></td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Short Integer) GPIB Address on the PNA. Choose a value between 0 and 30.</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**
Short Integer

**Default**
16

**Examples**
```
address=app.GPIBAddress(0) 'Read
app.GPIBAddress(0)=16 'Write
```

**C++ Syntax**
```cpp
HRESULT get_GPIBAddress(short busIndex, short* address);
HRESULT put_GPIBAddress(short busIndex,short address);
```

**Interface**
IApplication8

---

**Last Modified:**

16-Mar-2011    Add talker/listener
GPIBMode Property

Description: Changes the analyzer to a GPIB system controller or a talker/listener on the bus. The analyzer must be the controller if you want to use it to send commands to other instruments. The analyzer must be a talker/listener if you want to send it commands from another PC.

Note: This command has no affect in PNAs with dedicated Controller and Talker/Listener GPIB connectors.

VB Syntax: app.GPIBMode value

Variable: (Type) - Description
- app: An Application (object)
- value: (enum NAGPIBMode) - Choose either:
  0 - naTalkerListener - the analyzer is a talker/listener
  1 - naSystemController - the analyzer is the system controller

Return Type: Long Integer
Default: 0 - naTalkerListener

Examples:
- app.GPIBMode = naTalkerListener 'Write
- mode = app.GPIBMode 'Read

C++ Syntax:
- HRESULT get_GPIBMode(tagGPIBModeEnum* eGpibMode)
- HRESULT put_GPIBMode(tagGPIBModeEnum eGpibMode)

Interface: IApplication

Last Modified: 10-Nov-2008 Removed link to obsolete topic.
GPIBPortCount Property

Description
Returns the number of GPIB ports that are present on the PNA rear-panel.

VB Syntax
\[ value = cap.GPIBPortCount \]

Variable (Type) - Description
\[ value \] (Long) - Variable to store the returned integer value of the number of GPIB ports.
\[ cap \] A Capabilities (object)

Return Type
Long

Default
Not Applicable

Examples
\[ value = cap.GPIBPortCount \]

C++ Syntax
HRESULT get_GPIBPortCount(long * gpibPorts);

Interface
ICapabilities3

Last Modified:
23-Oct-2008   Removed 1.1 GHz description
Grid Property

**Description**  Set and return the inner lines of all grid in all windows, and the grid frame in inactive windows for the PNA display or hardcopy print.

**VB Syntax**  
```
colors.Grid = value
```

**Variable**  

- **colors**  A [ComColors](#) (object)
- **value**  (Long Integer) - RGB color of the Grid pen.

Convert the three RGB colors to an integer as follows:

```
RGB = R+(G*2^8)+(B*2^16)
```

To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

**Return Type**  Long

**Default**  
- Display = 175,175,175
- Print = 0,0,0 (Black)

**Examples**

R = 10
G = 10
B = 10

```
RGB = R+(G*2^8)+(B*2^16)
```

```
colors.Grid = RGB 'Write
color = colors.Grid 'Read
```

**C++ Syntax**

```
HRESULT get_Grid(long* pVal);
HRESULT put_Grid(long newVal);
```

**Interface**  IComColors

---

Last Modified:  
7-Aug-2009  MX New topic
GridLineType Property

Description
Set and return whether the grid lines are displayed in solid or dotted lines for all open windows. Grid lines are returned to solid when the PNA is Preset.

VB Syntax
app.GridLineType = value

Variable
(Type) - Description
app A Application Object (object)
value (Enum as naLineType) - Choose from:
0 - naLineTypeSolid
1 - naLineTypeDotted

Return Type
Enum

Default
naLineTypeSolid

Examples
app.GridLineType = naLineTypeSolid 'Write
grid = app.GridLineType 'Read

C++ Syntax
HRESULT get_GridLineType(tag naLineType* pVal);
HRESULT put_GridLineType(tag naLineType newVal);

Interface
IApplication

Last Modified:
16-Mar-2010 MX New topic
HandshakeEnable Property

**Description**

Turns handshake ON/OFF.

To enable handshake, the main trigger enable must ALSO be set using `Enable`.

When ON, PNA acquisition waits indefinitely for the input line to be asserted before continuing with the acquisition.

*Note:* Use on PNA-X ONLY. Other models do NOT have an Aux Input.

**VB Syntax**

```
 auxTrig.HandshakeEnable = state
```

**Variable**

*Type* - Description

- `auxTrig` An `AuxTrigger` *(object)*
- `state` *(boolean)* -
  - `True` - Handshake enabled
  - `False` - Handshake NOT enabled

**Return Type**

Boolean

**Default**

False

**Examples**

```
 auxTrig.HandshakeEnable = True 'Write

value = auxTrig.HandshakeEnable 'Read
```

**C++ Syntax**

```
HRESULT get_HandshakeEnable(VARIANT_BOOL * enable);
HRESULT put_HandshakeEnable(VARIANT_BOOL enable);
```

**Interface**

`IAuxTrigger`

---

Last Modified:

- 5-Sep-2008    Added Note
- 14-Dec-2006    MX New topic
HasItem Property

Description
Returns a value indicating whether the specified external devices is configured.

VB Syntax
value = extDevices.HasItem(name:driver)

Variable (Type) - Description
value (Boolean) Variable to store one of the following returned values.
False - Item (name:driver) is NOT present in the External Devices collection.
True - Item (name:driver) IS present in the External Devices collection.

extDevices An ExternalDevices (collection)

name:driver (String) External Device for which to search the collection in the form (name:driver) where:
- name = name of the external device
- driver = See a list of supported drivers

Return Type Boolean

Default Not Applicable

Examples
has = extDevices.HasItem('mysource':ESG) 'Read

NOTE: the example on the collections page does not include the 'driver as part of this commands

C++ Syntax
HRESULT get_HasItem( VARIANT Index, BOOL *pVal);

Interface IExternalDevices

Last Modified:
31-Jul-2009 MX New topic
# HighestOrderProduct Property

**Description**
Returns the highest order product that can be measured by SweptIMD.

**VB Syntax**
```vbnet
value = imd.HighestOrderProduct
```

**Variable**
- **Type**: Long Integer
- **Description**: Variable in which to store the returned value.

```vbnet
value
```

*imd*  
A [SweptIMD](#) Object

**Return Type**
Integer

**Default**
Always returns 9

**Examples**
```vbnet
value = imd.HighestOrderProduct  'Read
```

**C++ Syntax**
```cpp
HRESULT get_HighestOrderProduct(long *pVal)
```

**Interface**
ISweptIMD

---

**Last Modified:**
- 19-Sep-2008
- MX New topic
## ID Property

**Description**
Returns the test set ID number. For GPIB testsets, the ID is equivalent to the GPIB address. For testset I/O testsets, the ID is the base address of the testset (0 for the first testset, 1 for the second, and so on).

**VB Syntax**
```vbnet
value = tset.ID
```

**Variable (Type) - Description**
- `value` (Long) variable to store the returned information.
- `tsets(1)` A `TestsetControl` object.
  OR
  An `E5091Testset` object.

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```vbnet
value = testset1.ID
```

See E5091A Example Program
See External Testset Program

**C++ Syntax**
```cpp
HRESULT get_ID(long *idNumber);
```

**Interface**
- `ITestsetControl`
- `IE5091Testset`
Read-only

IDString Property

Description
Returns the ID of the analyzer, including the Model number, Serial Number, and the Software revision number.

Note: Beginning with Rev 6.01, this command now returns the software revision with 6 digits instead of 4. For example, A.06.01.02.

VB Syntax
value = app.IDString

Variable (Type) - Description
app  An Application (object)
value (string) - variable to contain the returned ID string

Return Type
String

Default
Not Applicable

Examples
id = app.IDString

C++ Syntax
HRESULT IDString(BSTR* IDString)

Interface
IApplication
### IFBandwidthOption Property

**Description**
Enables the IFBandwidth to be set on individual sweep segments. This property must be set True before `seg.IFBandwidth = value` is sent. Otherwise, this command will be ignored.

**VB Syntax**
```
segs.IFBandwidthOption = value
```

**Variable (Type) - Description**

- **segs**
  A Segments collection (`object`)

- **value**
  (boolean)
  - **True** - Enables variable IFBandwidth setting for segment sweep
  - **False** - Disables variable IFBandwidth setting for segment sweep

**Return Type**
Boolean

**Default**
False

**Examples**
```
segs.IFBandwidthOption = True  'Write
IFOption = IFBandwidthOption  'Read
```

**C++ Syntax**
- `HRESULT get_IFBandwidthOption(VARIANT_BOOL *pVal)`
- `HRESULT put_IFBandwidthOption(VARIANT_BOOL newVal)`

**Interface**
`ISegments`
IFBandwidth Property

Description
Sets or returns the IF Bandwidth of the channel.
Sets or returns the IF Bandwidth of the segment.
Returns the IF Bandwidth used in the Cal Set

VB Syntax
object. IFBandwidth = value

Variable (Type) - Description
object
Channel (object)
or
Segment (object)
or
CalSet (object) - Read-only property

value (double) - IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. (Click to see the lists.) If an invalid number is specified, the analyzer will round up to the closest valid number.

Return Type
Double

Default
See Preset IFBW for your PNA model.

Examples
chan. IFBandwidth = 3e3 'sets the IF Bandwidth of for the channel object to 3 kHz. -Write
seg. IFBandwidth = 5 'sets the IF Bandwidth of the segment to 5 Hz. -Write

ifbw = chan. IFBandwidth -Read

C++ Syntax
HRESULT get_IFBandwidth(double *pVal);
HRESULT put_IFBandwidth(double newVal);

Interface
IChannel
ISegment
ICalSet3
### IFDenominator Property

**Description**
Sets or returns the denominator value of the IF Fractional Multiplier. Only applies to 2 stage mixers. If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

**VB Syntax**

```vbnet
conv.IFDenominator = value
```

**Variable**

- **Type** - Description
  - `conv` A `Converter` (object)
  - `value` (long) IF Denominator value.

**Return Type**
Long

**Default**
1

**Examples**

```
Print mixer.IFDenominator 'prints the value of the IFDenominator
```

**C++ Syntax**

```
HRESULT get_IFDenominator(long *pVal)
HRESULT put_IFDenominator(long newVal)
```

**Interface**
IConverter4

---

Last Modified:

29-Oct-2010  MX New topic
## IFFilterSampleCount Property

### Description

**Note:** This setting applies only to the E836X Opt. H11.

Sets or retrieves the number of taps in the IF filter when the IFFilterSource property is set to \(\text{IFFilterSourceManual}\.\)

### VB Syntax

\[ \text{IfConfig.IFFilterSampleCount} = \text{value} \]

### Variable (Type) - Description

**IfConfig**  
\(\text{IFConfiguration} \) (object)

**value**  
\(\text{(long)}\) – The IF filter sample count. The minimum and maximum allowed values for this property can vary by model number, but can be queried using the MinimumIFFilterSampleCount and MaximumIFFilterSampleCount properties.

### Return Type

Long Integer

### Default

PNA Model number dependent

### Examples

\[\text{App.ActiveChannel.IFConfiguration.IFFilterSampleCount} = 200 \ \text{Write}\]

\[\text{variable} = \text{App.ActiveChannel.IFConfiguration.IFFilterSampleCount} \ \text{Read}\]

See an example program

### C++ Syntax

HRESULT get_IFFilterSampleCount( long * pSampleCount )

HRESULT put_IFFilterSampleCount( long sampleCount )

### Interface

IIFConfiguration2
### IFFilterSamplePeriod Property

**Description**  
**Note**: This setting applies only to the E836X Opt. H11.

Sets or returns the IF filter sample period time. This time is only used if the `IFFilterSamplePeriodMode` is set to `naManual`.

**VB Syntax**  
`IfConfig.IFFilterSamplePeriod = value`

**Variable (Type) - Description**

- **IfConfig**  
  `IFConfiguration` (object)

- **value**  
  (double) – The sample period time in seconds. Valid sample period times can be queried using the `IFFilterSamplePeriodList` property.

**Return Type**
Double

**Default**
PNA Model number dependent.

**Examples**

- `App.ActiveChannel.IFConfiguration.IFFilterSamplePeriod = .000006 'Write`

- `variable = App.ActiveChannel.IFConfiguration.IFFilterSamplePeriod 'Read`

See an example program

**C++ Syntax**

```cpp
HRESULT get_IFFilterSamplePeriod( double * pSamplePeriod );
HRESULT put_IFFilterSamplePeriod( double samplePeriod );
```

**Interface**
`IIFConfiguration2`
**Read-only**

**IFFilterSamplePeriodList Property**

**Description**
Retrieves the list of available IF filter sample periods for the instrument.

*Note:* This setting applies only to the E836X Opt. H11.

**VB Syntax**
```
variable = IfConfig.IFFilterSamplePeriodList
```

**Variable**
*Type* - Description

variable (Array) An array of permissible values that can be passed to the IFFilterSamplePeriod property.

**IfConfig**
IFConfiguration (object)

**Return Type**
Array

**Default**
Not applicable

**Examples**
```
Dim Variable
Variable = App.ActiveChannel.IFConfiguration.IFFilterSamplePeriodList 'Read
MsgBox "First IF Sample Period Value: " & Variable(0)
```

See an example program

**C++ Syntax**
```
HRESULT get_IFFilterSamplePeriodList( SAFEARRAY ** ppSamplePeriodList );
```

**Interface**
IIFConfiguration2
**IFFilterSamplePeriodMode Property**

**Description**
Sets or returns the IF filter sample period mode.

*Note:* This setting applies only to the E836X Opt. H11.

**VB Syntax**

```vbnet
IfConfig.IFFilterSamplePeriodMode = value
```

**Variable**

<table>
<thead>
<tr>
<th><strong>Type</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>IfConfig</td>
<td>IFConfiguration (object)</td>
</tr>
<tr>
<td>value</td>
<td>(enum NAModes) -</td>
</tr>
<tr>
<td></td>
<td>0 - naAuto - IF filter sample period is chosen automatically.</td>
</tr>
<tr>
<td></td>
<td>1 - naManual - the IF filter sample period is the value specified by the IFSamplePeriod property.</td>
</tr>
</tbody>
</table>

**Return Type**
NAModes

**Default**
0 - naAuto

**Examples**

```vbnet
App.ActiveChannel.IFConfiguration.IFFilterSamplePeriodMode = naAuto
'Write

variable = App.ActiveChannel.IFConfiguration.IFFilterSamplePeriodMode
'Read

See an example program
```

**C++ Syntax**

```c
HRESULT get_IFFilterSamplePeriodMode( tagNAModes * pMode);
HRESULT put_IFFilterSamplePeriodMode ( tagNAModes mode );
```

**Interface**
IIFConfiguration2
# IFFilterSource Property

**Description**
Sets or retrieves type of IF filter to be used.

**Note:** This setting applies only to the E836X Opt. H11.

**VB Syntax**
```vbnet
IfConfig.IFFilterSource = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>IFConfiguration (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(NAIFFilterSource)</td>
<td>The type of filter to use. Choose from:</td>
</tr>
<tr>
<td>naIFFilterSourceAuto</td>
<td>The IF filter type is automatically chosen.</td>
</tr>
<tr>
<td>naIFFilterSourceManual</td>
<td>The IF filter is a predetermined shape where the IFFilterSampleCount determines the number of taps in the filter.</td>
</tr>
</tbody>
</table>

**Return Type**
NAIFFilterSource

**Default**
naIFFilterSourceAuto

**Examples**
```vbnet
App.ActiveChannel.IFConfiguration.IFFilterSource = naIFFilterSourceManual 'Write
Variable = App.ActiveChannel.IFConfiguration.IFFilterSource 'Read
```

See an example program

**C++ Syntax**
```c
HRESULT get_IFFilterSource( tagNAIFFilterSource * pFilterSource );
HRESULT put_IFFilterSource(tagNAIFFilterSource filterSource );
```

**Interface**
IIFConfiguration2
IFGainLevel Property

Description  
**Note:** This setting applies only to the E836x and selected N5230A models.
Manually sets the gain level for the specified receiver.

VB Syntax  
`IfConfig.IFGainLevel (id) = value`

Variable  
(Type) - Description
`IfConfig`  An **IFConfiguration** (object)
`id`  Receiver for which to set the gain level. Choose from: 'A', 'B', 'R1', 'R2'.
`value`  (long Integer) Gain Level. Choose from:
0 - Lowest gain setting
1
2 - Highest gain setting

Return Type  Long Integer

Default  0 (Lowest setting)

Examples  
`IfConfig.IFGainLevel("A") = 1`

C++ Syntax  
HRESULT get_IFGainLevel (BSTR IDString, *GainLevel)
HRESULT put_IFGainLevel (BSTR IDString, GainLevel)

Interface  IFConfiguration

Last Modified:
14-Apr-2008  Clarified models
21-Sep-2007  Fixed example
**IFGainMode Property**

**Description**  
*Note:* This setting applies only to the E836x and selected N5230A models.  
Sets the gain state for ALL receivers to Auto or Manual.

**VB Syntax**  
`IfConfig.IFGainMode (id) = value`

**Variable (Type) - Description**

- **IfConfig**  
  An `IFConfiguration` (object)
- **id**  
  Receivers for which to set the state. Choose 'ALL'.
- **value**  
  (enum as NAModes)  
  Choose from:
  - 0 - naAUTO
  - 1 - naMANUAL (use `IFGainLevel Property` to manually set gain level)

**Return Type**  
NAModes

**Default**  
0 - naAUTO

**Examples**  
`IfConfig.IFGainMode("ALL") = naAUTO`

**C++ Syntax**

- HRESULT `get_IFGainMode` (BSTR IDString, NAModes *gainMode)
- HRESULT `put_IFGainMode` (BSTR IDString, NAModes gainMode)

**Interface**  
`IIFConfiguration`

---

Last Modified:

14-Apr-2008  
Clarified models
Write/Read

IFGateEnable Property

Description
Sets or retrieves the state of the IF Gate.

*Note:* This setting applies only to the E836X Opt. H11.

VB Syntax
`IfConfig.IFGateEnable = value`

Variable *(Type)* - Description

`IfConfig` *IFConfiguration (object)*

`value` *(Boolean)* – The state of the IF Gate.

*True* – The IF Gate is in use.

*False* – The IF Gate is not in use.

Return Type
Boolean

Default
False

Examples
`App.ActiveChannel.IFConfiguration.IFGateEnable = True 'Write`

`variable = App.ActiveChannel.IFConfiguration.IFGateEnable 'Read`

See an example program

C++ Syntax
`HRESULT get_IFGateEnable( VARIANT_BOOL * pGateEnabled );`

`HRESULT put_IFGateEnable( VARIANT_BOOL gateEnabled );`

Interface
*IIFConfiguration*
IFNumerator Property

Description
Sets or returns the numerator value of the IF Fractional Multiplier.
Only applies to 2 stage mixers.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax
conv.IFNumerator = value

Variable
(Type) - Description
conv  A Converter (object)
value  (Long) IF Numerator value.

Return Type
long
Default
1

Examples
Print mixer.IFNumerator 'prints the value of the IFNumerator

C++ Syntax
HRESULT get_IFNumerator(long *pVal)
HRESULT put_IFNumerator(long newVal)

Interface
IConverter4

Last Modified:
29-Oct-2010  New topic
**Description**
Sets and returns the IF frequency for ALL receiver paths being used for the specified channel. To set this frequency, `IFFrequencyMode Property` must be set to OFF (Manual).

**VB Syntax**
```
IfConfig.IFFrequency = value
```

**Variable**
(Type) - Description

IfConfig  An IFConfiguration (object)

value  (double) IF Frequency. Use `MaximumIFFrequency` and `MinimumIFFrequency` to determine the range of value for this command.

**Return Type**
Double

**Default**  9 MHz

**Examples**
```
IfConfig.IFFrequency = 9.3e6
```

**C++ Syntax**
```
HRESULT get_IFFrequency (double *pVal);
HRESULT put_IFFrequency (double pVal);
```

**Interface**
IFConfiguration3
Write/Read

IFFrequencyMode Property

Description
Sets and returns method for specifying the way the IF Frequency is determined.

VB Syntax
IfConfig.IFFrequencyMode = value

Variable (Type) - Description
IfConfig An IFConfiguration (object)

value (enum as NAModes) IF Frequency mode. Choose from:

0 - naAUTO The PNA determines the setting for the IF frequency. The IF frequency is based on many PNA settings, including measurement frequency. Therefore, it is NOT possible to read the IF frequency that is being used.

1 - naMANUAL (use IFFrequency Property to manually set frequency.

Return Type Enum

Default 0 naAuto

Examples
IfConfig.IFFrequencyMode = naMANUAL

C++ Syntax
HRESULT get_IFFrequencyMode (tagNAModes* pdspMode);
HRESULT put_IFFrequencyMode (tagNAModes* pdspMode);

Interface IFConfiguration3

Last Modified:
19-Oct-2010 Added Auto note
18-Jun-2007 MX New topic
Write/Read

About Mixer Configuration

IFSideband Property

Description
When two LOs are used, sets or returns whether to select the sum or difference for the IF1 product. (Input + or - LO1 = IF1)

- This setting corresponds to the + buttons on LO1.
- Also set OutputSideband to LOW or HIGH to determine the output frequency of the mixer.
- This setting is ignored when ONE LO is used.
- If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

Note: There is also an IFSideband_Property on the Mixer Interface.

VB Syntax

obj.IFSideband = value

Variable (Type) - Description

obj A Converter Object

value (enum as ConverterSideBand) - Choose from:
0 or naLowSide Minus (-) on the Mixer setup dialog
1 or naHighSide Plus (+) on the Mixer setup dialog

Return Type
Enum as ConverterSideBand

Default
0 - naLowSide

Examples

conv.IFSideband = naLowSide

C++ Syntax

HRESULT get_IFSideband(ConverterSideBand *pVal)
HRESULT put_IFSideband(ConverterSideBand newVal)

Interface
IConverter

Last Modified:
26-Mar-2009

New topic
**IFSideband Property**

**Description**
When two LOs are used, sets or returns whether to select the sum or difference for the IF1 product. (Input + or - LO1 = IF1)

- This setting corresponds to the buttons on LO1.
- Also set **OutputSideband** to LOW or HIGH to determine the output frequency of the mixer.
- This setting is ignored when ONE LO is used.
- If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**Note:** There is also an **IFSideband_Property** on the Converter Object.

**VB Syntax**

```
mixer.IFSideband = value
```

**Variable (Type) - Description**

- **mixer**
  - Type: A Mixer (object)
- **value**
  - Type: (enum as FCASideBand) - Choose from:
    - 0 or **LOW** Minus (-) on the Mixer setup dialog
    - 1 or **HIGH** Plus (+) on the Mixer setup dialog

**Return Type**
Enum as FCASideBand

**Default**
0 - LOW

**Examples**

```
Print mixer.IFSideband 'prints the value of the IFSideband
```

**C++ Syntax**

```
HRESULT get_IFSideband(FCASideBand *pVal)

HRESULT put_IFSideband(FCASideBand newVal)
```

**Interface**
IMixer

---

Last Modified:

- 2-Oct-2008  Clarification
- 4-Mar-2008  Added note.
IFSourcePath Property

Description

Note: This setting applies only to the E836X Opt. H11.

Sets the source path of the specified receiver. An error is returned if <id> is not valid, or if <value> is not valid for the specified <id>.

VB Syntax

IfConfig.IFSourcePath (id) = value

Variable (Type) - Description

IfConfig An IFCOnfiguration (object)

id Receiver for which to set the gain level. Choose from: 'A', 'B', 'R1', 'R2'.

Note: The A and R1 receivers are always switched together. B and R2 are also always switched together. For example, if you specify "A", R1 will also be switched.

value (Enum as NAIFSourcePath) Source path. Choose from:

0 - naNormalIFPath - the PNA decides the appropriate IF input paths.
1 - naExternalIFPath - always use the rear panel IF inputs.

Return Type NAIFSourcePath

Default 0 - naNormalIFPath

Examples IfConfig.IFSourcePath('A') = naNormalIFPath

C++ Syntax

HRESULT get_IFSourcePath (BSTR IDString, naiFSourcePath *IFSourcePath)
HRESULT put_IFSourcePath (BSTR IDString, naiFSourcePath IFSourcePath)

Interface IIFConfiguration
About Mixer Configuration

IFStartFrequency Property

Description
Sets or returns the start frequency value of the mixer IF frequency.
Only applies to 2 stage mixers.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

```vbnet
mixer.IFStartFrequency = value
```

Variable (Type) - Description

- `mixer` A Mixer (object)
- `value` (double) - Frequency in Hertz.

Return Type
Double

Default
Not Applicable

Examples
Print mixer.IFStartFrequency 'prints the value of the IFStartFrequency

C++ Syntax

```cpp
HRESULT get_IFStartFrequency(double *pVal)
HRESULT put_IFStartFrequency(double newVal)
```

Interface
IMixer

Last Modified:
4-Mar-2008  Added note.
**IFStopFrequency Property**

**Description**
Sets or returns the stop frequency value of the mixer IF frequency. Only applies to 2 stage mixers.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

**VB Syntax**

```
mixer.IFStopFrequency = value
```

**Variable**

- **mixer**: A Mixer (object)
- **value**: (double) - IF stop frequency in Hertz.

**Return Type**

Double

**Default**

Not Applicable

**Examples**

Print mixer.IFStopFrequency 'prints the value of the IFStopFrequency

**C++ Syntax**

```
HRESULT get_IFStopFrequency(double *pVal)
HRESULT put_IFStopFrequency(double newVal)
```

**Interface**

IMixer

---

Last Modified:

4-Mar-2008   Added note.
### ImpedanceStates Method

**Description**
Sets the number of impedance states to use during calibrated measurements.

**VB Syntax**

```vbnet
noise.ImpedanceStates = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>noise</code></td>
<td>A <code>NoiseFigure</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) - Impedance states. Choose between 4 and the maximum number allowed by the noise tuner device. If the specified number exceeds the capability of the device, the measurement will use the maximum number of states the device allows.</td>
</tr>
</tbody>
</table>

**Return Type**

Double

**Default**

4

**Examples**

```vbnet
noise.ImpedanceStates = 10 'Write
AvgNoise = noise.ImpedanceStates 'Read
```

**C++ Syntax**

- `HRESULT get_ImpedanceStates(double* pVal)`
- `HRESULT put_ImpedanceStates(double newVal)`

**Interface**

`INoiseFigure`

---

**Last Modified:**

29-May-2007    MN New topic
ImpulseWidth Property

**Description**
Sets or returns the Impulse Width of Time Domain transform windows

**VB Syntax**
`trans.ImpulseWidth = value`

**Variable**
- **trans** (Type) - A Transform (object)
- **value** (double) - Impulse Width in seconds. Range of settings depends on the frequency range of your analyzer.

**Return Type**
Double

**Default**
.98 / Default Span

**Examples**
```
trans.ImpulseWidth = 200e-12 'sets the Impulse width of a transform window -Write
IW = trans.ImpulseWidth 'Read
```

**C++ Syntax**
```
HRESULT get_ImpulseWidth(double *pVal)
HRESULT put_ImpulseWidth(double newVal)
```

**Interface**
ITransform
# IMToneIFBandwidth Property

**Description**
Sets and returns the IF Bandwidth for measurement of the intermodulation products.

**VB Syntax**
```vbnet
imd.IMToneIFBandwidth = value
```

**Variable** (Type) - Description

<table>
<thead>
<tr>
<th>Variable (Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>imd</td>
<td>A SweptIMD Object</td>
</tr>
<tr>
<td>value</td>
<td>(Double) IF Bandwidth in Hz. Choose from: 1</td>
</tr>
</tbody>
</table>

Learn more about setting IFBW for IMD.

If an invalid number is specified, the analyzer will round up to the closest valid number.

**Return Type**
Double

**Default**
1 kHz

**Examples**

```vbnet
imd.IMToneIFBandwidth = 2e3 'Write
```

```vbnet
value = imd.IMToneIFBandwidth 'Read
```

**C++ Syntax**

```c++
HRESULT get_IMToneIFBandwidth(double *pVal)
HRESULT put_IMToneIFBandwidth(double newVal)
```

**Interface**
ISweptIMD

---

Last Modified:

19-Aug-2008 MX New topic
InactiveLabels Property

Description  Set and return the Inactive (not selected) Window Labels for the PNA display or hardcopy print.

VB Syntax  \textit{colors.InactiveLabels} = \textit{value}

Variable  \begin{itemize}  
\item \textit{colors} (Type) - Description
\item \textit{value} (Long Integer) - RGB color of the Inactive Labels pen.
\end{itemize}

\begin{itemize}  
\item Convert the three RGB colors to an integer as follows:
\begin{equation*}
\text{RGB} = R + (G \times 2^8) + (B \times 2^{16})
\end{equation*}
\item To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.
\end{itemize}

Return Type  Long

Default  Display = 160,160,160  
Print = 0,0,0 (Black)

Examples  \begin{itemize}  
\item R = 10
\item G = 10
\item B = 10
\item \text{RGB} = R + (G \times 2^8) + (B \times 2^{16})
\end{itemize}

\textit{colors.InactiveLabels} = \text{RGB} \quad \text{'Write}

\textit{color} = \textit{colors.InactiveLabels} \quad \text{'Read}

C++ Syntax  \begin{itemize}  
\item HRESULT get_InactiveLabels(long* pVal);
\item HRESULT put_InactiveLabels(long newVal);
\end{itemize}

Interface  IComColors

Last Modified:

7-Aug-2009  MX New topic
Include2ndOrderProduct Property

Description
Sets and returns whether to include the second order products in the calibration. These frequencies of these products can be far from the main tones.

VB Syntax
\textit{imd.Include2ndOrderProduct} = \textit{value}

Variable (Type) - Description
\begin{itemize}
  \item \textit{imd} - A \texttt{SweptIMDCal} (object)
  \item \textit{value} - (Boolean) Choose from:
    \begin{itemize}
      \item \texttt{False} - Do NOT include 2nd order products
      \item \texttt{True} - Include 2nd order products
    \end{itemize}
\end{itemize}

Return Type
Boolean

Default
\texttt{False} - Do NOT include 2nd order products

Examples
\begin{itemize}
  \item \texttt{imd.Include2ndOrderProduct} = \texttt{true} 'Write
  \item incl = \texttt{imd.Include2ndOrderProduct} 'Read
\end{itemize}

C++ Syntax
\begin{itemize}
  \item HRESULT get_Include2ndOrderProduct(VARIANT_BOOL * Val)
  \item HRESULT put_Include2ndOrderProduct(VARIANT_BOOL newVal)
\end{itemize}

Interface
ISweptIMD

Last Modified:
9-Sep-2008   MX New topic
IncludeReverseSweep Property

**Description**
Sets whether to include SC12 sweeps during measurements.

**VB Syntax**
`obj.IncludeReverseSweep = bool`

**Variable (Type) - Description**
- `obj`: A `Mixer Interface` pointer to the `Measurement` (object)
- `bool`: (Boolean) -
  - `True`: Include the SC12 (reverse) sweep.
  - `False`: Do NOT Include the SC12 (reverse) sweep.

**Return Type**
Boolean

**Default**
True

**Examples**
`obj.IncludeReverseSweep = True`

**C++ Syntax**
```cpp
HRESULT get_IncludeReverseSweep(VARIANT_BOOL * val);
HRESULT put_IncludeReverseSweep(VARIANT_BOOL val);
```

**Interface**
IMixer12

Last Modified:
12-Feb-2010    MX New topic
## IndexState Property

**Description**
Determines the control of Material Handler connector Pin 20.

**VB Syntax**
handler.IndexState = value

**Variable**
- **(Type)**: Description
- **handler** (object) - A Handler I/O object
- **value** (boolean)
  - False - Pin 20 is controlled by Output Port B6
  - True - Pin 20 is controlled by the Index signal

**Return Type**
Boolean

**Default**
False

**Examples**
- handler.IndexState = False  'Write
- bState = handler.IndexState  'Read

**C++ Syntax**
HRESULT put_IndexState (BOOL *pVal);
HRESULT get_IndexState (BOOL newVal);

**Interface**
IHWMaterialHandlerIO2
### InputA Property - Obsolete

**Description**
This property has NO replacement and no longer works correctly. (Sept. 2004)
Sets a Port Extension value for Receiver A

**VB Syntax**

```vbnet
portExt.InputA = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>portExt</code></td>
<td>A Port Extension (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) - Port Extension value in seconds. Choose any number between -10 and 10</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**

```vbnet
portExt.InputA = 10e-6  'Write
inA = portExt.InputA  'Read
```

**C++ Syntax**

```cpp
HRESULT get_InputA(double *pVal)
HRESULT put_InputA(double newVal)
```

**Interface**
IPortExtension
## InputB Property - Obsolete

**Description**  
This property has NO replacement and no longer works correctly. (Sept. 2004)

Sets the Port Extension value for Receiver B

**VB Syntax**  
`portExt.InputB = value`

**Variable**  
*portExt* - A Port Extension *(object)*  
*value* - Port Extension value in seconds. Choose any number between -10 and 10

**Return Type**  
Double

**Default**  
0

**Examples**  
```vbnet
portExt.InputB = 10e-6 'Write
inB = portExt.InputB 'Read
```

**C++ Syntax**  
```cpp
HRESULT get_InputB(double *pVal)
HRESULT put_InputB(double newVal)
```

**Interface**  
IPortExtension
### InputC Property  Obsolete

**Description**  This property has NO replacement and no longer works correctly. (Sept. 2004)

Sets the Port Extension value for Receiver C

**VB Syntax**  
```vbnet
portExt.InputC = value
```

**Variable**  
**Type** - Description

- `portExt`  A Port Extension (object)
- `value`  (double) - Port Extension value in seconds. Choose any number between -10 and 10

**Return Type**  Double

**Default**  0

**Examples**  
```vbnet
portExt.InputC = 10e-6 'Write

inC = portExt.InputC 'Read
```

**C++ Syntax**  
HRESULT get_InputC(double *pVal)
HRESULT put_InputC(double newVal)

**Interface**  IPortExtension
Write/Read

About Mixer Configuration

InputDenominator Property

Description
Sets or returns the denominator value of the Input Fractional Multiplier.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

\texttt{obj.InputDenominator = value}

Variable (Type) - Description

\textit{obj} A \texttt{Mixer Interface} pointer to the \texttt{Measurement} (object)
Or
A \texttt{Converter Object}

\textit{value} \texttt{(Long)} - Input denominator value.

Return Type
Long

Default
1

Examples
Print mixer.InputDenominator 'prints the value of the InputDenominator

C++ Syntax

\texttt{HRESULT get\_InputDenominator(long \*pVal)}
\texttt{HRESULT put\_InputDenominator(long newVal)}

Interface
IMixer
IConverter

Last Modified:
2-Feb-2009 Added converter
4-Mar-2008 Added note.
**InputFixedFrequency Property**

**Description**
Sets or returns the mixer fixed Input frequency value.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**VB Syntax**
```vbnet
obj.InputFixedFrequency = value
```

**Variable**
- **(Type)** - **Description**
  - `obj` A **Mixer Interface** pointer to the **Measurement** (object)
  - Or
  - A **Converter Object**

  - `value` **(double)** - Input Fixed Frequency in Hertz.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```
mixer.InputFixedFrequency = 1e9
```

**C++ Syntax**
```cpp
HRESULT get_InputFixedFrequency(double *pVal)
HRESULT put_InputFixedFrequency(double newVal)
```

**Interface**
- IMixer6
- IConverter

---

**Last Modified:**
- 2-Feb-2009  Added converter
- 4-Mar-2008  Added note.
IsInputGreaterThanLO Property

Description
Specifies whether to use the Input frequency that is greater than the LO or less than the LO. To learn more, see the mixer setup dialog box help.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax
obj.IsInputGreaterThanLO (LO) = bool

Variable (Type) - Description

obj A Mixer Interface pointer to the Measurement (object)
Or
A Converter Object

LO (Integer) - LO stage number
Choose from 1 (default) or 2

bool (Boolean) -
True - Use the Input that is Greater than the specified LO.
False - Use the Input that is Less than the specified LO.

Return Type
Boolean

Default
True

Examples
mixer.IsInputGreaterThanLO(1) = True

C++ Syntax
HRESULT get_IsInputGreaterThanLO(VARIANT_BOOL * val);
HRESULT put_IsInputGreaterThanLO(VARIANT_BOOL val);

Interface
IMixer2
ICONverter

Last Modified:
2-Feb-2009 Added Converter
4-Mar-2008 Added note.
InputNumerator Property

Description  Sets or returns the numerator value of the Input Fractional Multiplier. If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax  

\[ \text{obj}.\text{InputNumerator} = \text{value} \]

Variable  

(obj) - Description

(obj) A **Mixer Interface** pointer to the **Measurement** (object)
Or
A **Converter Object**

(value) (Long) - Input numerator value.

Return Type  Long

Default  1

Examples  

Print mixer.InputNumerator 'prints the value of the InputNumerator

C++ Syntax  

HRESULT get_InputNumerator(long *pVal)
HRESULT put_InputNumerator(long newVal)

Interface  

IMixer
IConverter

Last Modified:

2-Feb-2009  Added converter
4-Mar-2008  Added note.
InputPower Property

**Description**
Sets or returns the value of the Input Power.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

**VB Syntax**
`obj.InputPower = value`

**Variable** (Type) - Description
- `obj` A Mixer Interface pointer to the Measurement (object)
  Or
  A Converter Object
- `value (double)` - Input power in dBm.

**Return Type**
Double

**Default**
-15 dBm for IMixer
-20 dBm for IConverter

**Examples**
Print mixer.InputPower 'prints the value of the InputPower

**C++ Syntax**
HRESULT get_InputPower(double *pVal)
HRESULT put_InputPower(double newVal)

**Interface**
IMixer
IConverter

---

Last Modified:

- 2-Feb-2009 Added converter
- 4-Mar-2008 Added note.
InputRangeMode Property

**Description**
Sets or returns the Input sweep mode.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**Note:** There is also a **InputRangeMode Property** on the Converter Object.

**VB Syntax**
```
obj.InputRangeMode = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>A <strong>Mixer Interface</strong> pointer to the <strong>Measurement</strong> (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Enum as <strong>MixerRangeMode</strong>) - Input sweep mode. Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - <strong>mixSwept</strong></td>
</tr>
<tr>
<td></td>
<td>1 - <strong>mixFixed</strong></td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
0 - mixSwept

**Examples**
```
mixer.InputRangeMode = mixSwept
```

**C++ Syntax**
```
HRESULT get_InputRangeMode(long *pVal)
HRESULT put_InputRangeMode(long newVal)
```

**Interface**
IMixer6

---

Last Modified:

4-Mar-2008 Added note.
**InputStartFrequency Property**

**Description**
Sets and returns the start frequency value of the mixer Input frequency. If you are changing several mixer configuration settings, you can make all the changes first and then issue the [Calculate](#) and [Apply](#) commands as you would do from the user interface.

**VB Syntax**
```
obj.InputStartFrequency = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>A Mixer Interface pointer to the Measurement (object) Or A Converter Object</td>
</tr>
</tbody>
</table>

| value | (double) - Input start frequency in Hertz. |

**Return Type**
Double

**Default**
Start frequency of the PNA

**Examples**
```
mixer.InputStartFrequency = Start_Freq
```

**C++ Syntax**
```
HRESULT get_InputStartFrequency(double *pVal)
HRESULT put_InputStartFrequency(double newVal)
```

**Interface**
IMixer
IConverter

---

Last Modified:

- 2-Feb-2009 Added converter
- 4-Mar-2008 Added note.
InputStartPower Property

Description
Sets and returns the Start Power value of the mixer Input Power.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

```
obj.InputStartPower = value
```

Variable

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>A Converter Object</td>
</tr>
<tr>
<td>value</td>
<td>(double) - Input start Power in dBm.</td>
</tr>
</tbody>
</table>

Return Type
Double

Default
Start Power of the PNA

Examples

```plaintext
mixer.InputStartPower = 0
```

C++ Syntax

```c++
HRESULT get_InputStartPower(double *pVal)
HRESULT put_InputStartPower(double newVal)
```

Interface
IConverter

Last Modified:

2-Feb-2009 New topic
### InputStopFrequency Property

**Description**
Sets and returns the stop frequency value of the mixer Input frequency.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**VB Syntax**

```vbnet
obj.InputStopFrequency = value
```

**Variable**

- **Type** - Description
  - `obj` A [Mixer Interface](#) pointer to the [Measurement](#) object
  - Or
  - A [Converter Object](#)

- **value** *(double)* - Input stop frequency in Hertz.

**Return Type**
Double

**Default**
Stop frequency of the PNA

**Examples**

```vbnet
mixer.InputStopFrequency = Stop_Freq
```

**C++ Syntax**

```cpp
HRESULT get_InputStopFrequency(double *pVal)
HRESULT put_InputStopFrequency(double newVal)
```

**Interface**
- IMixer
- IConverter

---

**Last Modified:**

- **2-Feb-2009**  Added Converter
- **4-Mar-2008**  Added note.
InputStopPower Property

Description: Sets and returns the Stop Power value of the mixer Input Power.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**VB Syntax**: `obj.InputStopPower = value`

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>obj</code></td>
<td>A Converter Object</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) - Input Stop Power in dBm.</td>
</tr>
</tbody>
</table>

**Return Type**: Double

**Default**: Stop Power of the PNA

**Examples**: `mixer.InputStopPower = 0`

**C++ Syntax**:

```cpp
HRESULT get_InputStopPower(double *pVal)
HRESULT put_InputStopPower(double newVal)
```

**Interface**: IConverter

Last Modified: 2-Feb-2009   New topic
**InternalTestsetPortCount Property**

**Description**  Returns the number of PNA test ports. This does not include the ports on an external test set.

**VB Syntax**  

```
value = cap.InternalTestsetPortCount
```

**Variable (Type) - Description**

- `value` (Long) - Variable to store the returned number of PNA test ports.
- `cap` A `Capabilities` (object)

**Return Type**  Long

**Default**  Not Applicable

**Examples**  

```
value = cap.InternalTestsetPortCount 'Read
```

**C++ Syntax**  

```
HRESULT get_InternalTestsetPortCount(long *numPorts);
```

**Interface**  ICapabilities
Interpolate Correction Property

**Description**
Turns ON and OFF correction interpolation which calculates new error terms when stimulus values change after calibration.

When this property is ON and error correction is being applied, the calibration subsystem attempts to interpolate the error terms whenever the stimulus parameters are changed.

When this property is OFF under the same circumstances, error correction is turned OFF.

**VB Syntax**
```
meas.InterpolateCorrection = value
```

**Variable**
- **meas** (A Measurement object)
- **value** (boolean) - Choose from:
  - **True** - Turns correction interpolation ON
  - **False** - Turns correction interpolation OFF

**Return Type**
Boolean

**Default**
True

**Examples**
```
meas.InterpolateCorrection = False
```
```
calInterpolate = InterpolateCorrection 'Read
```

**C++ Syntax**
```
HRESULT get_InterpolateCorrection(boolean *pVal)
HRESULT put_InterpolateCorrection(boolean newVal)
```

**Interface**
IMeasurement
## Interpolated Property

**Description**

Turns marker Interpolation ON and OFF. Marker interpolation enables X-axis resolution beyond the discrete data values. The analyzer will calculate the x and y-axis data values between discrete data points. Use meas.**Interpolate** to change interpolation of **all** markers in a measurement. This command will override the measurement setting.

**VB Syntax**

```
mark.Interpolated = value
```

**Variable**

**Type** - Description

- **mark**
  - A **Marker** (object)

- **value**
  - **(boolean)**
    - **False** - Turns interpolation OFF
    - **True** - Turns interpolation ON

**Return Type**

Boolean

**Default**

True

**Examples**

```
mark.Interpolated = True 'Write
interpolate = mark.Interpolated 'Read
```

**C++ Syntax**

```
HRESULT get_Interpolated(VARIANT_BOOL *pVal)
HRESULT put_Interpolated(VARIANT_BOOL newVal)
```

**Interface**

IMarker
InterpolateNormalization Property  Superseded

Description  
**Note:** This property is replaced by DoReceiverPowerCal Method.

Turns ON and OFF normalization interpolation which calculates new divisor data when stimulus values change after normalization.

When this property is ON and normalization is being applied, the Normalization algorithm attempts to interpolate the divisor data whenever the stimulus parameters are changed.

When this property is OFF under the same circumstances, normalization is turned OFF.

Normalization is currently supported only on measurements of unratioed power for the purpose of performing a receiver power calibration.

**VB Syntax**  
`meas.InterpolateNormalization = value`

**Variable**  
*(Type)* - Description

`meas` *(object)* - A Measurement object

`value` *(boolean)*

False – Turns normalization interpolation OFF  
True – Turns normalization interpolation ON

**Return Type**  
Boolean

**Default**  
False -OFF

**Examples**  
`meas.InterpolateNormalization = False`  
*Write*

`normalized = meas.InterpolateNormalization`  
*Read*

**C++ Syntax**  
`HRESULT put_InterpolateNormalization(VARIANT_BOOL bState);`

`HRESULT get_InterpolateNormalization(VARIANT_BOOL *bState);`

**Interface**  
IMeasurement
## Interrupt Property

**Description**
Reads the boolean that represents the state of the Interrupt In line (pin 13) on the external test set connector.

**VB Syntax**
```vbnet
value = ExtIO.Interrupt
```

**Variable**
- **Type**: Description
  - `value`: boolean - Variable to store the returned data
  - `ExtIO`: object - An ExternalTestSetIO object

**Return Type**
Boolean
- **False**: indicates the line is being held at a TTL High
- **True**: indicates the line is being held at a TTL Low

**Default**
Not Applicable

**Examples**
```vbnet
value = ExtIO.Interrupt
```

**C++ Syntax**
```cpp
HRESULT get_Interrupt( VARIANT_BOOL* bValue);
```

**Interface**
IHWExternalTestSetIO
### Invert Property

**Description**
Sets whether to invert the polarity of the pulse.

**VB Syntax**

```vbnet
pulse.Invert (n) = value
```

**Variable**  

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pulse</td>
<td>A <strong>PulseGenerator</strong> (object)</td>
</tr>
</tbody>
</table>

- **n** *(Integer)*
  Pulse generator number. Choose from 0 to 4.  
  0 is the generator that pulses the ADC.

- **value**
  - **True** - Invert the pulse generator polarity. This causes the pulse ON time to be active low and OFF be active high.
  - **False** - Do NOT Invert the pulse generator polarity.

**Return Type**

Boolean

**Default**

False

**Examples**

```vbnet
pulse.Invert(1) = True 'Write
value = pulse.Invert(4) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_Invert(VARIANT_BOOL *pVal);
HRESULT put_Invert(VARIANT_BOOL newVal);
```

**Interface**

IPulseGenerator3

---

Last Modified:

8-Oct-2010  MX New topic
# IOConfiguration Property

**Description**
Sets and returns the method of communication and address for the external device.

**VB Syntax**
```vbnet
extDevices.IOConfiguration = value
```

**Variable** *(Type) - Description*
- **extDevices** (Type) - ExternalDevice (object)
- **value** (String) - Configuration path. Any valid VISA resource shown in the IO Configuration field of the External Devices dialog, enclosed in quotes.
  
  Do NOT use the ID string of a PMAR USB power sensor as the resource string. The ID string is returned by the USBPowerMeterCatalog Property

**Return Type**
String

**Default**
" " Empty String

**Examples**
```vbnet
extDevices.IOConfiguration = "TCP/IP::141.121.76.239::inst0::INSTR" Write
value = extDevices.IOConfiguration 'Read
See example program to configure PMAR device
See example program to configure External Source
```

**C++ Syntax**
```cpp
HRESULT get_IOConfiguration( BSTR* value);
HRESULT put_IOConfiguration( BSTR newVal);
```

**Interface**
IExternalDevices

_Last Modified:_
31-Jul-2009   MX New topic
IOEnable Property

**Description**
Sets and returns whether an external device is enabled for IO communication with the PNA.

**VB Syntax**
```vbscript
extDevices.IOEnable = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>extDevices</code></td>
<td>An ExternalDevice (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Boolean) Choose from:</td>
</tr>
<tr>
<td></td>
<td>True - Device is available for IO communication.</td>
</tr>
<tr>
<td></td>
<td>False - Device is NOT available for IO communication.</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**
```vbscript
extDevices.IOEnable = True 'Write
bool = extDevices.IOEnable 'Read
See example program to configure PMAR device
See example program to configure External Source
```

**C++ Syntax**
```c#
HRESULT get_IOEnable( VARIANT_BOOL* value);
HRESULT put_IOEnable( VARIANT_BOOL newVal);
```

**Interface**
IExternalDevices

---

Last Modified:
31-Jul-2009  MX New topic
Read-only

IsContinuous Property

Description
Returns whether or not a channel is in continuous mode. To set the channel to continuous mode, use Continuous Method.

VB Syntax
value = chan.IsContinuous

Variable (Type) - Description
value (boolean) - Choose either:
False - Channel trigger is NOT set to continuous.
True - Channel trigger IS set to continuous.

chan Channel (object)

Return Type
Boolean

Default
Not Applicable

Examples
trig = chan.IsContinuous 'Read

C++ Syntax
HRESULT get_IsContinuous ( VARIANT_BOOL *bContinuous);

Interface IChannel3
IsECALModuleFoundEx Property

Description
This property replaces IsECALModuleFound Property. Returns true or false depending on whether communication was established between the PNA and the specified ECal module.

VB Syntax
modFound = cal.IsECALModuleFoundEx(module)

Variable (Type) - Description
modFound (boolean) - Variable to store the returned test result.
- True - The PNA identified the presence of the specified ECal module.
- False - The PNA did NOT identify the presence of the specified ECal module.

cal (object) - A Calibrator object

module (long integer) - ECal module. Choose from modules 1 through 8

Use GetECALModuleInfoEx to return the model and serial number of each module.

Return Type
Boolean

Default
Not applicable

Examples
Set cal = pna.ActiveChannel.Calibrator
moduleFound = cal.IsECALModuleFoundEx(1)

C++ Syntax
HRESULT get_IsECALModuleFoundEx(long moduleNumber, VARIANT_BOOL *bModuleFound);

Interface
ICalibrator4
IsFrequencyOffsetPresent Property

Description  Returns a value indicating the presence of Frequency Offset Option 080 in the remote PNA.

VB Syntax  

\[ value = cap.IsFrequencyOffsetPresent \]

Variable  

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Boolean) - Variable to store the returned value</td>
</tr>
<tr>
<td></td>
<td>True - Frequency Offset Option 080 is present</td>
</tr>
<tr>
<td></td>
<td>False - Frequency Offset Option 080 is not present</td>
</tr>
</tbody>
</table>

\[ cap \] A Capabilities (object)

Return Type  Boolean

Default  Not Applicable

Examples  

\[ value = cap.isFrequencyOffsetPresent(1) \]  'Read

C++ Syntax  

HRESULT get_IsFrequencyOffsetPresent(VARIANT_BOOL * present);

Interface  ICapabilities
### IsHold Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns whether or not a channel is in hold mode. To set the channel to hold mode, use <a href="#">Hold Method</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>value = chan.IsHold</code></td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(boolean) - Choose either:</td>
</tr>
<tr>
<td></td>
<td>- <code>False</code> - Channel trigger is NOT set to hold.</td>
</tr>
<tr>
<td></td>
<td>- <code>True</code> - Channel trigger IS set to hold.</td>
</tr>
<tr>
<td><code>chan</code></td>
<td>Channel (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>Boolean</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>trig = chan.IsHold</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_IsHold (VARIANT_BOOL *bHold);</td>
</tr>
<tr>
<td>Interface</td>
<td>IChannel3</td>
</tr>
</tbody>
</table>
IsMarkerOn Property

Description
Returns whether or not a marker was used for the specified tuning sweep.

VB Syntax
value = embedLODiag.IsMarkerOn (n)

Variable (Type) - Description
value (Boolean) Variable to store the returned data.

embedLODiag An EmbeddedLODiagnostic (object)

n Tuning sweep number. Use NumberOfSweeps to find the number of sweeps taken.

Return Type (String)

Default Not Applicable

Examples
data= embedLO.IsMarkerOn 3 'read

C++ Syntax
HRESULT IsMarkerOn(long sweep, VARIANT_BOOL* markerOn);

Interface IEmbeddedLODiagnostic

---

Last Modified:
13-Apr-2007 MX New topic
IsolationIncrementAveraging Property

Description
Value by which to increment (increase) the channel's averaging factor during measurement of isolation in an ECal calibration.

Note: If <value> is greater than 1 and the channel currently has averaging turned OFF, averaging will be turned ON only during the isolation measurements and with the averaging factor equal to <value>.

VB Syntax
`cal.IsolationIncrementAveraging = value`

Variable

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>cal</td>
<td>A Calibrator (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Long) Incremental Averaging factor. The maximum averaging factor is 65536 (2^16).</td>
</tr>
</tbody>
</table>

Return Type
Long Integer

Default
8

Examples
```vbnet
oCal.IsolationAveragingIncrement = 16 'Write
avgIncr = oCal.IsolationAveragingIncrement ' Read
```

C++ Syntax
```cpp
HRESULT get_IsolationAveragingIncrement(long *pVal);
HRESULT put_IsolationAveragingIncrement(long newVal);
```

Interface
ICalibrator7

Last Modified:
16-Apr-2007   MX New topic
IsOn Property

**Description**
Sets and returns the ON/OFF state of Embedded LO.

**VB Syntax**

```vbnet
obj.IsOn = value
```

**Variable**

**Type** - Description

- `obj` An **EmbeddedLO** *(object)* or A **ConverterEmbeddedLO**(object)

- `value` *(Boolean)*
  - **False** - Turns Embedded LO OFF
  - **True** - Turns Embedded LO ON

**Return Type** *(Boolean)*

**Default**
False (OFF)

**Examples**

```vbnet
eembedLO.IsOn = True 'write

data= embedLO.IsOn 'read
```

**C++ Syntax**

```c++
HRESULT get_IsOn( VARIANT_BOOL* IsOn);
HRESULT put_IsOn( VARIANT_BOOL IsOn);
```

**Interface**

IEembeddedLO

---

**Last Modified:**

- **12-Aug-2009**  Added ConvEmbedLO object
- **18-Apr-2007**  MX New topic
IsReceiverStepAttenuatorPresent Property

**Description**
Returns a value indicating the presence of Receiver step attenuators in the remote PNA.

**VB Syntax**

\[\text{value} = \text{cap}.\text{IsReceiverStepAttenuatorPresent}(n)\]

**Variable**

- **value** *(Boolean)* - Variable to store the returned value.
  - True - Receiver step attenuators are present.
  - False - Receiver step attenuators are not present.
- **cap** A [Capabilities](#) *(object)*
- **n** *(Long)* - Port number to query for step attenuators

**Return Type**
Boolean

**Default**
Not Applicable

**Examples**

\[\text{value} = \text{cap}.\text{IsReceiverStepAttenuatorPresent}(1)\]

**C++ Syntax**

```cpp
HRESULT get_IsReceiverStepAttenuatorPresent(long portNumber, VARIANT_BOOL *present );
```

**Interface**
ICapabilities
### IsReferenceBypassSwitchPresent Property

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Returns a value indicating the presence of a Reference Bypass Switch in the remote PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>value = cap.IsReferenceBypassSwitchPresent(n)</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>Type</strong> - Description</td>
</tr>
<tr>
<td></td>
<td><code>value</code> (Boolean) - Variable to store the returned value.</td>
</tr>
<tr>
<td></td>
<td>True - Reference Bypass Switch is present.</td>
</tr>
<tr>
<td></td>
<td>False - Reference Bypass Switch is not present.</td>
</tr>
<tr>
<td></td>
<td><code>cap</code> - A <code>Capabilities</code> (object)</td>
</tr>
<tr>
<td></td>
<td><code>n</code> (Long) - port number to query for reference bypass switch</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Boolean</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>value = cap.IsReferenceBypassSwitchPresent(1)</code> 'Read'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_IsReferenceBypassSwitchPresent(long portNumber, VARIANT_BOOL * present );</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ICapabilities</td>
</tr>
</tbody>
</table>
Read-only

**IsSParameter Property**

**Description**  Returns true if measurement represents an S-Parameter

**VB Syntax**  
```
value = meas.IsSparameter
```

**Variable (Type) - Description**

- **meas**  A Measurement (object)
- **value (Boolean)**
  - True - measurement is an S-Parameter
  - False - measurement is NOT an S-Parameter

**Return Type**  Boolean

**Default**  True

**Examples**  
```
print app.IsSparameter
```

**C++ Syntax**  
```
HRESULT IsSparameter( [out, retval] VARIANT_BOOL * bVal);
```

**Interface**  IMeasurement2
### Item Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Add or change a name-value pair in the Cal Set, or read the value associated with a name. After editing, <strong>Save</strong> the CalSet to the PNA.</th>
</tr>
</thead>
</table>

#### About Name-Value pairs

A Cal Set name-value pair is a general purpose data structure that maps a name to a value. This property allows you to associate a name with a value. Then, using this same property, you can read the value using the name.

For example, one of the items added by the PNA firmware to every Cal Set is named 'Created By'. The value attached to this item is the name of the PNA App that created the Cal Set. When an SMC cal is performed, you can query the Cal Set for the 'Create By' item, and it will return 'Scalar Mixer/Converter'. The same query on an NFx channel returns 'Noise Figure Converters'.

**Warning** - Do NOT change the name or value of any Items that you did NOT create. Otherwise, the PNA firmware may behave unpredictably.

#### See Also

- **EnumerateItems Method**
- **RemoveItem Method**

#### VB Syntax

```vbnet
CalSet.Item (name) = value
```

#### Variable (Type) - Description

- **CalSet** *(object)* - A **Cal Set** object
- **name** *(String)* - Name of the name-value pair.
- **value** *(Variant)* - Can be an integer, float, double, string, or a single-dimensioned array of integer, float, double, string.

#### Return Type

**Variant**

**Default**

Not Applicable

#### Example

```vbnet
' Create the pna object
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
' Get a handle to the calsets collection
Dim calsets
```
Set calsets = pna.GetCalManager.calsets
' Get a handle to the cal set to be edited
Dim MyCalSet
Set MyCalSet = calsets.Item("CalSet_1")
' Add a name-value pair(item) to MyCalSet
MyCalSet.Item("MyItem")=15
' Save the edited Cal Set to the PNA
MyCalSet.Save
' Loop thru the name-value pairs in the Cal Set
Dim CSetItems
CSetItems = MyCalSet.EnumerateItems
for i=lbound(CSetItems) to Ubound(CSetItems)
    ' List the item names in MyCalSet
    Dim name
    name = CSetItems(i)
    wscript.echo name
    ' List the value for each item name
    Dim value
    value = myCalSet.Item(name)
    wscript.echo value
Next
' Delete the new name-value pair
MyCalSet.RemoveItem("MyItem")

**C++ Syntax**

```cpp
HRESULT get_Item( BSTR name, VARIANT *value);
HRESULT put_Item( BSTR name, VARIANT value);
```

**Interface**

ICalSet6

**Last Modified:**

24-Sep-2010    MX New topic
## Items Property

**Description**
Returns a list of configured external devices in the collection.

**VB Syntax**
```
array = extDevices.Items
```

**Variable (Type) - Description**
- **value**
  - (array) Variable to store the returned values. Each item in the array is in the format (name:driver), where:
    - name = name of the external device
    - driver = [See a list of supported drivers](#).

**extDevices**
An [ExternalDevices](#) (collection)

**Return Type**
Variant Array

**Default**
Not Applicable

**Examples**
```
array = extDevices.Items 'Read
```

**C++ Syntax**
```
HRESULT get_Items( VARIANT* array);
```

**Interface**
IExternalDevices

---

Last Modified:

31-Jul-2009   MX New topic
Write/Read

IterationNumber Property

**Description**
Sets and returns the maximum leveling sweep iterations to be used in order to achieve the tolerance setting.

**VB Syntax**
RxLevel.IterationNumber(srcPort) = value

**Variable**
<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RxLevel</td>
<td>A ReceiverLeveling Object</td>
</tr>
<tr>
<td>srcPort</td>
<td>(Long Integer) Source port for which to set the IterationNumber for Receiver Leveling. <strong>Note:</strong> If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use chan.getPortNumber to translate the string into a port number. To learn more see <a href="#">Remotely Specifying a Source Port</a>.</td>
</tr>
<tr>
<td>value</td>
<td>(Long Integer) Max iterations. Choose a value between 1 and 25.</td>
</tr>
</tbody>
</table>

**Return Type**
(Long Integer)

**Default**
5

**Examples**
rxLevel.IterationNumber (1) = 10 ' Write
value = rxLevel.IterationNumber 2' Read

**C++ Syntax**
HRESULT get_IterationNumber(long port, long* pVal);
HRESULT put_IterationNumber(long port, long newVal);

**Interface**
IReceiverLevelingConfiguration

Last Modified:
13-Feb-2009  MX New topic
**IterationsTolerance Property**

**Description**
This command, along with `MaximumIterationsPerPoint Property` deal with adjustments made to the source power.

Sets the maximum desired deviation from the sum of the test port power and the offset value. Power readings will continue to be made, and source power adjusted, until a reading is within this tolerance value or the max number of readings has been met. The last value to be read is the valid reading for that data point.

The following two commands allow for settling of power readings:

- `ReadingsPerPoint Property`
- `ReadingsTolerance Property`

**VB Syntax**
```vbnet
pwrCal.IterationsTolerance = value
```

**Variable (Type) - Description**

- `pwrCal` (object) - A `SourcePowerCalibrator` (object)
- `value` (Double) – Tolerance value in dBm. Choose any number between 0 and 5

**Return Type**
Double

**Default**
.05 dB

**Examples**
```
Set powerCalibrator = pna.SourcePowerCalibrator
powerCalibrator.IterationsTolerance = .1 'Write
ReadTol = powerCalibrator.IterationsTolerance 'Read
```

**C++ Syntax**
```cpp
HRESULT get_IterationsTolerance( double *pVal);
HRESULT put_IterationsTolerance( double newVal);
```

**Interface**
ISourcePowerCalibrator3

---

**Last Modified:**

17-Apr-2007 Clarified verbage
**KaiserBeta Property**

**Description**  
Sets or returns the Kaiser Beta of Time Domain transform windows

**VB Syntax**  
`trans.KaiserBeta = value`

**Variable** (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>trans</code></td>
<td>A Transform (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(single) - Kaiser Beta. Choose any number between 0 and 13.</td>
</tr>
</tbody>
</table>

**Return Type**  
Single

**Default**  
0

**Examples**

```vbnet
trans.KaiserBeta = 6 'sets the Kaiser Beta of a transform window
KB = trans.KaiserBeta 'Read
```

**C++ Syntax**  
HRESULT get_KaiserBeta(float *pVal)  
HRESULT put_KaiserBeta(float newVal)

**Interface**  
ITransform
### L0 Property

**Description** Sets and Returns the L0 (L-zero) value (the first inductance value) for the calibration standard.

To set the other inductance values, use L1, L2, L3.

**VB Syntax**  
```vbnet
 calstd.L0 = value
```

**Variable** *(Type)* - Description

- `calstd` A CalStandard *(object)*. Use calKit.GetCalStandard to get a handle to the standard.
- `value` *(single)* - Value for L0 in femtohenries(1E-15)

**Return Type** Single

**Default** Not Applicable

**Examples**

```vbnet
  calstd.L0 = 15 'Write the value of L0 = 15 femtohenries
  Induct0 = calstd.L0 'Read the value of L0
```

**C++ Syntax**

```c++
HRESULT get_L0(float *pVal)
HRESULT put_L0(float newVal)
```

**Interface** ICalStandard
# L1 Property

**Description**
Sets and Returns the L1 value (the second inductance value) for the calibration standard.
To set the other inductance values, use L0, L2, L3.

**VB Syntax**
```
calstd.L1 = value
```

**Variable**

- `calstd` (object) - A CalStandard. Use calKit.GetCalStandard to get a handle to the standard.
- `value` (single) - Value for L1.

**Return Type**
Single

**Default**
Not Applicable

**Examples**
```
calstd.L1 = 15 'Write the value of L1
Induct1 = calstd.L1 'Read the value of L1
```

**C++ Syntax**
```
HRESULT get_L1(float *pVal)
HRESULT put_L1(float newVal)
```

**Interface**
ICalStandard
L2 Property

Description
Sets and Returns the L2 value (the third inductance value) for the calibration standard. To set the other inductance values, use L0, L1, L3.

VB Syntax
`calstd.L2 = value`

Variable (Type) - Description
- `value` (single) - Value for L2.

Return Type
Single

Default
Not Applicable

Examples
- `calstd.L2 = 15 'Write the value of L2.`
- `Induct2 = calstd.L2 'Read the value of L2`

C++ Syntax
- `HRESULT get_L2(float *pVal)`
- `HRESULT put_L2(float newVal)`

Interface
ICalStandard
**L3 Property**

**Description**  
Sets and Returns the L3 value (the third inductance value) for the calibration standard.  
To set the other inductance values, use L0, L1, L2.

**VB Syntax**  
`calstd.L3 = value`

**Variable**  
(**Type**) Description  

`value` (single) - Value for L3.

**Return Type**  
Single

**Default**  
Not Applicable

**Examples**  
`calstd.L3 = 15 'Write the value of L3.'`

`Induct3 = calstd.L3 'Read the value of L3`  

**C++ Syntax**  
HRESULT get_L3(float *pVal)
HR EULT put_L3(float newVal)

**Interface**  
ICalStandard
Label Property

**Description**
Sets and Returns the label for the calibration standard. The label is used to prompt the user to connect the specified standard.

**VB Syntax**
```
calstd.Label = value
```

**Variable**
- **calstd** A CalStandard *(object)*. Use calKit.GetCalStandard to get a handle to the standard.
- **value** *(string)* - between 1 and 12 characters long. Cannot begin with a numeric.

**Return Type**
String

**Default**
Not Applicable

**Examples**
```
calstd.Label = "Short" 'Write
stdLabel = calstd.Label 'Read
```

**C++ Syntax**
```
HRESULT get_Label(BSTR *pVal)
HRESULT put_Label(BSTR newVal)
```

**Interface**
ICalStandard
Label Property

**Description**
Sets or gets the display label for a given channel/testset combination. The label appears in a status bar at the bottom of the PNA display when the `ShowProperties` property is set to TRUE.

**VB Syntax**
`tset.Label(chNum) = value`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>tset</code></td>
<td>A <code>TestsetControl</code> object. Obtained from the <code>ExternalTestsets</code> collection.</td>
</tr>
<tr>
<td><code>chNum</code></td>
<td>(Integer) Channel number of the measurement.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(String) The text of the label.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
None

**Examples**
'The following sets the label for channel 5 corresponding to a given testset object.'
```
testset1.label(5) = 'High-power output'
```

**C++ Syntax**
```cpp
HRESULT get_Label(long channelNum, BSTR *pLabel);
HRESULT put_Label(long channelNum, BSTR label);
```

**Interface**
`ITestsetControl`
LANConfiguration Property

Description
Returns information about the current status of the PNA’s computer networking configuration. This is the same set of information that is returned in an NA_IPConfiguration data structure by the GetIPConfigurationStruct method.

VB Syntax
value = app.LANConfiguration

Variable (Type) - Description
value (String) Variable to contain the PNA’s LAN configuration string
app An Application (object)

Return Type Comma-delimited string

Default Not Applicable

Examples
networkConfigInfo = app.LANConfiguration

C++ Syntax
HRESULT get_LANConfiguration (BSTR * pStrConfig);

Interface IApplication13

Last Modified:
2-Jun-2008 MX New topic
### LastCalPassedTolerance Property

**Description**  
Returns the pass / fail status of the tolerance limits of the target power from the most recent source power cal.

**VB Syntax**  
\[ \text{value} = \text{pwrCal}.\text{LastCalPassedTolerance} \]

**Variable**  
- `pwrCal` (object) – A [SourcePowerCalibrator](#) (object)
- `value` (boolean) -  
  - **False** – Source power cal did NOT achieve the specified tolerance limits.
  - **True** – Source power cal DID achieve the specified tolerance limits.

**Return Type**  
Boolean

**Default**  
Not Applicable

**Examples**  
\[ \text{status} = \text{powerCal}.\text{LastCalPassedTolerance} \]

**C++ Syntax**  
```
HRESULT get_LastCalPassedTolerance(VARIANT_BOOL *bState);
```

**Interface**  
ISourcePowerCalibrator7

---

Last Modified:  
17-Mar-2010   MX New topic
Write/Read

About Receiver Leveling

LastLevelingAsSPC Property

Description
Sets and returns the state of **Use Last Result for Source Power Cal**. When Leveling Mode is switched back to Internal, this feature turns Source Power Cal correction ON using the latest receiver leveling correction data.

VB Syntax

\[ RxLevel.LastLevelingAsSPC(srcPort) = value \]

Variable (Type) - Description

**RxLevel**
A **ReceiverLeveling** Object

**srcPort**
(Long Integer) Source port being used for Receiver Leveling.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use **chan.getPortNumber** to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#)

**value**
(Boolean). Choose from:

**True** - When Leveling Mode is switched back to Internal, Source Power Cal correction is turned ON using the latest receiver leveling correction data.

**False** - When Leveling Mode is switched back to Internal, Source Power Cal correction is NOT turned ON.

Return Type
Variant Boolean

Default
False

Examples

\[ rxLevel.LastLevelingAsSPC(1) = True ' Write \]

\[ value = rxLevel.LastLevelingAsSPC 2' Read \]

C++ Syntax

```cpp
HRESULT get_LastLevelingAsSPC(long port, VARIANT_BOOL* pVal);
HRESULT put_LastLevelingAsSPC(long port, VARIANT_BOOL newVal);
```

Interface
IRxReceiverLevelingConfiguration2

Last Modified:
16-Sep-2010

MX New topic
About Cal Sets

LastModified Property

**Description**
Returns the time stamp of the last modification to this Cal Set.

This property always returns a time stamp based on the Greenwich Mean Time (GMT) regardless of the local time zone setting of the PNA.

The Cal Set properties that are viewed on the PNA user interface are converted to the local time of the PNA.

**VB Syntax**
```vbnet
value = Object.LastModified
```

**Variable (Type) - Description**
- **object** Channel (object)
  - or
    - CalSet (object) - Read-only property
- **value** (Variant) – Variable to store the time stamp.

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
```vbnet
date = CalSet.LastModified 'Read
```

**C++ Syntax**
```cpp
HRESULT get_LastModified(VARIANT* datetime)
```

**Interface**
ICalSet3

Last Modified:

25-Apr-2007 Added GMT note
About Receiver Leveling

LevelingIFBW Property

**Description**
Sets and returns the IFBW to be used for leveling sweeps. Enable separate IFBW for leveling sweeps using **FastMode Property**.

**VB Syntax**
```vbnet
RxLevel.LevelingIFBW(srcPort) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RxLevel</td>
<td>A <strong>ReceiverLeveling</strong> Object</td>
</tr>
<tr>
<td><strong>srcPort</strong></td>
<td>(Long Integer) Source port for which to set the LevelingIFBW for Receiver Leveling.</td>
</tr>
</tbody>
</table>

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see **Remotely Specifying a Source Port**

| **value**    | (Double) IFBW for leveling sweeps in Hz. The list of valid IF Bandwidths is different depending on the PNA model. See list. If an invalid number is specified, the PNA will round up to the closest valid number. |

**Return Type**
(Double)

**Default**
100 kHz

**Examples**
```vbnet
rxLevel.LevelingIFBW (1) = 1e3 ' Write
value = rxLevel.LevelingIFBW 2' Read
```

**C++ Syntax**
```cpp
HRESULT get_LevelingIFBW(long port, double* pVal);
HRESULT put_LevelingIFBW(long port, double newVal);
```

**Interface**
IRxReceiverLevelingConfiguration

---

Last Modified:
13-Feb-2009   MX New topic
Limit Property

**Description**
Sets and returns the power limit for the specified port.

**VB Syntax**
\[ gpl.Limit \ (port) = value \]

**Variable**
- **(Type)** - Description
  - **gpl**: A **GlobalPowerLimit** (object)
  - **port**: (Long) Port number for which power limit value is to be set.
  - **value**: (Double) Power limit value. Choose a value between -27 dBm (approximately) and the max settable power for the PNA.

**Return Type**
Double

**Default**
100 dBm for all ports

**Examples**
\[ gpl.Limit(1) = 0 \ 'Write \]
\[ Limit = gpl.Limit(2) \ 'Read \]

**C++ Syntax**
HRESULT get_Limit(long port, double *pVal)
HRESULT put_Limit(long port, double newVal)

**Interface**
IGlobalPowerLimit

Last Modified:
10-Aug-2009   MX New topic
## LimitFrequency Property

**Description**
Enable or disable the use of the power meter min and max frequencies.

**VB Syntax**
```
pwrSensor.LimitFrequency = value
```

**Variable (Type) - Description**
- **pwrSensor**: A PowerSensorAsReceiver (Object)
- **value**: (Boolean) - State of min and max frequency use. Choose from:
  - `False`: Min and max frequencies disabled.
  - `True`: Min and max frequencies enabled.

**Return Type**
Boolean

**Default**
False

**Examples**
```
pwrSensor.LimitFrequency = False 'Write

limit = pwrSensor.LimitFrequency 'Read
```

**C++ Syntax**
```
HRESULT put_LimitFrequency(VARIANT_BOOL newVal);
HRESULT get_LimitFrequency(VARIANT_BOOL* pVal);
```

**Interface**
IPowerSensorAsReceiver

---

Last Modified:

25-Aug-9  MX New topic
**LimitTestFailed Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the results of limit testing for the measurement.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>testFailed = meas.LimitTestFailed</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>testFailed</code></td>
<td><em>(boolean)</em> Variable to store the returned value</td>
</tr>
<tr>
<td></td>
<td><strong>False</strong> - Limit Test Passed</td>
</tr>
<tr>
<td></td>
<td><strong>True</strong> - Limit Test Failed</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Boolean</td>
</tr>
<tr>
<td>Default</td>
<td>False returned if there is no testing in progress.</td>
</tr>
<tr>
<td>Examples</td>
<td><code>Dim testRes As Boolean</code></td>
</tr>
<tr>
<td></td>
<td><code>testRes = meas.LimitTestFailed</code></td>
</tr>
<tr>
<td></td>
<td><code>MsgBox (testRes)</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT get_LimitTestFailed(VARIANT_BOOL*</code></td>
</tr>
<tr>
<td></td>
<td><code>trueIfFailed)</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>
### EndStimulus Property

**Description**  
When constructing a limit line, specifies the stimulus value for the end of the segment.

**VB Syntax**  
`limtseg.EndStimulus = value`

**Variable**  
- **Type**: - Description
  - `limtseg`: A LimitSegment *(object)*
  - `value`: *(double)* - End Stimulus X-axis value. No units

**Return Type**  
Double

**Default**  
0

**Examples**

```
Set limtseg = meas.LimitTest(1)
limtseg.EndStimulus = 8e9  'Write

EndStim = limtseg.EndStimulus  'Read
```

**C++ Syntax**

```
HRESULT get_EndStimulus(double *pVal)
HRESULT put_EndStimulus(double newVal)
```

**Interface**  
ILimitSegment
**EndResponse Property**

**Description**
When constructing a limit line, specifies the amplitude value at the end of the limit segment.

**VB Syntax**
`limtseg.EndResponse = value`

**Variable**
- **Type**: Description
  - `limts`  A LimitSegment (**object**)
  - `value` (**double**) - Y-axis value of the End Response limit. No units

**Return Type**
Double

**Default**
0

**Examples**
```vbnet
Set limtseg = meas.LimitTest(1)
limtseg.EndResponse = 10 'Write
EndResp = limtseg.EndResponse 'Read
```

**C++ Syntax**
```cpp
HRESULT get_EndResponse(double *pVal)
HRESULT put_EndResponse(double newVal)
```

**Interface**
ILimitSegment
### Type (limit) Property

**Description**  
Specifies the Limit Line type.

**VB Syntax**  
`limt(index).Type = value`

**Variable**  
- **(Type)** - Description
  - **limt**  
    A LimitSegment (object)
  - **index**  
    (variant) - Limit line number in the LimitTest collection
  - **value**  
    (enum NALimitSegmentType) - Limit Line type. Choose from:
    - 0 - `naLimitSegmentType_OFF` - turns limit line OFF
    - 1 - `naLimitSegmentType_Maximum` - limit line fails with a data point ABOVE the line
    - 2 - `naLimitSegmentType_Minimum` - limit line fails with a data point BELOW the line

**Return Type**  
Long Integer

**Default**  
0 - OFF

**Examples**  
```vba
Set limts = meas.LimitTest
limts.Type = naLimitSegmentType_Maximum 'Write

limitType = limts.Type 'Read
```

**C++ Syntax**  
```cpp
HRESULT put_Type(tagNALimitSegmentType *pVal)
HRESULT get_Type(tagNALimitSegmentType newVal)
```

**Interface**  
ILimitSegment
# LineDisplay Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Turns the display of limit lines ON or OFF. To turn limit TESTING On and OFF, use State Property.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Note</td>
<td>Trace data must be ON to view limit lines</td>
</tr>
</tbody>
</table>

## VB Syntax

```vbnet
limitst.LineDisplay = state
```

## Variable

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>limitst</code></td>
<td>A LimitTest (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean)</td>
</tr>
<tr>
<td><code>False</code></td>
<td>- Turns the display of limit lines OFF</td>
</tr>
<tr>
<td><code>True</code></td>
<td>- Turns the display of limit lines ON</td>
</tr>
</tbody>
</table>

## Return Type

Long Integer

## Default

True

## Examples

```vbnet
Limtttest.LineDisplay = true 'Write
lineDsp = Limtttest.LineDisplay 'Read
```

## C++ Syntax

```cpp
HRESULT get_LineDisplay(VARIANT_BOOL *pVal)
HRESULT put_LineDisplay(VARIANT_BOOL newVal)
```

## Interface

ILimitTest
## LoadCharFromFile Property

**Description**
Sets and returns whether a Mixer characterization file is to be loaded. Specify and load the filename with CharFileName Property.

**VB Syntax**
```vbnet
VMC.LoadCharFromFile = bool
```

**Variable (Type) - Description**
- **VMC VMCType** (object)
- **bool** (Boolean)
  - **True** - Load from file
  - **False** - Perform mixer characterization

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
value = VMC.LoadCharFromFile
```

**C++ Syntax**
```cpp
HRESULT put_LoadCharFromFile(VARIANT_BOOL bLoadCharFromfile);
HRESULT get_LoadCharFromFile(VARIANT_BOOL *bLoadCharFromfile);
```

**Interface**
VMCType
**LoadPort Property**

**Description**
Returns the load port number associated with an S-parameter reflection measurement. If the measurement is not a reflection S-parameter, the number returned by this property will have no meaning.

**VB Syntax**

```
loadPort = meas.LoadPort
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>loadPort</td>
<td>(long integer) - The reflection measurement's load port number.</td>
</tr>
<tr>
<td>meas</td>
<td>A Measurement (object)</td>
</tr>
</tbody>
</table>

**Return Type**

Long Integer

**Default**

Not Applicable

**Examples**

```
Set meas = pna.ActiveMeasurement
loadPort = meas.LoadPort
```

**C++ Syntax**

```
HRESULT get_LoadPort(long *pPortNumber);
```

**Interface**

IMeasurement
## LocalLockoutState Property

### Description
Prevents use of the mouse, keyboard, and front panel while your program is running. Use of these controls while this property is set TRUE causes an error message on the PNA display. To prevent these messages, see About Error Messages.

### VB Syntax
```vbnet
app.LocalLockoutState = bool
```

### Variable
**app** - An Application (object)

**bool** - Choose either:
- **False** - User Interface is NOT locked out.
- **True** - User Interface IS locked out.

### Return Type
Boolean

### Default
False

### Examples
```vbnet
app.LocalLockoutState = True 'Write
block = app.LocalLockoutState 'Read
```

### C++ Syntax
```cpp
HRESULT get_LocalLockoutState(VARIANT_BOOL *State);
HRESULT put_LocalLockoutState(VARIANT_BOOL *State);
```

### Interface
IApplication4
Locator Property

**Description**
Specifies the location, address, or ID string of the power meter / sensor that is used during a source power calibration. Use **Interface Property** to specify the type of interface.

**VB Syntax**
```
pwrMtrInterface.Locator = value
```

**Variable (Type) - Description**

- **pwrMtrInterface (object)** - A **PowerMeterInterface** (object)
- **value (string)** Location of the power meter / sensor, depending on the type of **Interface**.
  - For **naGPIB**, address of the power meter. Choose any integer between 0 and 30.
  - For **naUSB**, the ID string of the power sensor. Use **USBPowerMeterCatalog Property** to see a list of ID strings of connected power sensors.
  - For **naLAN**, the hostname or IP address of the power meter.

**Return Type**
String

**Default**
Not applicable

**Examples**
```
pwrMeterInterface.Locator = "13"  'GPIB address
pwrMeterInterface.Locator = "Agilent Technologies,U2000A,MY12345678"  'USB ID string
pwrMeterInterface.Locator = "mymeter.agilent.com"  'LAN
```

**C++ Syntax**
```
HRESULT put_Locator( BSTR pValue );
HRESULT get_Locator(BSTR* pValue );
```

**Locator**
IPowerMeterInterface

*Last Modified:*
- 29-Sep-2008   Removed Rev from Example
- 24-Jul-2007   MX New topic
## Lock Property

**Description**  
Enables or disables the ability to change the power limit values through the user interface.

**VB Syntax**  
`gpl.Lock = state`

**Variable**  
*(Type)* - Description

- `gpl`  
  A GlobalPowerLimit *(object)*

- `state`  
  *(boolean)*
  - **False** - Disables the ability to change the power limit values from the user interface.
  - **True** - Enables the ability to change the power limit values from the user interface.

**Return Type**  
Boolean

**Default**  
False

**Examples**  
`gpl.Lock = True 'Write`

`UILock = gpl.Lock 'Read`

**C++ Syntax**  
HRESULT get_Lock(BOOL *pVal)
HRESULT put_Lock(BOOL newVal)

**Interface**  
IGlobalPowerLimit

---

**Last Modified:**

5-Aug-2009  
MX New topic
LODeltaFound Property

**Description**
Returns the LO frequency delta from the specified tuning sweep.

**VB Syntax**
```
value = embedLODiag.LODeltaFound (n)
```

**Variable**

- **(Type)**: Description
  - **value**: (Double) Variable to store the returned data.
  - **embedLODiag**: An **EmbeddedLODiagnostic** (object)
    - **n**: (Long) Tuning sweep number. Use **NumberOfSweeps** to find the number of sweeps taken.
  - **Default**: Not Applicable

**Examples**
```
data= embedLO.LODeltaFound 3  'read
```

**C++ Syntax**
```
HRESULT LODeltaFound(long sweep, double* deltaLO);
```

**Interface**
IEmbeddedLODiagnostic

---

Last Modified:

13-Apr-2007 MX New topic
## LODenominator Property

### Description
Sets or returns the denominator value of the LO Fractional Multiplier.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

### VB Syntax

```
obj.LODenominator (n) = value
```

### Variable

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>A Mixer Interface pointer to the Measurement (object) Or A Converter Object</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>(Long)</td>
<td>LO denominator value</td>
</tr>
<tr>
<td>n</td>
<td>(Long)</td>
<td>LO stage number</td>
</tr>
</tbody>
</table>

Choose from 1 or 2

### Return Type
Long

### Default
1

### Examples

```vbnet
Print mixer.LODenominator(1) 'prints the value of the first LODenominator
```

### C++ Syntax

```cpp
HRESULT get_LODenominator(long *pVal)
HRESULT put_LODenominator(long newVal)
```

### Interface

IMixer
IConverter

---

**Last Modified:**
- 2-Feb-2009 Added Converter
- 4-Mar-2008 Added note.
Write/Read About Mixer Configuration

LOFixedFrequency Property

**Description**
Sets or returns the LO frequency value.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**VB Syntax**

```
obj.LOFixedFrequency(n) = value
```

**Variable (Type) - Description**

- **obj**: A **Mixer Interface** pointer to the **Measurement** (object)  
  Or
  A **Converter Object**

- **value**: **(double)** - LO Frequency in Hertz.

- **n**: **(Long)** - LO stage number
  Choose from 1 or 2

**Return Type**
Double

**Default**
0 Hz

**Examples**

Print mixer.LOFixedFrequency(1) 'prints the value of the first LO fixed frequency

**C++ Syntax**

```c++
HRESULT get_LOFixedFrequency(double *pVal)
HRESULT put_LOFixedFrequency(double newVal)
```

**Interface**

- IMixer
- IConverter

---

Last Modified:

- 2-Feb-2009  Added Converter
- 4-Mar-2008  Added note.
Read/Write About Embedded LO

LOFrequencyDelta Property

**Description**
Sets and returns LO Frequency Delta. There is usually no need to set this value. Read this value to determine the difference between the LO Frequency that is stated in the Mixer dialog box and the last measured LO Frequency.

**VB Syntax**

```
obj.LOFrequencyDelta = value
```

**Variable**

*(Type) - Description*

- **obj**
  - An EmbeddedLO (object) or
  - A ConverterEmbeddedLO (object)

- **value**
  - (Double) LO Frequency delta in Hertz.

**Return Type**

(Double)

**Default**

Not Applicable

**Examples**

```vba
embedLO.LOFrequencyDelta = 0 'write

value = embedLO.LOFrequencyDelta 'read
```

**C++ Syntax**

```cpp
HRESULT get_LOFrequencyDelta(double* val);

HRESULT put_LOFrequencyDelta(double val);
```

**Interface**

IEmbededLO

---

Last Modified:

- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
LogMagnitudeOffset Property  Superseded

Description  Note: This property is replaced by DoReceiverPowerCal Method.

For Receiver Calibration - Sets or returns the value to offset the normalized unratioed power measurement data. The unratioed power measurement is effectively calibrated to the power level specified by the value of LogMagnitudeOffset as soon as the Normalization property is set to ON after calling the DataToDivisor method.

To offset the data trace magnitude a specified value, use MagnitudeOffset Property

VB Syntax  meas.LogMagnitudeOffset = value

Variable  (Type) - Description

meas  (object) - A Measurement object

value  (double) - Offset value in dBm.

Return Type  Double

Default  0

Examples  meas.LogMagnitudeOffset = -10 'Write (-10 dBm)

calpower = meas.LogMagnitudeOffset 'Read
meas.DataToDivisor 'Store meas data as measurement divisor
meas.Normalize = 1 'Measurement is now calibrated to –10 dBm

C++ Syntax  HRESULT put_LogMagnitudeOffset(double newVal);
HRESULT get_LogMagnitudeOffset(double *pVal);

Interface  IMeasurement
# LOName Property

Sets or returns the name of the PNA internal source or external source to use as the LO in an FCA measurement.

**VB Syntax**

```vbnet
obj.LOName (n) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
</table>
| obj  | A Mixer Interface pointer to the Measurement (object)  
 Or  
 A Converter Object  |

| n    | (Long) - LO stage number  
 Choose from 1 or 2 |

| value | (string) - LO Source name. Use SourcePort Property to return a list of valid source ports. An external source must be configured and selected to be valid. Learn more about external source configuration. |

**Note:** If the port is defined by a string name, such as an external source or one of the Source 2 outputs on the 2-port, 2-source PNA-x model, then you must use cap.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

**Return Type**

String

**Default**

"Not controlled"

**Examples**

```plaintext
mixer.LOName(1) = "MySource"
```

**C++ Syntax**

```cpp
HRESULT get_LOName(string *pVal)  
HRESULT put_LOName(string newVal)
```

**Interface**

MIXer  
IConverter

---

**Last Modified:**

- 2-Feb-2009  Added Converter
- 24-Apr-2008  Added string names note
- 23-Jul-2007  Updated for external source config.
**LONumerator Property**

**Description**
Sets or returns the numerator value of the LO Fractional Multiplier.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

**VB Syntax**
```
obj.LONumerator (n) = value
```

**Variable (Type) - Description**
- `obj` A [Mixer Interface](#) pointer to the [Measurement](#) (object)
- Or
- A [Converter Object](#)
- `value` (Long) - LO denominator value
- `n` (Long) - LO stage number

Choose from 1 or 2

**Return Type**
Long

**Default**
1

**Examples**
```vbnet
Print mixer.LONumerator(1) 'prints the value of the first LO Numerator
```

**C++ Syntax**
```
HRESULT get_LONumerator(long *pVal)
HRESULT put_LONumerator(long newVal)
```

**Interface**
IMixer
IConverter

---

Last Modified:
- 2-Feb-2009  Added Converter
- 4-Mar-2008  Added note.
LOPower Property

**Description**
Sets or returns the value of LO Power.

**VB Syntax**
```
obj.LOPower (n) = value
```

**Variable (Type) - Description**
- **obj** A [Mixer Interface](#) pointer to the [Measurement](#) (object)
  - Or
  - A [Converter Object](#)
- **n** (Long) - LO stage number
  - Choose from 1 or 2
- **value** (double) - LO Power in dBm.

**Return Type**
Double

**Default**
-10dBm

**Examples**
```
Print mixer.LOPower(1) 'prints the value of the LO Power
```

**C++ Syntax**
```
HRESULT get_LOPower(double *pVal)
HRESULT put_LOPower(double newVal)
```

**Interface**
- IMixer
- IConverter

---

Last Modified:

2-Feb-2009 added converter
Write/Read

About Mixer Configuration

LORangeMode Property

Description
Sets or returns the LO sweep mode.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

Note: There is also a LORangeMode on the Converter Object.

VB Syntax
obj.LORangeMode (n) = value

Variable (Type) - Description

obj A Mixer Interface pointer to the Measurement (object)

n (Long) - LO stage number.
Choose from 1 or 2

value (Enum as MixerRangeMode) - LO sweep mode. Choose from:
0 - mixSwept
1 - mixFixed

Return Type
Enum

Default
0 - mixSwept

Examples
mixer.LORangeMode(1)=mixSwept

C++ Syntax
HRESULT get_LORangeMode(long LO, enum *pVal)
HRESULT put_LORangeMode(long LO, enum newVal)

Interface
IMixer3

Last Modified:

4-Mar-2008   Added note.
### Loss Property

**Description**
Sets and Returns the insertion loss for the calibration standard.

**VB Syntax**
```
calstd.loss = value
```

**Variable** *(Type) - Description*
- `calstd` A CalStandard *(object)*. Use calKit.[GetCalStandard](#) to get a handle to the standard.
- `value` *(single)* - Insertion loss in Gohms / sec. (Giga Ohms per second of electrical delay)

**Return Type**
Single

**Default**
Not Applicable

**Examples**
- `calstd.loss = 3.5 'Write 3.5 Gohms of loss`
- `stdLoss = calstd.loss 'Read the value of Loss`

**C++ Syntax**
- `HRESULT get_Loss(float *pVal)`
- `HRESULT put_Loss(float newVal)`

**Interface**
ICalStandard
Loss (Power Segment) Property

**Description**
Sets or returns the loss value associated with a PowerLossSegment.

**VB Syntax**

`lossSeg.Loss = value`

**Variable (Type) - Description**

- **lossSeg**
  - A `PowerLossSegment` (object)
  - A `PowerLossSegmentPMAR` (object)

- **value**
  - (double) – Loss value in dB. Loss is entered as a POSITIVE number.

**Return Type**
Double

**Default**
0

**Examples**

```
lossSeg.Loss = 0.5 'Write
lossVal = lossSeg.Loss 'Read
```

**C++ Syntax**

```
HRESULT put_Loss(Double newVal);
HRESULT get_Loss(Double *pVal);
```

**Interface**
- `IPowerLossSegment`
- `IPowerLossSegmentPMAR`

---

**Last Modified:**

- 27-Aug-2009 Added PMAR (9.0)
- 2-Jun-2008 Clarified negative value for loss
LOStage Property

**Description**
Sets or returns the number of LO stages present in the mixer.

**VB Syntax**

```vbnet
obj.LOStage = value
```

**Variable (Type) - Description**

- `obj`: A [Mixer Interface](#) pointer to the Measurement (object)
  - Or
  - A [Converter Object](#)

- `value`: (Long) - Number of LO stages. Choose from 1 or 2

**Return Type**
Long

**Default**
1

**Examples**

```vbnet
mixer.LOStage = 1 'sets the LO Stage value to 1
```

**C++ Syntax**

```cpp
HRESULT get_LOStage(long *pVal)
HRESULT put_LOStage(long newVal)
```

**Interface**

- IMixer
- IConverter

---

**Last Modified:**

2-Feb-2009   Added converter
Write/Read

About Mixer Configuration

LOStartFrequency Property

Description
Sets or returns the LO start frequency value. This command can only be used with SMC (not VMC) measurements.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

\[ \text{obj.LOStartFrequency}(n) = \text{value} \]

Variable (Type) - Description

<table>
<thead>
<tr>
<th>obj</th>
<th>A Mixer Interface pointer to the Measurement (object) Or A Converter Object</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(double) - LO Start Frequency in Hertz.</td>
</tr>
<tr>
<td>n</td>
<td>(Long) - LO stage number Choose from 1 or 2</td>
</tr>
</tbody>
</table>

Return Type
Double

Default
Not Applicable

Examples

Print mixer.LOStartFrequency(1) 'prints the value of the first LO start frequency

C++ Syntax

HRESULT get_LOStartFrequency(long id, double *pVal)
HRESULT put_LOStartFrequency(long id, double newVal)

Interface
IMixer3
IConverter

Last Modified:

2-Feb-2009      Added Converter
4-Mar-2008      Added note.
## LOStartPower Property

Sets or returns the Start value of a LO Power sweep. Also set `imdx.SweepType` to `nmDLOPowerSweep`.

### Description
Sets or returns the Start value of a LO Power sweep. Also set `imdx.SweepType` to `nmDLOPowerSweep`.

### VB Syntax
```vb
obj.LOStartPower (n) = value
```

### Variable (Type) - Description

- `obj` - A [Converter Object](#)
- `n` - LO stage number. Choose 1
- `value` - LO start power in dBm.

### Return Type
Double

### Default
-20 dBm

### Examples
```vb
convtr.LOStartPower(1) = -5 'Sets the LO Power sweep start value
start = convtr.LOStartPower(1) 'Reads the start value
```

### C++ Syntax
```cpp
HRESULT get_LOStartPower(long LOStage, double *pVal)
HRESULT put_LOStartPower(long LOStage, double newVal)
```

### Interface
IConverter

---

**Last Modified:**

- 27-Mar-2009
- MX New topic
## LOStopFrequency Property

**Description** Sets or returns the LO stop frequency value. This command can only be used with SMC (not VMC) measurements.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**VB Syntax**  

```
obj.LOStopFrequency (n) = value
```

**Variable**  

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>obj</code></td>
<td>A <strong>Mixer Interface</strong> pointer to the <strong>Measurement</strong> (object) Or A <strong>Converter Object</strong></td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(double)</strong> - LO Stop Frequency in Hertz.</td>
</tr>
<tr>
<td><code>n</code></td>
<td><strong>(Long)</strong> - LO stage number Choose from 1 or 2</td>
</tr>
</tbody>
</table>

**Return Type**  

Double

**Default**  

Not Applicable

**Examples**  

```
Print mixer.LOStopFrequency(1) 'prints the value of the first LO stop frequency
```

**C++ Syntax**  

```
HRESULT get_LOStopFrequency(long id, double *pVal)
HRESULT put_LOStopFrequency(long id,double newVal)
```

**Interface**  

IMixer3  
IConverter

---

Last Modified:  
2-Feb-2009   Added Converter  
4-Mar-2008   Added note.
LOStopPower Property

Description
Sets or returns the Stop value of a LO Power sweep. Also set imdx.SweepType to naIMDLOPowerSweep.

VB Syntax
obj.LOStopPower (n) = value

Variable (Type) - Description

obj A Converter Object

n (Long) - LO stage number. Choose 1

value (double) - LO Stop power in dBm.

Return Type Double

Default -10 dBm

Examples
convtr.LOStopPower(1) = -5 'Sets the LO Power sweep Stop value
Stop = convtr.LOStopPower(1) 'Reads the Stop value

C++ Syntax
HRESULT get_LOStopPower(long LOStage, double *pVal)
HRESULT put_LOStopPower(long LOStage, double newVal)

Interface IConverter

Last Modified:
27-Mar-2009  MX New topic
**LXIDeviceIDState Property**

**Description**
Opens and closes the LAN Status dialog with the LAN Status Indicator showing IDENTIFY.

The PNA supports this capability to satisfy a requirement of the LAN eXtensions for Instrumentation (LXI) standard. Changing the value of this property is the same operation that occurs when clicking the Toggle LXI Identification button on the Welcome web page presented by the PNA web server.

**VB Syntax**
```
app.LXIDeviceIDState = bool
```

**Variable**
*app* (An Application object)
*bool* (boolean) Choose from:
- **True** - Displays the LAN Status dialog with the Status Indicator showing IDENTIFY.
- **False** -
  
  - If the dialog had been opened by this property, then the LAN Status dialog is closed.
  
  - If the dialog was opened manually, then it will stay open.

**Return Type**
Boolean

**Default**
False

**Examples**
```
app.LXIDeviceIDState = True 'Write
value = app.LXIDeviceIDState 'Read
```

**C++ Syntax**
```
HRESULT get_LXIDeviceIDState( VARIANT_BOOL* pState);
HRESULT put_LXIDeviceIDState( VARIANT_BOOL  state);
```

**Interface**
IAplication14

---

Last Modified:
2-Jun-2008  MX New topic
**MagnitudeOffset Property**

**Description**
Offsets the data trace magnitude by the specified value.

To offset the data trace magnitude to a slope value that changes with frequency, use **MagnitudeSlopeOffset Property**.

To implement a Receiver Cal offset, use **LogMagnitudeOffset property**.

**VB Syntax**
```
meas.MagnitudeOffset = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>meas (object)</td>
<td>A Measurement object</td>
</tr>
<tr>
<td>value (double)</td>
<td>Offset value in dB.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**
```
meas.MagnitudeOffset = 4 'Write
offs = meas.MagnitudeOffset 'Read
```

**C++ Syntax**
```
HRESULT put_MagnitudeOffset(double newVal);
HRESULT get_MagnitudeOffset(double *pVal);
```

**Interface**
IMeasurement4
**MagnitudeSlopeOffset Property**

**Description**
Offsets the data trace magnitude to a value that changes linearly with frequency. The offset slope begins at 0 Hz.
To offset the entire data trace magnitude by a specified value, use `MagnitudeOffset Property`.

**VB Syntax**
```vbnet
meas.MagnitudeSlopeOffset = value
```

**Variable**
- **meas** *(object)* - A `Measurement` object
- **value** *(double)* - Offset slope value in dB / 1GHz.

**Return Type**
Double

**Default**
0

**Examples**
```vbnet
meas.MagnitudeSlopeOffset = 4 'Writes a slope offset of 4dB/1GHz.
offs = meas.MagnitudeSlopeOffset 'Read
```

**C++ Syntax**
```cpp
HRESULT put_MagnitudeSlopeOffset(double newVal);
HRESULT get_MagnitudeSlopeOffset(double *pVal);
```

**Interface**
`IMeasurement4`
# MainToneIFBandwidth Property

**Description**
Sets and returns the IF Bandwidth for measurement of the Main tones.

**VB Syntax**
```vbnet
imd.MainToneIFBandwidth = value
```

**Variable (Type) - Description**
- `imd` A SweptIMD Object
- `value` (Double) IF Bandwidth in Hz. Choose from: 1 | 2 | 3 | 5 | 7 | 10 | 15 | 20 | 30 | 50 | 70 | 100 | 150 | 200 | 300 | 500 | 700 | 1k | 1.5k | 2k | 3k | 5k | 7k | 10k | 15k | 20k | 30k | 50k | 70k | 100k | 150k | 200k | 280k | 360k | 600k

- **Learn more about setting IFBW for IMD.**
- If an invalid number is specified, the analyzer will round up to the closest valid number.

**Return Type**
Double

**Default**
1 kHz

**Examples**
```vbnet
imd.MainToneIFBandwidth = 2e3 'Write

value = imd.MainToneIFBandwidth 'Read
```

**C++ Syntax**
```cpp
HRESULT get_MainToneIFBandwidth(double *pVal)
HRESULT put_MainToneIFBandwidth(double newVal)
```

**Interface**
ISweptIMD

---

Last Modified: 19-Aug-2008   MX New topic
## MarkerAnnotation Property

**Description**
Returns the Y-axis marker value from the specified tuning sweep. Use `IsMarkerOn` to confirm if a marker was used for the tuning sweep.

**VB Syntax**
```vbnet
value = embedLODiag.MarkerAnnotation(n)
```

**Variable (Type) - Description**

- **value** (String) Variable to store the returned data.
- **embedLODiag** An `EmbeddedLODiagnostic` (object)
- **n** (Long) Tuning sweep number. Use `NumberOfSweeps` to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**
```vbnet
data = embedLO.MarkerAnnotation 3 'read
```

**C++ Syntax**
```cpp
HRESULT MarkerAnnotation(long sweep, BSTR* annotation);
```

**Interface**
IEmbededLODiagnostic

---

Last Modified:

- 13-Apr-2007  MX New topic
### MarkerFormat Property

**Description**
Sets the format of all the markers in the measurement. To override this setting for an individual marker, use `mark.Format`.

**VB Syntax**
```
meas.MarkerFormat = value
```

**Variable (Type) - Description**
- `meas` A Measurement (object)
- `value` (enum NAMarkerFormat) - Choose from:
  - 0 - naMarkerFormat_LinMag
  - 1 - naMarkerFormat_LogMag
  - 2 - naMarkerFormat_Phase
  - 3 - naMarkerFormat_Delay
  - 4 - naMarkerFormat_Real
  - 5 - naMarkerFormat_Imaginary
  - 6 - naMarkerFormat_SWR
  - 7 - naMarkerFormat_LinMagPhase
  - 8 - naMarkerFormat_LogMagPhase
  - 9 - naMarkerFormat_ReallImaginary
  - 10 - naMarkerFormat_ComplexImpedance
  - 11 - naMarkerFormat_ComplexAdmittance
  - 12 - naMarkerFormat_Kelvin
  - 13 - naMarkerFormat_Fahrenheit
  - 14 - naMarkerFormat_Centigrade

**Return Type**
Not Applicable

**Default**
1 - naMarkerFormat_LogMag

**Examples**
```
meas.MarkerFormat = naMarkerFormat_SWR  'Write
```

**C++ Syntax**
```
HRESULT put_MarkerFormat(tagNAMarkerFormat NewFormat)
```

**Interface**
IMeasurement

---

Last Modified:

1182
17-Nov-2010  Removed Read capability
1-Oct-2007  Added temperature formats
Write-only

InterpolateMarkers Method

Description
Turns All Marker Interpolation ON and OFF for the measurement. Marker interpolation enables X-axis resolution between the discrete data values. The analyzer will calculate the x and y-axis data values between discrete data points. To override this property for individual markers, use the Interpolated property.

VB Syntax
`meas.Interpolate = value`

Variable
(Type) - Description

`meas` A Measurement (object)

`value` (boolean)
False - Turns interpolation OFF for all markers in the measurement
True - Turns interpolation ON for all markers in the measurement

Return Type
Boolean

Default
True (ON)

Examples
`meas.Interpolate = 1`

C++ Syntax
`HRESULT InterpolateMarkers(VARIANT_BOOL bNewVal)`

Interface
IMeasurement
# Number Property

**Description**
Returns the number of the marker.

**VB Syntax**
```vbnet
marknum = mark.Number
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>marknum</td>
<td>(long)</td>
<td>Variable to store marker number</td>
</tr>
<tr>
<td>mark</td>
<td>A Marker (object)</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
Not applicable

**Examples**
```vbnet
marknum = mark.Number 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Number(long *pVal)
```

**Interface**
IMarker
Read-only

MarkerPosition Property

**Description**
Returns the X-axis marker position from the specified tuning sweep.

**VB Syntax**
```
value = embedLODiag.MarkerPosition(n)
```

**Variable**
*Type* - Description

- **value** (Double) Variable to store the returned data.

- **embedLODiag** An [EmbeddedLODiagnostic](#) (object)

- **n** (Long) Tuning sweep number. Use [NumberOfSweeps](#) to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**
```
data = embedLO.MarkerPosition 3 'read
```

**C++ Syntax**
```
HRESULT MarkerPosition(long sweep, double *position);
```

**Interface**
IE EmbeddedLODiagnostic

---

Last Modified:

13-Apr-2007    MX New topic
**MarkerReadout Property**

**Description**
Enables or disables the readout of markers in the window. To show the marker on the screen use ShowMarkerReadout Method.

**VB Syntax**

```
win.MarkerReadout = state
```

**Variable**

*(Type) - Description*

*win*  
A NAWindow (object)

*state*  
(boolean)

True - enables marker readout  
False - disables marker readout

**Return Type**
Boolean

**Default**
True

**Examples**

```
win.MarkerReadout = True 'Write
State = app.ActiveNAWindow.MarkerReadout 'Read
```

**C++ Syntax**

```
HRESULT get_MarkerReadout(VARIANT_BOOL *pVal)
HRESULT put_MarkerReadout(VARIANT_BOOL newVal)
```

**Interface**
INAWindow
## MarkerReadoutResponsePlaces Property

**Description**  
For the Y-axis (response), sets the number digits to display after the decimal point in marker readouts.

**VB Syntax**  
`win.MarkerReadoutResponsePlaces = value`

**Variable**  
*(Type)* - Description  

<table>
<thead>
<tr>
<th>win</th>
<th>A <code>NAWindow</code> (object)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th><em>(Long Integer)</em> Number of digits to display. Choose a value between 1 and 4.</th>
</tr>
</thead>
</table>

**Return Type**  
Long Integer

**Default**  
2

**Examples**  
`win.MarkerReadoutResponsePlaces = 3 'Write`

```
value = app.ActiveNAWindow.MarkerReadoutResponsePlaces 'Read
```

**C++ Syntax**  
```c++
HRESULT get_MarkerReadoutResponsePlaces(long *pVal)
HRESULT put_MarkerReadoutResponsePlaces(long newVal)
```

**Interface**  
`INAWindow3`

---

**Last Modified:**  
5-Aug-2010    MX New topic
MarkerReadoutSize Property

Description: Specifies the size of font used when displaying Marker Readout in the selected window.

VB Syntax: `win.MarkerReadoutSize = value`

Variable (Type) - Description:
- `win`: A NAWindow (object)
- `value`: (enum NAFontSize)
  - `0 - naDefault`: marker readout appears in default font size
  - `1 - naLarge`: marker readout appears in large font size

Return Type: Long Integer

Default: `naDefault`

Examples:
- `win.MarkerReadoutSize = naDefault` 'write default size for marker readout'
- `Dim Size As NAFontSize
  Size = app.ActiveNAWindow.MarkerReadoutSize` 'Read'

C++ Syntax:
- `HRESULT get_MarkerReadoutSize(tagNAFontSize *pVal)`
- `HRESULT put_MarkerReadoutSize(tagNAFontSize newVal)`

Interface: INAWindow
MarkerReadoutsPerTrace Property

**Description**
Sets the number of marker readouts to display per trace. Display up to 20 marker readouts per window.

**VB Syntax**

```vbnet
win.MarkerReadoutsPerTrace = value
```

**Variable**
**(Type) - Description**

- **win**: A `NAWindow` (object)
- **value**: (Long Integer) Number of marker readouts to display. Choose a value between 1 and 10.

**Return Type**
Long Integer

**Default**
5

**Examples**

```vbnet
win.MarkerReadoutsPerTrace = 3 'Write
value = app.ActiveNAWindow.MarkerReadoutsPerTrace 'Read
```

**C++ Syntax**

```cpp
HRESULT get_MarkerReadoutsPerTrace(long *pVal)
HRESULT put_MarkerReadoutsPerTrace(long newVal)
```

**Interface**
INAWindow3

---

Last Modified:

5-Aug-2010    MX New topic
## MarkerReadoutStimulusPlaces Property

**Description**
For the X-axis (stimulus), sets the number digits to display after the decimal point in marker readouts.

**VB Syntax**
```
win.MarkerReadoutStimulusPlaces = value
```

**Variable (Type) - Description**
- `win` A `NAWindow` (object)
- `value` (Long Integer) Number of digits to display. Choose a value between 2 and 6.

**Return Type**
Long Integer

**Default**
3

**Examples**
```
win.MarkerReadoutStimulusPlaces = 2 'Write
value = app.ActiveNAWindow.MarkerReadoutStimulusPlaces 'Read
```

**C++ Syntax**
- `HRESULT get_MarkerReadoutStimulusPlaces(long *pVal)`
- `HRESULT put_MarkerReadoutStimulusPlaces(long newVal)`

**Interface**
INAWindow3

---

Last Modified:
5-Aug-2010  MX New topic
Write/Read

MarkerReadoutXPosition Property

**Description**: Sets the X-axis position of marker readouts. Readouts are right-justified at the specified position.

**VB Syntax**: `win.MarkerReadoutXPosition = value`

**Variable (Type) - Description**

- `win`: A `NAWindow` (object)
- `value`: (Double) X-axis position. Choose a value between 1 (far left) and 10 (far right).

**Return Type**: Double

**Default**: 10

**Examples**: `win.MarkerReadoutXPosition = 3 'Write

value = app.ActiveNAWindow.MarkerReadoutXPosition 'Read`

**C++ Syntax**: HRESULT get_MarkerReadoutXPosition(double *pVal)

HRESULT put_MarkerReadoutXPosition(double newVal)

**Interface**: INAWindow3

---

Last Modified:

5-Aug-2010  MX New topic
## MarkerReadoutYPosition Property

**Description**
Sets the Y-axis position of marker readouts. Readouts are right-justified at the specified position.

**VB Syntax**
\[ win.MarkerReadoutYPosition = value \]

**Variable**
- **(Type)**: Description
  - \( win \) A NAWindow (object)
  - \( value \) (Double) Y-axis position. Choose a value between 1 (bottom) and 10 (top).

**Return Type**
Double

**Default**
10

**Examples**
- \( win.MarkerReadoutYPosition = 3 \) 'Write
- \( value = app.ActiveNAWindow.MarkerReadoutYPosition \) 'Read

**C++ Syntax**
- HRESULT get_MarkerReadoutYPosition(double *pVal)
- HRESULT put_MarkerReadoutYPosition(double newVal)

**Interface**
INAWindow3

---

Last Modified:

5-Aug-2010   MX New topic
Markers Property

**Description**  Set and return the color of data trace markers for nth trace in a window.

**VB Syntax**  
```
trace(n).Markers = value
```

**Variable**  
- **Type** - Description
  - `trace(n)`  One of the 8 [ComTraceColors](#) objects
  - `value`  (Long Integer) - RGB color of the Markers pen.

Convert the three RGB colors to an integer as follows:

```
RGB = R + (G*2^8) + (B*2^16)
```

To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

**Return Type**  Long

**Default**  Varies for each trace.

**Examples**

```
R = 10
G = 10
B = 10
RGB = 10+(10*2^8)+(10*2^16)
trace1.Markers = RGB 'Write
color = trace1.Markers 'Read
```

**C++ Syntax**

```cpp
HRESULT get_Markers(long* pVal);
HRESULT put_Markers(long newVal);
```

**Interface**  IComTraceColors

---

Last Modified: 7-Aug-2009  MX New topic
## MarkerState Property

### Description
Sets or returns the ON / OFF state of the specified marker.

### VB Syntax
```
meas.MarkerState (n) = state
```

### Variable (Type) - Description
- **meas**: A Measurement (object)
- **n**: (Long Integer) Marker number to turn on or off.
- **state**: (boolean) -
  - True - turns the specified marker ON
  - False - turns the specified marker OFF

### Return Type
Boolean

### Default
False

### Examples
- `meas.MarkerState(1) = True`
- `reference = meas.MarkerState(2)`

### C++ Syntax
- `HRESULT get_MarkerState(long markerNum, VARIANT_BOOL bState)`
- `HRESULT put_MarkerState(long markerNum, VARIANT_BOOL* bState)`

### Interface
IMeasurement3
**MarkerSymbol Property**

**Description**
Sets the symbol to display for marker position.

**VB Syntax**

```vbnet
win.MarkerSymbol = value
```

**Variable (Type) - Description**

- **win**
  A **NAWindow** (object)

- **value**
  (Enum as **naMarkerSymbol**) Choose from:
  - 0 - **naMarkerSymbolTriangle**
  - 1 - **naMarkerSymbolFlag**
  - 2 - **naMarkerSymbolLine**

  [See pictures of symbols](#)

**Return Type**

Enum

**Default**

0 - **naMarkerSymbolTriangle**

**Examples**

```vbnet
win.MarkerSymbol = naMarkerSymbolLine
'Write

value = app.ActiveNAWindow.MarkerSymbol
'Read
```

**C++ Syntax**

```cpp
HRESULT get_MarkerSymbol(double *pVal)
HRESULT put_MarkerSymbol(double newVal)
```

**Interface**

INAWindow3

---

**Last Modified:**

5-Aug-2010  MX New topic
### Type (Marker) Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and reads the marker type.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.Type = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>chan</code></td>
<td>A Marker (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(enum NAMarkerType) - Marker Type. Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - naMarkerType_Normal - the X-axis value for a normal marker will always be determined by the measurement data of the marker.</td>
</tr>
<tr>
<td></td>
<td>1 - naMarkerType_Fixed - retains and keeps its x-axis value at the time the marker type is set.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Long Integer</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>naMarkerType_Normal</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.Type = naMarkerType_Normal 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>MrkType = mark.Type 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_Type(tagNAMarkerType *pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Type(tagNAMarkerType newVal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>
### Stimulus Property

**Description**
Sets and reads the X-Axis value of the marker. If the marker is a delta marker, the value will be relative to the reference marker.

**VB Syntax**
```vbnet
mark.Stimulus = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>mark</td>
<td>A Marker (object)</td>
</tr>
<tr>
<td>value</td>
<td>(double) - X-Axis value. Choose any number within the full span of the channel or User Range (if set).</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
First activated Marker turns ON in the middle of the X-axis range. Subsequent markers turn ON at the position of the most recently active marker.

**Examples**
```
mark.Stimulus = 3e9 'Write
XVal = mark.Stimulus 'Read
```

**C++ Syntax**
```c++
HRESULT get_Stimulus(double *pVal)
HRESULT put_Stimulus(double newVal)
```

**Interface**
IMarker
Description
Reads the Y-axis value of the marker. If the marker is a delta marker, the value will be relative to the reference marker.

You cannot set the Y-axis value of a marker. The marker remains at the position at the time you set `marker.Type`.

**Note:** To accurately read the marker Y-axis value with trace smoothing applied, the requested format must match the displayed format. Otherwise, the returned value is unsmoothed data. For example, to read the smoothed marker value when measuring group delay, both the display format and the marker format must be set to (Group) Delay.

VB Syntax

```
YValue = mark.Value (format)
```

Variable

<table>
<thead>
<tr>
<th><strong>Variable</strong></th>
<th><strong>(Type)</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>YValue</code></td>
<td><strong>A variable to store the Y-axis value</strong></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>mark</strong></th>
<th><strong>A Marker (object)</strong></th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th><strong>format</strong></th>
<th><strong>(enum NAMarkerFormat)</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>naMarkerFormat_LinMag (1)</td>
<td>The format in which to return the marker's Y-axis value. The number in parenthesis following the format is the number of values that are returned in a variant array. Choose from:</td>
</tr>
<tr>
<td>1</td>
<td>naMarkerFormat_LogMag (1)</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>naMarkerFormat_Phase (1)</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>naMarkerFormat_Delay (1)</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>naMarkerFormat_Real (1)</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>naMarkerFormat_Imaginary (1)</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>naMarkerFormat_SWR (1)</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>naMarkerFormat_LinMagPhase (2)</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>naMarkerFormat_LogMagPhase (2)</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>naMarkerFormat_ReallImaginary (2)</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>naMarkerFormat_ComplexImpedance (3)</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>naMarkerFormat_ComplexAdmittance (3)</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>naMarkerFormat_Kelvin (1)</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>naMarkerFormat_Fahrenheit (1)</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>naMarkerFormat_Celsius (1)</td>
<td></td>
</tr>
</tbody>
</table>

Return Type

`Variant` - The (parens) in the previous list of formats indicates the number of values that are returned in a variant array.
**Default**  
Not applicable

**Examples**  
`YVal = mark.Value(0)` *Read*

`or`

`YVal = mark.Value(naMarkerFormat_LinMag)`

**C++ Syntax**  
`HRESULT get_Value(tagNAMarkerFormat format, VARIANT *pVal)`

**Interface**  
IMarker

---

Last modified:

- 13-Apr-2011  Fixed example
- 1-Oct-2007    Added temperature formats
- 4-Dec-2006    Added smoothing note
Read/Write

**MatchCorrectPower Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Turns match-correction ON or OFF. Use this command AFTER performing a Guided Power Cal. <a href="#">Learn more.</a></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>corrMethods.MatchCorrectPower = value</code></td>
</tr>
</tbody>
</table>
| Variable    | *(Type)* - Description \[
| `corrMethods` | CorrectionMethods (object)                                                                    |
| `value`     | *(Boolean)* \[
| True        | Turns match-correction ON                                                                      |
| False       | Turns match-correction OFF                                                                     |
| Return Type | Boolean                                                                                         |
| Default     | True                                                                                           |
| Example     | `corrMethods.MatchCorrectPower = True`                                                          |
| C++ Syntax  | HRESULT get_MatchCorrectPower(VARIANT_BOOL* val);                                               |
|            | HRESULT put_MatchCorrectPower(VARIANT_BOOL newVal);                                             |
| Interface   | ICorrectionMethods                                                                              |

Last Modified:

10-Sep-2010  MX New topic
# MaximumFrequency Property

**Description**
Sets and Returns the maximum frequency for the calibration standard.

**VB Syntax**
`calstd.MaximumFrequency = value`

**Variable**
- **calstd** A CalStandard *(object)*. Use calKit.GetCalStandard to get a handle to the standard.
- **value** *(double)* - Maximum frequency in Hertz.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
- `calstd.MaximumFrequency = 9e9` *(Write)*
- `maxFrequency = calstd.MaximumFrequency` *(Read)*

**C++ Syntax**
- `HRESULT get_MaximumFrequency(double *pVal)`
- `HRESULT put_MaximumFrequency(double newVal)`

**Interface**
ICalStandard
MaximumFrequency Property

**Description**  
Returns the maximum frequency of the remote PNA.

**VB Syntax**  
\[ value = cap.MaximumFrequency \]

**Variable**  
(Type) - Description

\[ value \] (Double) - Variable to store the returned maximum frequency of the PNA.

\[ cap \] A Capabilities (object)

**Return Type**  
Double

**Default**  
Not Applicable

**Examples**  
\[ value = cap.MaximumFrequency \]

**C++ Syntax**  
HRESULT get_MaximumFrequency(&frequencyMax);

**Interface**  
ICapabilities
### MaximumFrequency Property

**Description**  
Set and returns the maximum usable frequency specified for the power sensor.

**VB Syntax**  
```vbnet
pwrSensor.MaximumFrequency = value
```

**Variable (Type) - Description**

- **pwrSensor**: A `PowerSensor` (Object) or a `PowerSensorAsReceiver` (Object)
- **value**: (double) - Frequency in Hertz.

**Return Type**  
Double

**Default**  
Device dependent

**Examples**

- `pwrSensor.MaximumFrequency = 6e9 'Write`
- `MaxFreq = pwrSensor.MaximumFrequency 'Read`

**C++ Syntax**

```cpp
HRESULT put_MaximumFrequency(double newVal);
HRESULT get_MaximumFrequency(double *pVal);
```

**Interface**

- `IPowerSensor`
- `IPowerSensorPMAR`
MaximumIFFilterSampleCount Property

**Description**
Returns the maximum allowed value for the `IFFilterSampleCount` property for the queried PNA.

**Note:** This setting applies only to the E836X Opt. H11.

**VB Syntax**
```
value = IfConfig.MaximumIFFilterSampleCount
```

**Variable**
- **(Type)** long
- **Description**
The maximum allowed value that can be applied to the `IFFilterSampleCount` property.

**IfConfig**
`IFConfiguration` (object)

**Return Type**
Long Integer

**Default**
PNA Model number dependent

**Examples**
```
variable = App.ActiveChannel.IFConfiguration.MaximumIFFilterSampleCount
```

See an example program

**C++ Syntax**
```
HRESULT get_MaximumIFFilterSampleCount( long * pMaxSampleCount );
```

**Interface**
`IIFConfiguration2`
## MaximumIFFrequency Property

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Returns the Maximum allowed value for the <strong>IFFrequency Property</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>value = IfConfig.MaximumIFFrequency</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(double)</em> Variable to store the returned maximum allowed frequency that can be applied to the <strong>IFFrequency Property</strong>.</td>
</tr>
<tr>
<td><code>IfConfig</code></td>
<td><strong>IFConfiguration</strong> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Double</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>val = App.ActiveChannel.IFConfiguration.MaximumIFFrequency</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_MaximumIFFrequency( double * pMaxFreq);</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><strong>IIFConfiguration3</strong></td>
</tr>
</tbody>
</table>

---

**Last Modified:**

18-Jun-2007   MX New topic
**MaximumIterationsPerPoint Property**

**Description**  This command, along with **IterationsTolerance Property** deal with adjustments made to the source power.

Sets the maximum number of readings to take at each data point for iterating the source power. Power readings will continue to made, and source power adjusted, until a reading is within the **IterationsTolerance** value or this max number of readings has been met. The last value to be read is the valid reading for that data point.

The following two commands allow for settling of power readings.

**ReadingsPerPoint Property**
**ReadingsTolerance Property**

**VB Syntax**

```
pwrCal.MaximumIterationsPerPoint = value
```

**Variable**  *(Type)* - Description

- **pwrCal** *(object)* - A **SourcePowerCalibrator** (object)
- **value** *(Long)* – Maximum number of readings. Choose any number between 1 and 100.

**Return Type**  Long Integer

**Default**  5

**Examples**

```
Set powerCalibrator = pna.SourcePowerCalibrator
powerCalibrator.MaximumIterationsPerPoint = 5 'Write
MaxReads = powerCalibrator.MaximumIterationsPerPoint 'Read
```

**C++ Syntax**

```
HRESULT get_MaximumIterationsPerPoint( long *pVal);
HRESULT put_MaximumIterationsPerPoint( long newVal);
```

**Interface**  ISourcePowerCalibrator3

---

Last Modified:

17-Apr-2007  Clarified verbage
### MaximumNumberOfChannels Property

**Description**
Returns the maximum possible number of channels that can be used in the PNA.

**VB Syntax**

```vbnet
value = cap.MaximumNumberOfChannels
```

**Variable**

- **Type**: (Long)
  - **Description**: Variable to store the returned maximum value for number of channels.

- **cap**: A Capabilities (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**

```vbnet
value = cap.MaximumNumberOfChannels 'Read
```

**C++ Syntax**

```c++
HRESULT get_MaximumNumberOfChannels(long * maximumNumberOfChans);
```

**Interface**
ICapabilities2
MaximumNumberOfTracesPerWindow Property

**Description**
Returns the maximum possible number of traces that can reside in any window.

**VB Syntax**

```vbnet
value = cap.MaximumNumberOfTracesPerWindow
```

**Variable (Type)** - Description

- **value** (Long) - Variable to store the returned maximum value for number of traces.
- **cap** A [Capabilities](object) (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**

```vbnet
value = cap.MaximumNumberOfTracesPerWindow 'Read
```

**C++ Syntax**

```cpp
HRESULT get_MaximumNumberOfTracesPerWindow(long * maximumNumberOfTraces);
```

**Interface**
ICapabilities2
MaximumNumberOfWindows Property

**Description**
Returns the maximum possible number of windows that can be present on the PNA screen.

**VB Syntax**

```vbnet
value = cap.MaximumNumberOfWindows
```

**Variable**

- **value** *(Long)* - Variable to store the returned maximum value for number of windows.
- **cap** A Capabilities *(object)*

**Return Type**
Long

**Default**
Not Applicable

**Examples**

```vbnet
value = cap.MaximumNumberOfWindows 'Read
```

**C++ Syntax**

```cpp
HRESULT get_MaximumNumberOfWindows(long * maximumNumberOfWindows);
```

**Interface**
ICapabilities2
MaximumNumberOfPoints Property

Description
Returns the maximum possible number of data points.

VB Syntax
\[ value = obj.MaximumNumberOfPoints \]

Variable (Type) - Description
\[ value \text{ (Long)} - \text{Variable to store the returned maximum value for number of points.} \]
\[ cap \text{ A Capabilities (object)} \]
\[ \text{or} \]
\[ \text{A GainCompression (object)} \]

Return Type
Long

Default
Not Applicable

Examples
\[ value = cap.MaximumNumberOfPoints \]

C++ Syntax
\[ \text{HRESULT get\_MaximumNumberOfPoints(long * maximumNumberOfPoints)}; \]

Interface
ICapabilities
IGainCompression

Last Modified:
3-Dec-2007 MX New topic
MaximumPowerCompression
MaximumReceiverStepAttenuator Property

**Description**
Returns the maximum amount of receiver attenuation.

**VB Syntax**
```
value = cap.MaximumReceiverStepAttenuator(n)
```

**Variable**

- **value** *(Double)* - Variable to store the returned value of maximum receiver attenuation.
- **cap** A **Capabilities** *(object)*
- **n** *(Long)* - port number to query for step attenuators

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```
value = cap.MaximumReceiverStepAttenuator'Read
```

**C++ Syntax**
```
HRESULT get_MaximumReceiverStepAttenuator(long portNumber, double * attenuation);
```

**Interface**
ICapabilities
MaximumSourceALCPower Property

**Description**
Returns a value indicating the maximum amount of source ALC power.

**VB Syntax**

```vbnet
value = cap.MaximumSourceALCPower (n)
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Double) - Variable to store the returned value for the maximum amount of source ALC power.</td>
</tr>
<tr>
<td>cap</td>
<td>ACapabilities (object)</td>
</tr>
<tr>
<td>n</td>
<td>(Long) - source number to query for maximum ALC power</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not Applicable

**Examples**

```vbnet
value = cap.MaximumSourceALCPower 'Read
```

**C++ Syntax**

```cpp
HRESULT get_MaximumSourceALCPower(long sourceNum, double * power );
```

**Interface**
ICapabilities
MaximumSourceStepAttenuator Property

Description
Returns a value for the maximum amount of source attenuation.

VB Syntax
value = cap.MaximumSourceStepAttenuator (n)

Variable (Type) - Description
value (Double) - Variable to store the returned value for the maximum amount of source attenuation.
cap A Capabilities (object)
n (Long) - port number to query for the maximum amount of source attenuation

Return Type Double
Default Not Applicable

Examples
value = cap.MaximumSourceStepAttenuator 2 'Read

C++ Syntax
HRESULT get_MaximumSourceStepAttenuator(long portNumber, double * attenuation );

Interface ICapabilities

Last Modified:
19-Sep-2007 Added port arg to ex.
# MaxPreciseTuningIterations Property

**Description**
Sets and returns the maximum number of tuning iterations to achieve the precise tolerance.

**VB Syntax**
```
obj.MaxPreciseTuningIterations = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>- Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>An EmbeddedLO (object) or A ConverterEmbeddedLO (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Long) Maximum number of tuning iterations.</td>
</tr>
</tbody>
</table>

**Return Type**

| (Long) |

**Default**
5

**Examples**

```
embedLO.MaxPreciseTuningIterations = 3 'write
```

```
value = embedLO.MaxPreciseTuningIterations 'read
```

**C++ Syntax**

```c++
HRESULT get_MaxPreciseTuningIterations(long* iter);
HRESULT put_MaxPreciseTuningIterations(long iter);
```

**Interface**
IEmbeddedLO

---

Last Modified:

- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
### MaxProduct Property

**Description**  
Sets and returns the maximum intermod product frequencies to be calibrated.

**VB Syntax**  
`imd.MaxProduct = value`

**Variable**  
<table>
<thead>
<tr>
<th>Variable (Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>imd</code> A SweptIMDCal (object)</td>
<td></td>
</tr>
<tr>
<td><code>value</code> Maximum IM products to calibrate. Choose from:</td>
<td></td>
</tr>
<tr>
<td>2 - second order products</td>
<td></td>
</tr>
<tr>
<td>3 - third order products</td>
<td></td>
</tr>
<tr>
<td>5 - fifth order products</td>
<td></td>
</tr>
<tr>
<td>7 - seventh order products</td>
<td></td>
</tr>
<tr>
<td>9 - ninth order products</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**  
Long Integer

**Default**  
3

**Examples**  
`imd.MaxProduct = 7 'Write`

`mprod = imd.MaxProduct 'Read`

**C++ Syntax**  
HRESULT get_MaxProduct(long * Val)

HRESULT put_MaxProduct(long newVal)

**Interface**  
ISweptIMD

---

**Last Modified:**

9-Sep-2008   MX New topic
### Mean Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the mean value of the measurement. To retrieve all 3 statistics value at the same time, use <code>meas.GetTraceStatistics</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>average = meas.Mean</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>average</strong> <em>(single)</em> - Variable to store mean value</td>
</tr>
<tr>
<td></td>
<td><strong>meas</strong> - A Measurement <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Single</td>
</tr>
<tr>
<td>Default</td>
<td>Not applicable</td>
</tr>
</tbody>
</table>
| Examples    | ```vbnet  
Dim average as Single  
average = meas.Mean 'Read```                                                                 |
| C++ Syntax  | HRESULT get_Mean(float* mean)                                                                                                   |
| Interface   | IMeasurement                                                                                                                    |
MeasurementClass Property

Description
Returns the measurement class name from the channel. Use CreateCustomMeasurementEx to create a measurement from a class other than standard S-Parameters.

VB Syntax
class = chan.MeasurementClass

Variable
(Type) - Description

class (string) - Variable to store the returned measurement class name.

chan Channel (object)

Return Type
String

Default
Not Applicable

Examples
class = chan.MeasurementClass "Read For a standard S-Parameter channel, returns...
"Standard"

C++ Syntax
HRESULT get_MeasurementClass();

Interface
IChannel15

Last Modified:
6-Oct-2008 Corrected IChannel
12-May-2008 MX New topic
### Medium Property

**Description**  
Sets and Returns the media type of the calibration standard.

**VB Syntax**  
```vbnet
calstd.Medium = value
```

**Variable**  
**Type** - Description


- `value`: (enum NACalStandardMedium) - Medium of the transmission line of the standard. Choose from:
  - **0 - naCoax**: Coaxial Cable
  - **1 - naWaveGuide**:

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**

```vbnet
  calstd.Medium = naCoax  'Write
  stdMedium = calstd.Medium  'Read
```

**C++ Syntax**

```cpp
HRESULT get_Medium(tagNACalStandardMedium *pVal)
HRESULT put_Medium(tagNACalStandardMedium newVal)
```

**Interface**

ICalStandard
# Memory Property

**Description**  
Set and return the memory trace color for nth trace in a window.

**VB Syntax**  
`trace(n).Memory = value`

**Variable**  
**Type** - Description  
- `trace(n)` One of the 8 `ComTraceColors` objects  
- `value` (Long Integer) - RGB color of the Memory pen.  
  Convert the three RGB colors to an integer as follows:  
  \[
  \text{RGB} = R + (G \times 2^8) + (B \times 2^{16})
  \]
  To find the three RGB values from the Display Colors dialog, click **Change Color**, then **Define Custom Color**.

**Return Type**  
Long

**Default**  
Varies for each trace.

**Examples**  
\[
\begin{align*}
R &= 10 \\
G &= 10 \\
B &= 10 \\
\text{RGB} &= 10 + (10 \times 2^8) + (10 \times 2^{16})
\end{align*}
\]

```vbnet
trace1.Memory = RGB 'Write
color = trace1.Memory 'Read
```

**C++ Syntax**  

```cpp
HRESULT get_Memory(long* pVal);
HRESULT put_Memory(long newVal);
```

**Interface**  
IComTraceColors

---

Last Modified:

7-Aug-2009    MX New topic
### MemoryMarkers Property

**Description**
Set and return the color of memory trace markers for nth trace in a window.

**VB Syntax**

```
trace(n).MemoryMarkers = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>trace(n)</td>
<td>One of the 8 ComTraceColors objects</td>
</tr>
</tbody>
</table>
| value   | (Long Integer) - RGB color of the MemoryMarkers pen. Convert the three RGB colors to an integer as follows:

\[
RGB = R + (G \times 2^8) + (B \times 2^{16})
\]

To find the three RGB values from the Display Colors dialog, click Change Color, then Define Custom Color.

**Return Type**
Long

**Default**
Varies for each trace.

**Examples**

```
R = 10
G = 10
B = 10
RGB = 10 + (10 \times 2^8) + (10 \times 2^{16})
trace1.MemoryMarkers = RGB 'Write
color = trace1.MemoryMarkers 'Read
```

**C++ Syntax**

```c++
HRESULT get_MemoryMarkers(long* pVal);
HRESULT put_MemoryMarkers(long newVal);
```

**Interface**
IComTraceColors

---

Last Modified:
7-Aug-2009    MX New topic
MinimumFrequency Property

**Description**
Sets and Returns the minimum frequency for the calibration standard.

**VB Syntax**
`calsd.MinimumFrequency = value`

**Variable (Type) - Description**
- `value` (double): Minimum frequency in Hertz.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
`calsd.MinimumFrequency = 300e3 'Write`

`minFrequency = calsd.MinimumFrequency 'Read`

**C++ Syntax**
```
HRESULT get_MinimumFrequency(double *pVal)
HRESULT put_MinimumFrequency(double newVal)
```

**Interface**
ICalStandard
## MinimumFrequency Property

**Description**
Returns the minimum frequency of the remote PNA.

**VB Syntax**
`value = cap.MinimumFrequency`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>value</code></td>
<td>(Double) - Variable to store the returned minimum frequency of the PNA.</td>
</tr>
<tr>
<td><code>cap</code></td>
<td>A Capabilities (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not Applicable

**Examples**
`value = cap.MinimumFrequency` 'Read

**C++ Syntax**
`HRESULT get_MinimumFrequency(double *pVal)`

**Interface**
ICapabilities
MinimumFrequency Property

**Description**
Sets and returns the minimum usable frequency specified for the power sensor.

**VB Syntax**
```
pwrSensor.MinimumFrequency = value
```

**Variable (Type) - Description**

- `pwrSensor`:
  - A `PowerSensor` (Object) or
  - A `PowerSensorAsReceiver` (Object)

- `value`:
  - (double) - Frequency in Hertz.

**Return Type**
Double

**Default**
Device dependent

**Examples**
```
pwrSensor.MinimumFrequency = 300e3 'Write
MinFreq = pwrSensor).MinimumFrequency 'Read
```

**C++ Syntax**
```
HRESULT put_MinimumFrequency(double newVal);
HRESULT get_MinimumFrequency(double *pVal);
```

**Interface**
IPowerSensor
IPowerSensorPMAR

---

Last Modified:

25-Aug-2009   Added PMAR
MinimumIFFilterSampleCount Property

**Description**
Returns the Minimum allowed value for the IFFilterSampleCount property for the queried PNA.

**Note:** This setting applies only to the E836X Opt. H11.

**VB Syntax**

```vbnet
type = IfConfig.MinimumIFFilterSampleCount
```

**Variable**

- **Type:** (long)
- **Description:** The minimum allowed value that can be applied to the IFFilterSampleCount property.

- **IfConfig**
  - **Type:** IFFConfiguration (object)

**Return Type**
Long Integer

**Default**
PNA Model number dependent

**Examples**

```vbnet
variable = App.ActiveChannel.IFConfiguration.MinimumIFFilterSampleCount
```

See an example program

**C++ Syntax**

```cpp
HRESULT get_MinimumIFFilterSampleCount( long * pMinSampleCount );
```

**Interface**
IIFConfiguration2
MinimumIFFrequency Property

**Description**
Returns the minimum allowed value for the IFFrequency Property.

**VB Syntax**
```vbnet
value = IfConfig.MinimumIFFrequency
```

**Variable** *(Type)* - Description

*value* *(double)* Variable to store the returned minimum allowed frequency that can be applied to the IFFrequency Property.

*IfConfig* *(object)* IFConfiguration (object)

**Return Type** Double

**Default** Not Applicable

**Examples**
```vbnet
val = App.ActiveChannel.IFConfiguration.MinimumIFFrequency
```

**C++ Syntax**
```cpp
HRESULT get_MinimumIFFrequency( double * pMinFreq);
```

**Interface** IIFConfiguration3

---

Last Modified:
18-Jun-2007  MX New topic
**MinimumNumberOfPoints Property**

**Description**
Returns the minimum possible number of data points for a data trace.

**VB Syntax**

```
value = cap.MinimumNumberOfPoints
```

**Variable**

- **value** (Long) - Variable to store the returned minimum value for number of points.

- **cap** A [Capabilities](object) (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**

```
value = cap.MinimumNumberOfPoints 'Read
```

**C++ Syntax**

```
HRESULT get_MinimumNumberOfPoints(double * minimumNumberOfPoints);
```

**Interface**
ICapabilities
**MinimumReceiverStepAttenuator Property**

**Description**
Returns a value indicating the minimum amount of receiver attenuation.

**VB Syntax**
```
value = cap.MinimumReceiverStepAttenuator(n)
```

**Variable**
- *(Type)* - *Description*
  - `value` *(Double)* - Variable to store the returned minimum value of receiver attenuation.
  - `cap` A **Capabilities** *(object)*
  - `n` *(Long)* - port number to query for minimum value of receiver attenuation

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```
value = cap.MinimumReceiverStepAttenuator'Read
```

**C++ Syntax**
```
HRESULT get_MinimumReceiverStepAttenuator(long portNumber, double * attenuation );
```

**Interface**
ICapabilities
# MinimumSourceALCPower Property

**Description**
Returns a value indicating the minimum amount of source ALC power.

**VB Syntax**
```vbnet
value = cap.MinimumSourceALCPower(n)
```

**Variable**
- **value** (Double) - Variable to store the returned minimum value of source ALC power.
- **cap** (Capabilities (object))
- **n** (Long) - source number to query for the minimum value of source ALC power

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```vbnet
value = cap.MinimumSourceALCPower 'Read
```

**C++ Syntax**
```csharp
HRESULT get_MinimumSourceALCPower(long sourceNum, double * power);
```

**Interface**
ICapabilities
**MixerCharacterizationFile Property**

**Description**
Set the filename of the S2P file used to characterize the calibration mixer.

**VB Syntax**

```
smc.MixerCharacterizationFile = value
```

**Variable** *(Type) - Description*

- **smc**
  An **SMCType** (object)
- **value**
  (String) Full path, file name, and extension of the mixer characterization file.

**Return Type**
String

**Default**
Not applicable

**Example**

```
SMC.MixerCharacterizationFile = "C:/Program Files/Agilent/Network Analyzer/Documents/default.S2P"
```

**C++ Syntax**

```
HRESULT put_MixerCharacterizationFile(BSTR Value);
```

**Interface**
SMCType5
## Mode Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets the type of transform.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>trans.Mode = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>trans</code></td>
<td>A Transform <em>(object)</em></td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(enum NATransformMode)</em> - Choose from:</td>
</tr>
<tr>
<td>0</td>
<td><code>naTransformBandpassImpulse</code></td>
</tr>
<tr>
<td>1</td>
<td><code>naTransformLowpassImpulse</code></td>
</tr>
<tr>
<td>2</td>
<td><code>naTransformLowpassStep</code></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td><code>NATransformMode</code></td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>0 - <code>naTransformBandpassImpulse</code></td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>trans.Mode = naTransformLowpassStep</code> <em>(Write)</em></td>
</tr>
<tr>
<td></td>
<td><code>transmode = trans.Mode</code> <em>(Read)</em></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_Mode(tagNATransformMode *pVal)</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_Mode(tagNATransformMode newVal)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>ITransform</code></td>
</tr>
</tbody>
</table>
Mode Property

**Description**
Sets and returns the stimulus mode for balanced measurements.

**VB Syntax**
`balStim.Mode = value`

**Variable (Type) - Description**
- `balStim` A `BalancedStimulus (object)`
- `value` (Enum `NABALSTIMulus`) - Stimulus Mode. True modes are applicable only with Opt 460 - iTMSA. When a True-Mode is selected, the Balanced port powers are automatically uncoupled.

Choose from:
- 0 - `naSEStim`: Single-Ended stimulus
- 1 - `naTMStim`: True-Mode stimulus
- 2 - `naFTMStim`: Forward only True-Mode stimulus
- 3 - `naRTMStim`: Reversed only True-Mode stimulus

**Return Type**
Enum

**Default**
0 - `naSEStim`: Single-Ended stimulus

**Examples**
```vbnet
balStim.Mode = naTMStim 'Write
variable = balStim.Mode 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Mode(tagNABALSTIMulus *eVal );
HRESULT put_Mode(tagNABALSTIMulus eVal );
```

**Interface**
`IBalancedStimulus`

---

Last Modified:
15-May-2008  MX New topic
Multipler Property

**Description**
Sets and returns the multiplier value to be used when coupling this range to the primary range. Learn more about multiplier value. This setting is valid only if this range is **coupled** to the primary range.

**VB Syntax**
```vbnet
FOMRange.Multiplier = value
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>An <strong>FOMRange</strong> (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Double) - Multiplier value.-(Unitless)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
1

**Examples**
- `fomRange.Multiplier = 2` 'Write
- `Mult = fomRange.Multiplier` 'Read

**C++ Syntax**
```cpp
HRESULT get_Multiplier(double *pVal)
HRESULT put_Multiplier(double pVal)
```

**Interface**
**IFOMRange**

---

1234
**Name Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the Name of the Cal Set.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><em>CalSet.Name = value</em></td>
</tr>
<tr>
<td><strong>Variable (Type)</strong> - <strong>Description</strong></td>
<td></td>
</tr>
<tr>
<td><em>CalSet</em> (object) - A <em>Cal Set</em> object</td>
<td></td>
</tr>
<tr>
<td><em>value</em> (string) - Name of the Cal Set.</td>
<td></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>String</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**Examples**

```vbscript
Dim pna
set pna=CreateObject("AgilentPNA835x.Application")

Dim calsets
set calsets=pna.getcalmanager.calsets

Dim c
for each c in calsets
    wscript.echo c.name
    'Changes the name of CalSet_1
    if c.name="CalSet_1" then c.name="New"
next
```

**C++ Syntax**

```c++
HRESULT get_Name(BSTR *name)
HRESULT put_Name(BSTR name);
```

**Interface**

ICalSet4
### Name (CalKit) Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and Returns a name for the selected calibration kit.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><strong>calKit.Name</strong> = <strong>value</strong></td>
</tr>
<tr>
<td><strong>Variable (Type) - Description</strong></td>
<td></td>
</tr>
<tr>
<td>calKit</td>
<td>A CalKit (object).</td>
</tr>
<tr>
<td>value</td>
<td>(string) - Calibration Kit name. Any string name, can include numerics, period, and spaces; any length (although the dialog box display is limited to about 30 characters).</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>String</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td></td>
</tr>
<tr>
<td>calKit.Name = &quot;MyCalKit&quot;</td>
<td>'Write</td>
</tr>
<tr>
<td>KitName = calKit.Name</td>
<td>'Read</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_Name(BSTR *pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Name(BSTR newVal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ICalKit</td>
</tr>
</tbody>
</table>
Name (PathConfig) Property

**Description**
Returns the name of the current configuration only if NO individual element settings had been changed since selecting or storing a configuration. When element settings change, the path configuration name is cleared.

**VB Syntax**

\[ \text{name} = \text{pathConfig}.\text{Name} \]

**Variable**

- **name** *(String)* Variable to store the returned configuration name.
- **pathConfig** A **PathConfiguration** *(object)*

**Return Type**

String

**Default**

'default' - name of the default factory configuration

**Examples**

\[ \text{name} = \text{pathConf.Name} \]

**C++ Syntax**

HRESULT get_Name(BSTR* ppName)

**Interface**

IPathConfiguration

---

Last Modified:

14-Dec-2006   MX New topic
**Name Property**

Description: Returns the name of the current element object

**VB Syntax**

\[ name = pathElement.Name \]

**Variable**

(Type) - Description

\[ name (String) \] Variable to store the returned element name.

*pathElement* A [PathElement](#) (object)

**Return Type** String

**Default** Not Applicable

**Examples**

\[ name=pathElement.Name \]

**C++ Syntax**

HRESULT get_Name(BSTR* ppName)

**Interface** IPathElement
### Name (ExternalDevice) Property

**Description**
Sets and returns the name of the External Device.

**VB Syntax**
```vbnet
extDev.Name = value
```

**Variable**
**Type** - Description

- `value` (string) - External Device name. Any string name limited to alpha-numeric characters.

**Return Type**
String

**Default**
Device

**Examples**
```
extDev.Name = "MySource" 'Write
extDevName = extDev.Name 'Read
```

See example program to configure [PMAR device](#)
See example program to configure [External Source](#)

**C++ Syntax**
```cpp
HRESULT get_Name(BSTR *pVal)

HRESULT put_Name(BSTR newVal)
```

**Interface**
IEternalDevice

---

**Last Modified:**
31-Jul-2009  MX New topic
**Read-only**

**Name Property**

- **Description**: Returns the name of this FOM range object.
- **VB Syntax**: `value = FOMRange.Name`
- **Variable**
  - **(Type)** - Description
  - `value` *(string)* - Variable to store the returned range name.
- **FOMRange**: An *FOMRange* *(object)*
- **Return Type**: String
- **Default**: Not Applicable
- **Examples**: `Rname = fomRange.Name 'Read`
- **C++ Syntax**: `HRESULT get_Name(BSTR *pRName)`
- **Interface**: IFOMRange

---

**Last Modified:**

8-Mar-2007  Major Modifications
**Name (Measurement) Property**

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Sets (or returns) the Name of the measurement. Measurement names must be unique among the set of measurements. Measurement names cannot be an empty string.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Note:</strong></td>
<td>This is the same name as trace.Name; when one changes, the other changes.</td>
</tr>
</tbody>
</table>

**VB Syntax**

```vbnet
meas.Name = value
```

**Variable**

- **Type**: Description
  - `meas` A Measurement (object)
  - `value` (string) - A user defined name of the measurement

**Return Type**

- String

**Default**

- "CH1_S11_1" - name of the default measurement

**Examples**

```vbnet
meas.Name = "Filter BPass" 'Write
MName = meas.Name 'Read
```

**C++ Syntax**

```cpp
HRESULT get_Name(BSTR *pVal)
HRESULT put_Name(BSTR newVal)
```

**Interface**

- IMeasurement
**Name Property**

**Description**
Sets and returns the name of the Power Sensor (object) to be used as part of a Guided Power Calibration.

**VB Syntax**
```vbnet
pSensor.Name = value
```

**Variable (Type) - Description**
- `pSensor` (object)
- `value` (string) - Name of the power sensor. The power sensor must be already configured as a PMAR device using this name. **Learn how to remotely configure a PMAR device.**

**Return Type**
String

**Default**
Not Applicable

**Examples**
```vbnet
Sensor.Name = "26GHzSensor" 'write
value = pSensor.Name 'Read
```

**C++ Syntax**
```c++
HRESULT get_Name(BSTR *pPsensorName)
HRESULT put_Name(BSTR pPsensorName)
```

**Interface**
IGuidedCalibrationPowerSensor

---

Last Modified:
8-Feb-2011    New topic
### Name (trace) Property

**Description**
Sets or returns the name of the Trace. Use the trace name to identify the trace and refer to the trace in the collection.

*Note:* This is the same name as meas.Name; when one changes, the other changes.

**VB Syntax**
```
trac.Name = value
```

**Variable**
- **(Type)** - Description
  - *trac*  A Trace *(object)*
  - *value*  *(String)* Trace name

**Return Type**
String

**Default**
“CH1_S11_1” - name of the default measurement

**Examples**
```
trace.Name = "myTrace"  'Write

traceName = Name.Trace  'Read
```

**C++ Syntax**

- HRESULT put_Name(BSTR name)
- HRESULT get_Name(BSTR *name)

**Interface**
ITrace
NetworkFilename Property

Description: Specifies the S2P filename to embed or de-embed on the input or output of your mixer measurement.

VB Syntax: object.NetworkFilename(n) = filename

Variable: (Type) - Description

object

SMCType (object) or VMCTYPE (object)

n (Integer) Apply network to input or output of mixer. Choose from:

1 - Input of mixer
2 - Output of mixer

filename (String) Filename of the S2P used for embedding or de-embedding. Use the full path name, file name, and .S2P suffix, enclosed in quotes.

Return Type: String

Default: Not Applicable

Examples: VMC.Filename(2) = "C:/Program Files/Agilent/Network Analyzer/Documents/WaveguideAdapt.S2P"

C++ Syntax:

HRESULT put_NetworkFilename(short networkNum, BSTR filename);
HRESULT get_NetworkFilename(short networkNum, BSTR *filename);

Interface: SMCType2

VMCTYPE2

Last Modified:

29-Feb-2008  Several edits
**NetworkMode Property**

**Description**
Allows you to embed (add) or de-embed (remove) circuit network effects on the input and output of your mixer measurement.

**VB Syntax**
```
object.NetworkMode(n) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td><strong>SMCTYPE</strong> (object) or <strong>VMCTYPE</strong> (object)</td>
</tr>
<tr>
<td>n</td>
<td>(Integer) Apply network to input or output of mixer. Choose from:</td>
</tr>
<tr>
<td></td>
<td>1 - Input of mixer</td>
</tr>
<tr>
<td></td>
<td>2 - Output of mixer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>(Enum as ENetworkMode) Choose from:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>NO_NETWORK</strong></td>
<td>Do nothing with effects of S2P file</td>
</tr>
<tr>
<td><strong>EMBED_NETWORK</strong></td>
<td>Add effects of S2P file from the measurement results.</td>
</tr>
<tr>
<td><strong>DEEMBED_NETWORK</strong></td>
<td>Remove effects of S2P file from the measurement results.</td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
**NO_NETWORK**

**Examples**
```
VMC.NetworkMode = EMBED_NETWORK
```

**C++ Syntax**
```
HRESULT put_NetworkMode(short networkNum, enum ENetworkMode networkMode);
HRESULT get_NetworkMode(short networkNum, enum ENetworkMode *networkMode);
```

**Interface**
```
SMCTYPE2
VMCTYPE2
```
## NetworkPortMap Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Set and return the port mapping for a 4-port SNP file to be embedded.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>fixture.NetworkPortMap (network) = inA, inB, outA, outB</code></td>
</tr>
<tr>
<td><strong>Variable (Type) - Description</strong></td>
<td></td>
</tr>
<tr>
<td><code>fixture</code></td>
<td>A <code>Fixturing</code> (object)</td>
</tr>
<tr>
<td><code>network</code></td>
<td>Network position. Choose from 1 or 2.</td>
</tr>
<tr>
<td><code>inA, inB, outA, outB</code></td>
<td>Port Mapping. Use four port numbers in any order.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Comma-separated long integers</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>1,2,3,4</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>fixture.NetworkPortMap = 1,3,2,4</code></td>
</tr>
<tr>
<td><code>values = fixture.NetworkPortMap 'Read</code></td>
<td></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IFixturing6</td>
</tr>
</tbody>
</table>

Last Modified:

16-Nov-2010    MX New topic
### NoiseAverageFactor Property

**Description**
Sets and reads the averaging of the noise receiver.

**VB Syntax**

```vbnet
noise.NoiseAverageFactor = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>noise</code></td>
<td>A NoiseFigure (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(long integer) - Averaging value. Choose a number between 1 and 99.</td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
1

**Examples**

```vbnet
noise.NoiseAverageFactor = 10 'Write
AvgNoise = noise.NoiseAverageFactor 'Read
```

**C++ Syntax**

```cpp
HRESULT get_NoiseAverageFactor(long* pVal)
HRESULT put_NoiseAverageFactor(long newVal)
```

**Interface**
INoiseFigure

---

Last Modified:

29-May-2007   MN New topic
## NoiseAverageState Property

**Description**
Turns Noise Averaging ON and OFF.

**VB Syntax**

```vbnet
noise.NoiseAverageState = value
```

**Variable (Type) - Description**

- **noise**
  A [NoiseFigure](#) *(object)*

- **value** *(boolean)* - Averaging state.
  - **False** - Turns Noise Averaging OFF
  - **True** - Turns Noise Averaging ON

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
noise.NoiseAverageState = OFF 'Write
NoiseAvgState = noise.NoiseAverageState 'Read
```

### C++ Syntax

- `HRESULT get_NoiseAverageState(VARIANT_BOOL* on);`
- `HRESULT put_NoiseAverageState(VARIANT_BOOL on);`

**Interface**
INoiseFigure

---

Last Modified:

- 21-Jun-2007  MX New topic
**NoiseBandwidth Property**

**Description**
Set the bandwidth of the noise receiver.

**VB Syntax**
```
noise.NoiseBandwidth = value
```

**Variable**
(Type) - Description

- **noise**
  A NoiseFigure (object)

- **value**
  (double) Bandwidth value.
  
  For **NoiseReceiver** = naNoiseReceiver (Opt 029) choose from: 800 KHz, 2 MHz, 4 MHz, 8 MHz, or 24 MHz or the numerical equivalent, such as 8e6 and so forth.
  
  For **NoiseReceiver** = naStandardReceiver (Opt 028) choose from: 720 kHz or 1.2 MHz
  
  If the value does not match one of these, it is rounded up to the next valid bandwidth value.

**Return Type**
Double

**Default**
4 MHz for naNoiseReceiver
1.2 MHz for naStandardReceiver

**Examples**
```
noise.NoiseBandwidth = 2E6 'Write

NoiseBW = noise.NoiseBandwidth 'Read
```

**C++ Syntax**
```
HRESULT get_NoiseBandwidth(double *pVal);
HRESULT put_NoiseBandwidth(double newVal);
```

**Interface**
INoiseFigure

---

Last Modified:

- 21-Apr-2010  Added noise receiver options
- 20-Sep-2007  MX New topic
NoiseReceiver Property

Description  Sets and returns the receiver to use for noise measurements.

VB Syntax  

\[ \text{noise.NoiseReceiver} = \text{value} \]

Variable  (Type) - Description

- **noise**  (Object) 

- **value**  (Enum as NANOiseReceiverMode) Noise receiver. Choose from:
  - 0 - naStandardReceiver  The standard PNA receiver. (Opt 028)
  - 1 - naNoiseReceiver  The noise receiver. (Opt 029)

Return Type  Enum

Default  1 - naNoiseReceiver

Examples  

- `noise.NoiseReceiver = naNoiseReceiver  'Write`

- `NoiseRec = noise.NoiseReceiver  'Read`

C++ Syntax  

- `HRESULT get_NoiseReceiver(tagNoiseReceiverMode *pVal);`
- `HRESULT put_NoiseReceiver(tagNoiseReceiverMode newVal);`

Interface  INoiseFigure5

Last Modified: 2-Mar-2010  MX New topic
NoiseReceiverSweepTime Property

Description
Returns the APPROXIMATE time the channel will take to make one noise receiver sweep given the current setup. This, along with the sweep time for a standard receiver measurement and the following calculations, can tell you how long a “single” sweep would take so that you can set an appropriate “timeout” in your program.

Use **Sweep Time Property** to perform the standard sweep time query, shown as **SSwpTime** below.

**To calculate the total sweep time:**
Noise Figure on amplifiers (Vector Correction ON):

- \( 2 \times \text{SSwpTime} + X \times \text{NoiseReceiverSweepTime} \)
  
  Where \( X = \) the number of noise receiver impedance state sweeps. (Default is 4).

Noise Figure on converters (NFX) correction on - increased number of sweeps due to extra mixer sweeps and source pulling:

- \( 4 \times \text{SSwpTime} + X \times \text{NoiseReceiverSweepTime} \) (without source pulling)
  
  \( 8 \times \text{SSwpTime} + X \times \text{NoiseReceiverSweepTime} \) (with source pulling)
  
  Where \( X = \) the number of noise receiver impedance state sweeps. (Default is 4).

**Note:** The number of sweeps to perform a noise measurement is annotated at the bottom of the Noise Figure screen.

**VB Syntax**

```
swpTime = NF.NoiseReceiverSweepTime
```

**Variable**

*(Type) - Description*

- **swpTime** Variable to store the returned sweep time value (in seconds).
- **NF** A **NoiseFigure** *(object)*

**Return Type**
Double

**Default**
Not Applicable

**Examples**

```
swpTime = NF.NoiseReceiverSweepTime()
```

**C++ Syntax**

```
HRESULT get_NoiseReceiverSweepTime (Double* SwpTime);
```

**Interface**
INoiseFigure3
Write/Read

NoiseGain Property

Description
Sets and reads the gain state of the noise receiver. This setting is NOT used when NoiseReceiver = naStandardReceiver (Opt 028)

VB Syntax
noise.NoiseGain = value

Variable
Type - Description

noise A NoiseFigure (object)

value (long integer) - Gain value. Choose from:
0 Low Gain  select if the gain of your DUT is relatively high (>35 dB).
15 Medium Gain  select if the gain of your DUT is about average (20 dB to 45 dB).
30 High Gain ..select if the gain of your DUT is relatively low (<30 dB).
If the value does not match one of these, it is rounded up to the next legal value.

Learn more about Noise Receiver Gain setting.

Return Type
Long Integer

Default
30

Examples
noise.NoiseGain = 30 'Write
GainNoise = noise.NoiseGain 'Read

C++ Syntax
HRESULT get_NoiseGain(long* pVal)
HRESULT put_NoiseGain(long newVal)

Interface
INoiseFigure

Last Modified:
29-May-2007   MN New topic
NoiseSourceCalKitType Property

Description
Set and read the Cal Kit that will be used for the Noise Source adapter.

An adapter is always necessary to connect a 346C Noise Source to the PNA port 2. Select a Cal Kit that is the same type and gender as the noise source connector.

If the Noise Source mates directly to PNA port 2, then set this type to "None".

VB Syntax
noise.NoiseSourceCalKitType = value

Variable (Type) - Description
noise A NoiseCal (object)
value (string) Cal Kit type. To read possible cal kit strings for the adapter:

1. Change the port connector type to that of the noise source using: ConnectorType
2. Then read the possible cal kit strings for that port using: CompatibleCalKits

Return Type
String

Default
Not applicable

Examples
noise.NoiseSourceCalKitType = "N4691-60004 ECAL" 'Write
calkit = noise.NoiseSourceCalKitType 'Read

C++ Syntax
HRESULT get_NoiseSourceCalKitType(BSTR* pValue)
HRESULT put_NoiseSourceCalKitType(BSTR pNewValue)

Interface
INoiseCal

Last Modified:
29-May-2007  MN New topic
### NoiseSourceCold Property

**Description**
Sets and returns the temperature of the noise source connector.

**VB Syntax**
`noise.NoiseSourceCold = value`

**Variable**
- **Type** - Description
- `noise` (A NoiseCal object)
- `value` (double) Noise source temperature in Kelvin.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```
noise.NoiseSourceCold = 295 'Write

temp = noise.NoiseSourceCold 'Read
```

**C++ Syntax**
- `HRESULT get_NoiseSourceCold(double* pTemp)`
- `HRESULT put_NoiseSourceCold(double pNewTemp)`

**Interface**
INoiseCal

---

Last Modified:
- 21-Jun-2007   MX New topic
**NoiseSourceConnectorType Property**

**Description**  Set and read the Noise Source connector type and gender.

The Agilent 346C has an "APC 3.5 male" connector.

**VB Syntax**  

```vbnet
noise.NoiseSourceConnectorType = value
```

**Variable (Type) - Description**

- `noise`  A `NoiseCal` (object)

- `value`  (string) Connector type. Use `ValidConnectorType` to return a list of valid connector types.

**Return Type**  String

**Default**  Not applicable

**Examples**

```vbnet
noise.NoiseSourceConnectorType = "APC 3.5 male"  'Write

connector = noise.NoiseSourceConnectorType  'Read
```

**C++ Syntax**

```cpp
HRESULT get_NoiseSourceConnectorType(BSTR* pConnectorType)
HRESULT put_NoiseSourceConnectorType(BSTR pConnectorType)
```

**Interface**  INoiseCal

---

Last Modified:

29-May-2007  MN New topic
About Noise Figure

NoiseSourceState Property

**Description**
Sets and reads the noise source (28V) ON and OFF.

**VB Syntax**
```
app.NoiseSourceState = state
```

**Variable**
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
<tr>
<td>state</td>
<td>(boolean)</td>
</tr>
<tr>
<td></td>
<td>False (0) - Turns Noise Source OFF</td>
</tr>
<tr>
<td></td>
<td>True (1) - Turns Noise Source ON</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
For PNA models with a Noise Figure option (028/029/H29), the 28V line is ON at application start and after a preset. The ON/OFF state is also available from a PNA softkey menu.

For PNA models WITHOUT a Noise Figure option (028/029/H29), the 28V line is OFF at application start and it’s state is not affected by a preset. The ON/OFF state is NOT available from a PNA softkey menu.

**Examples**
```
app.NoiseSourceState = True 'Write

coupl = app.NoiseSourceState 'Read
```

**C++ Syntax**
```
HRESULT put_NoiseSourceState(VARIANT_BOOL bState)
HRESULT get_NoiseSourceState(VARIANT_BOOL *bState)
```

**Interface**
IApplication13

---

Last Modified:

<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3-Nov-2010</td>
<td>Available for PNA models without a NF option.</td>
</tr>
<tr>
<td>29-May-2007</td>
<td>MN New topic</td>
</tr>
</tbody>
</table>
### NoiseTuner Property

#### Description
Sets and returns the noise tuner identifier, which is an ECAl model and serial number string.

#### VB Syntax

```vbnet
noise.NoiseTuner = value
```

#### Variable *(Type)* - Description

**noise**
- A [NoiseFigure](#) *(object)*

**value** *(string)* Noise Tuner. Return the connected ECAl identifiers by sending GetCalKitTypeString and passing the module number.

#### Return Type
String

#### Default
Not Applicable

#### Examples

```vbnet
noise.NoiseTuner = "N4691-60004 ECAl 02822"  'Write

noiseT = noise.NoiseTuner  'Read
```

#### C++ Syntax

```csharp
HRESULT get_NoiseTuner(BSTR* pValue)

HRESULT put_NoiseTuner(BSTR pNewValue)
```

#### Interface
INoiseFigure

---

**Last Modified:**

18-Jun-2007  MN New topic
**NoiseTunerIn Property**

**Description**  Sets and returns the port identifier of the ECal noise tuner that is connected to the PNA Source.

**VB Syntax**  

```vb
noise.NoiseTunerIn = value
```

**Variable**  

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>noise</td>
<td>A NoiseFigure (object)</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>(string) Noise Tuner port identifier that is connected to the PNA source, as it is labeled on the ECal module. For example, for 2-port ECal modules, choose either &quot;A&quot; or &quot;B&quot;.</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**  String

**Default**  Not Applicable

**Examples**  

```vb
noise.NoiseTunerIn = "A" 'Write
EcalPort = noise.NoiseTunerIn 'Read
```

**C++ Syntax**  

```cpp
HRESULT get_NoiseTunerIn(BSTR* pValue)
HRESULT put_NoiseTunerIn(BSTR pNewValue)
```

**Interface**  INoiseFigure

Last Modified:  29-May-2007  MN New topic
**NoiseTunerOut Property**

**Description**
Sets and returns the port identifier of the ECal noise tuner that is connected to the DUT.

**VB Syntax**

`noise.NoiseTunerOut = value`

**Variable**

- `noise` (Type) - Description
  - `noise` (A NoiseFigure (object))

- `value` (string) Noise Tuner port identifier that is connected to the DUT, as it is labeled on the ECal module. For example, for 2-port ECal modules, choose either "A" or "B".

**Return Type**

String

**Default**

Not Applicable

**Examples**

`noise.NoiseTunerOut = "A"`  

`EcalPort = noise.NoiseTunerOut`  

**C++ Syntax**

- `HRESULT get_NoiseTunerOut(BSTR* pValue)`  
- `HRESULT put_NoiseTunerOut(BSTR pNewValue)`

**Interface**

INoiseFigure

---

**Last Modified:**

29-May-2007   MN New topic
NominalIncidentPowerState Property

Description
Toggles the Nominal Incident Power setting ON and OFF. This setting is ONLY to be used with SMC measurements, not VMC. Learn more about Nominal Incident Power.

VB Syntax
`mixer.NominalIncidentPowerState = bool`

Variable (Type) - Description
- `mixer` A Mixer (object)
- `bool` (boolean) - Nominal Incident Power State. Choose from:
  - 1 -(True) Turn nominal incident power ON
  - 0 -(False) Turn nominal incident power OFF

Return Type
Boolean

Default
0 -(False)

Examples
`mixer.NominalIncidentPowerState = True 'sets NominalIncidentPowerState to ON`

C++ Syntax
`HRESULT get_NominalIncidentPowerState(VARIANT_BOOL *pVal)`
`HRESULT put_NominalIncidentPowerState(VARIANT_BOOL val );`

Interface
IMixer4
**NormalizePoint Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the sweep data point around which to perform broadband and precise tuning.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>obj.NormalizePoint = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>obj</code></td>
<td>An <strong>EmbeddedLO (object)</strong> or A <strong>ConverterEmbeddedLO (object)</strong></td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(Long)</strong> Mixer Sweep data point. Choose a data point number between 1 and the max number of data points in the sweep that has the least amount of expected noise.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td><strong>(Long)</strong></td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Center point in the sweep span</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>embedLO.NormalizePoint = 101 'write</code></td>
</tr>
<tr>
<td></td>
<td><code>value = embedLO.NormalizePoint 'read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_NormalizePoint(long *point);</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_NormalizePoint(long point);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IEmbeddedLO</td>
</tr>
</tbody>
</table>

Last Modified:

- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
**NormalizePoint Property**

**Description**  
Sets or returns the data point used for normalizing the phase measurement.

**VB Syntax**  
`obj.NormalizePoint = value`

**Variable (Type) - Description**

- `obj`  
  A [Mixer Interface](#) pointer to the [Measurement](#) (object)  
  A [Converter](#) (Object)

- `value`  
  (Long) - Normalization data point. Choose a data point number between 1 and the max number of data points in the sweep that has the least amount of expected noise.

**Return Type**  
Long Integer

**Default**  
Center point in the sweep

**Examples**  
`mixer.NormalizePoint = 101`

**C++ Syntax**  
`HRESULT getNormalizePoint(Long * val);`  
`HRESULT putNormalizePoint(Long val);`

**Interface**  
IMixer13

Last Modified:  
25-Mar-2010   MX New topic
**Number (Measurement) Property**

**Description**
Returns the Number of the measurement. Measurement numbers are assigned internally.

**Note**: Measurement numbers are NOT the same as their number in the Measurements collection. Measurement number is used to identify the measurement associated with an event.

This property is used to identify measurements when events occur through the `OnMeasurementEvent` callback. For example:

```
OnMeasurementEvent (naEventId_MSG_LIMIT_FAILED, 3)
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>measNum</code></td>
<td><code>long</code></td>
<td>variable to store the measurement number</td>
</tr>
<tr>
<td><code>meas</code></td>
<td><code>A Measurement (object)</code></td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
"1" - number of the default measurement

**Examples**
`measNum = meas.Number`

**C++ Syntax**
`HRESULT get_Number(long *MeasurementNumber)`

**Interface**
`IMeasurement`
**Number Property**

**Description**
Returns the number of the Auxiliary Trigger connector pair currently being used with the instance of the AuxTrigger object. Set the trigger pair with the AuxTrig object.

**VB Syntax**
```
value = auxTrig.Number
```

**Variable (Type) - Description**
- **auxTrig** An AuxTrigger (object)
- **value** (Long Integer) - Connector pair. PNA-X returns 1 or 2. All other models that do not have the Aux trigger connector returns 1.

**Return Type**
Single

**Default**
Not Applicable

**Examples**
```
value = auxTrig.Number 'Read the value
```

**C++ Syntax**
```
HRESULT get_Number(long *auxID);
```

**Interface**
IAuxTrigger

---

Last Modified:
- 7-Apr-2009  Updated to "all other models..
- 14-Dec-2006  MX New topic
Number Of Frequency Points Property

**Description**
Set and read the number of frequency points for a Gain Compression channel. Applies to all acquisition modes.

**VB Syntax**
`gca.NumberOfFrequencyPoints = value`

**Variable**

- **Type**: `GainCompression` (object)
- **Variable**: `value` (integer) - Frequency points. Do not exceed the max number of points. [Learn more.]

**Return Type**
Integer

**Default**
201

**Examples**
```vbnet
gca.NumberOfFrequencyPoints = 101 'Write
freqPts = gca.NumberOfFrequencyPoints 'Read
```

**C++ Syntax**
```cpp
HRESULT get_NumberOfFrequencyPoints(int* pVal)
HRESULT put_NumberOfFrequencyPoints(int newVal)
```

**Interface**
`IGainCompression`

---

**Last Modified:**
11-Sep-2007   MX New topic
**NumberOfPoints Property**

**Description**
Sets or returns the Number of Points of the channel.
Sets or returns the Number of Points of the segment.

**See Also**
- [Measurement2 Interface](#) to learn how this method differs from `meas.NumberofPoints`
- [Gain Compression Number of Points](#)
- [Swept IMD limitations](#)

**VB Syntax**
```
object.NumberOfPoints = value
```

**Variable**
- `(Type)`: Description
  - `object` Channel *(object)*
  - or
  - CalSet *(object)* - Read-only property

- `value` *(long)*: Number of Points.
  - For channel, choose any number from 1 to the [PNA max number of points](#).
  - For segment, the total number of points in all segments cannot exceed the PNA maximum. A segment can have as few as 1 point.

**Return Type**
Long Integer

**Default**
- 201 for channel
- 21 for segment

**Examples**
```
chan.NumberOfPoints = 201 'sets the number of points for all measurements in the channel. -Write
```
```
numofpts = chan.NumberOfPoints 'Read
```

**C++ Syntax**
```
HRESULT get_NumberOfPoints(long *pVal)
HRESULT put_NumberOfPoints(long newVal)
```

**Interface**
- IChannel
- ISegment
- |CalSet3

---

Last Modified:
<table>
<thead>
<tr>
<th>Date</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>5-Aug-2008</td>
<td>Fixed bad link</td>
</tr>
<tr>
<td>21-Jun-2007</td>
<td>Increased max</td>
</tr>
</tbody>
</table>
### Read-only

**NumberOfPoints Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the number of data points of the measurement. To understand how this property is useful, see <a href="#">IMeasurement2 Interface</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>value = meas.NumberOfPoints</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(Long)</em> - variable to store the returned value</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Long Integer</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>Print meas.NumberOfPoints 'prints the number of data points</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT get_NumberOfPoints(long *pVal);</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement2</td>
</tr>
</tbody>
</table>
Read-only

**NumberOfPorts Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the number of hardware source ports on the PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>value = app.NumberOfPorts</code></td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(long integer) - variable to contain the returned value</td>
</tr>
<tr>
<td>Return Type</td>
<td>(long integer)</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>iNumPorts = app.NumberOfPorts</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT NumberOfPorts( long* NumPorts)</td>
</tr>
<tr>
<td>Interface</td>
<td>IApplication</td>
</tr>
</tbody>
</table>
### NumberOfPorts Property

**Description**
Returns the number of ports on the specified testset. Returns 0 if no test set is connected.

**VB Syntax**
```
value = tset.NumberOfPorts
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Long) variable to store the returned information.</td>
</tr>
</tbody>
</table>

| tset | A `TestsetControl` object. |
| OR   | An `E5091Testset` object. |

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```
value = testset1.NumberOfPorts
```

See E5091A Example Program

See External Testset Program

**C++ Syntax**
```
HRESULT get_NumberOfPorts(long *numberOfPorts);
```

**Interface**
- `ITestsetControl`
- `IE5091Testset`
### NumberOfPowerPoints Property

**Description**
Set and read the number of data points in each power sweep. Applies ONLY to 2D acquisition modes.

**VB Syntax**
```
gca.NumberOfPowerPoints = value
```

**Variable**
- **Type**: GainCompression (object)
- **value**: integer - Power points. Do not exceed the max number of points.
  - For 2D sweeps, total = frequency x power. Max = 20,001
  - For Smart sweep, total = frequency. Max = 10,000.

**Return Type**
Integer

**Default**
26

**Examples**
```
gca.NumberOfPowerPoints = 31 'Write
pwrPts = gca.NumberOfPowerPoints 'Read
```

**C++ Syntax**
```
HRESULT get_NumberOfPowerPoints(int* pVal)
HRESULT put_NumberOfPowerPoints(int newVal)
```

**Interface**
IGainCompression

---

Last Modified:

11-Sep-2007    MX New topic
**NumberOfSweeps Property**

- **Description**: Returns the number of tuning sweeps used for the latest embedded LO measurement.

- **VB Syntax**: `value = embedLODiag.NumberOfSweeps`

- **Variable** *(Type)* - Description
  - `value` *(Long)*: Variable to store the returned data.
  - `embedLODiag` [An EmbeddedLODiagnostic (object)]

- **Return Type**: (Long)

- **Default**: Not Applicable

- **Examples**: `data = embedLODiag.NumberOfSweeps 'read`

- **C++ Syntax**: `HRESULT get_NumberOfSweeps(long * numSweeps);`

- **Interface**: IEmbededLODiagnostic

---

**Last Modified:**

12-Apr-2007  MX New topic
Offset Property

Description
Sets and returns the offset value to be used when coupling this range to the primary range.
This setting is valid only if the specified range is coupled to the primary range.

VB Syntax
FOMRange.Offset = value

Variable (Type) - Description
object An FOMRange (object)
value (Double) - Offset value.-(Unitless)

Return Type Double

Default 0

Examples
fomRange.Offset = 1e9 'Write
Offs = fomRange.Offset 'Read

C++ Syntax
HRESULT get_Offset(double *pVal)
HRESULT put_Offset(double pVal)

Interface IFOMRange

Last Modified: 8-Mar-2007 Modified links.
OffsetReceiverAttenuator Property

**Description**
Set and return whether to offset the reference receiver by the amount of receiver attenuation.

Learn more.

This setting remains until changed or until the hard drive is changed or reformatted.

**VB Syntax**
```vbnet
pref.OffsetReceiverAttenuator = value
```

**Variable**
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pref</td>
<td>A Preferences (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Boolean) - Choose from:</td>
</tr>
<tr>
<td></td>
<td>False Do NOT offset the test port receivers.</td>
</tr>
<tr>
<td></td>
<td>True Offset the test port receivers.</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
True PNA-X models
False E836xB and PNA-L models

**Examples**
```vbnet
pref.OffsetReceiverAttenuator = 1 'Write
Rcvroffset = pref.OffsetReceiverAttenuator 'Read
```

**C++ Syntax**
```c++
HRESULT get_OffsetReceiverAttenuator(VARIANT_BOOL * val);
HRESULT put_OffsetReceiverAttenuator(VARIANT_BOOL val);
```

**Interface**
IPreferences6

**Last Modified:**
14-Jan-2007 MX New topic
OffsetSourceAttenuator Property

**Description**
Set and return whether to mathematically offset the reference receivers by the amount of source attenuation. [Learn more.]

This setting remains until changed or until the hard drive is changed or reformatted.

**VB Syntax**

```vbnet
pref.OffsetSourceAttenuator = value
```

**Variable**
- **Type** - Description
  - `pref` A [Preferences](#) (object)
  - `value` (Boolean) - Choose from:
    - `False` Do NOT offset the reference receivers.
    - `True` Offset the reference receivers.

**Return Type**

Boolean

**Default**

- **True** PNA-X models
- **False** E836xB and PNA-L models

**Examples**

```vbnet
pref.OffsetSourceAttenuator = 1 'Write
Rcvroffset = pref.OffsetSourceAttenuator 'Read
```

**C++ Syntax**

- HRESULT get_OffsetSourceAttenuator(VARIANT_BOOL * val);
- HRESULT put_OffsetSourceAttenuator(VARIANT_BOOL val);

**Interface**

IPreferences6

---

**Last Modified:**

14-Jan-2007  MX New topic
Read/Write

OmitIsolation Property - Superseded

Description
This command is replaced with SetIsolationPaths and GetIsolationPaths. Sets and returns whether Isolation portion of the calibration will be performed or not.

VB Syntax

obj.OmitIsolation = bool

Variable

(obj) - Description

obj - SMCType (object)
   or
   VMCType (object)

bool - (Boolean)
   True - Isolation is NOT performed
   False - Isolation is performed

Return Type
Boolean

Default
True

Examples
value = SMC.OmitIsolation

C++ Syntax

HRESULT put_OmitIsolation (VARIANT_BOOL bState)
HRESULT get_OmitIsolation (VARIANT_BOOL *bState)

Interface
SMCType
VMCType
OneMarkerReadoutPerTrace Property - Superseded

**Description**
Either show marker readout of only the active trace or all of the traces simultaneously.

**Note:** This method is replaced by MarkerReadoutsPerTrace Property

**VB Syntax**
```
win.OneReadoutPerTrace = state
```

**Variable (Type) - Description**

*win*  
A *NAWindow* (object)

*value* (boolean)
*True* - Shows the readout of only the active marker for each trace.
*False* - Shows up to 5 marker readouts per trace, up to 20 total readouts.

**Return Type**
Boolean

**Default**
False

**Examples**
```
win.OneMarkerReadoutPerTrace = True 'Write
State = app.ActiveNAWindow.OneMarkerReadoutPerTrace 'Read
```

**C++ Syntax**
```
HRESULT get_OneMarkerReadoutPerTrace(VARIANT_BOOL *pVal)
HRESULT put_OneMarkerReadoutPerTrace(VARIANT_BOOL newVal)
```

**Interface**
INAWindow
## Options Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns a string identifying the analyzer option configuration.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>value = app.Options</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(string) - variable to contain the returned string</td>
</tr>
<tr>
<td>Return Type</td>
<td>String</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>availOptions = app.Options</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT Options(BSTR* OptionString)</td>
</tr>
<tr>
<td>Interface</td>
<td>IApplication</td>
</tr>
</tbody>
</table>
Write/Read

OrientECALModule Property

Description
Specifies if the PNA should perform orientation of the ECal module during calibration. Orientation is a technique by which the PNA automatically determines which ports of the module are connected to which ports of the PNA. Orientation begins to fail at very low power levels or if there is much attenuation in the path between the PNA and the ECal module.

Note: For guided calibrations where Orient is OFF and the same ECal module is used in more than one Connection Step, you are not allowed to specify how the ECal module is connected. Instead, the PNA determines the orientation. The PNA does not verify that you made the connection properly.

This setting remains until the PNA is restarted or this command is sent again.

This command, and ECALPortMapEx, can be used to perform ECal using the Guided Calibration interface.

VB Syntax

cal.OrientECALModule = value

Variable (Type) - Description

cal A Calibrator (object)
value (boolean)

False - DoECAL1PortEx and DoECAL2PortEx methods will use the value of the ECALPortMapEx property to determine the port connections.

True - DoECAL1PortEx and DoECAL2PortEx methods will use auto Orientation technique to determine port connections.

Return Type
Boolean

Default
True

Examples
Dim cal As Calibrator
Dim bOrient As Boolean
Set cal = PNAapp.ActiveChannel.Calibrator
cal.OrientECALModule = False 'Write
bOrient = cal.OrientECALModule 'Read

C++ Syntax

HRESULT put_OrientECALModule(VARIANT_BOOL bOrient);
HRESULT get_OrientECALModule(VARIANT_BOOL *bOrient);

Interface
ICalibrator3

Last Modified:
7-May-2007  Added note about Guided Cal.
OutputFixedFrequency Property

Description
Sets or returns the mixer output fixed frequency value.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

\[ \text{obj.OutputFixedFrequency} = \text{value} \]

Variable (Type) - Description

obj A Mixer Interface pointer to the Measurement (object)
Or
A Converter Object

value (double) - Output Fixed Frequency in Hertz.

Return Type
Double

Default
Not Applicable

Examples

Print mixer.OutputFixedFrequency 'prints the output fixed frequency value of the mixer.

C++ Syntax

HRESULT get_OutputFixedFrequency(double *pVal)
HRESULT put_OutputFixedFrequency(double newVal)

Interface
IMixer3
IConverter

Last Modified:

2-Feb-2009  Added Converter
4-Mar-2008  Added note.
OutputPort Property

Description
Switches an input to one of the valid outputs on the specified E5091A. The following are valid input/output combinations. If a combination other than these are sent, an “invalid argument” error will occur.

<table>
<thead>
<tr>
<th>Input</th>
<th>Valid Outputs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A</td>
</tr>
<tr>
<td></td>
<td>T1 - If Port 2 already is connected to T1, then Port 2 will be switched to T2.)</td>
</tr>
<tr>
<td>2</td>
<td>T1 - If Port 1 already is connected to T1, then Port 1 will be switched to A.</td>
</tr>
<tr>
<td></td>
<td>T2</td>
</tr>
<tr>
<td>3</td>
<td>R1+</td>
</tr>
<tr>
<td></td>
<td>R2+</td>
</tr>
<tr>
<td></td>
<td>R3+ If option 007 (7port), R2 is selected.</td>
</tr>
<tr>
<td>4</td>
<td>R1-</td>
</tr>
<tr>
<td></td>
<td>R2-</td>
</tr>
<tr>
<td></td>
<td>R3- If option 007 (7port), R2 is selected.</td>
</tr>
</tbody>
</table>

Note: Do not confuse the similar Testset. OutputPorts Property, which sets or gets the port mapping for ALL ports.

VB Syntax
\[ \text{testsets}(1).\text{OutputPort} \ (\text{chNum},\text{input}) \ = \ \text{output} \]

Variable (Type) - Description

testsets(1) An item from Testsets (collection)
Learn how to identify a testset in the collection.

chNum (Long) Channel number of the measurement.

input (Long) Testset Input port. Choose from 1|2|3|4.

output (Enum as NAE5091OutputPort) Output port to switch to specified Input. Choose from:
0 or naE5091PortA - Port A
1 or naE5091PortT1 - Port T1
2 or naE5091PortT2 - Port T2
3 or **naE5091PortR1** - Port R1
4 or **naE5091PortR2** - Port R2
5 or **naE5091PortR3** - Port R3

**Return Type**  Enum

**Default**  Not Applicable

**Examples**  See Example Program

**C++ Syntax**

```cpp
HRESULT get_OutputPort(long channelNum, long inputPort, E5091OutputPort *outPort);
HRESULT put_OutputPort(long channelNum, long inputPort, E5091OutputPort outPort);
```

**Interface**  IE5091Testsets
**OutputPorts (Cal Set) Property**

**Description**
Returns the port mapping for the Cal Set.

**VB Syntax**

```vbnet
portMap = calset.OutputPorts
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>portMap</code></td>
<td>(String) Variable to store the returned string. The returned values are the physical ports. The POSITION of the returned values corresponds to the logical ports. For example, with an N44xx test set, if the returned string is &quot;PNA 1,TS 2,PNA 2, TS 4&quot; this means:</td>
</tr>
<tr>
<td><code>calset</code></td>
<td>A <code>Cal Set</code> object.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Depends on the test set.

**Example**

```vbnet
portMap = calset.OutputPorts
```

**C++ Syntax**

```cpp
HRESULT get_OutputPorts(BSTR *mapping);
```

**Interface**
ICalset5

Last modified:
9/18/06 MQ Added for multiport
### OutputPorts Property

**Description**
Sets or gets the port mappings for ALL ports. An “invalid argument” error will occur if you attempt to set an illegal port combination. 
Refer to your testset documentation for valid port combinations.  
**Note:** Do not confuse the similar E5091. **OutputPort Property**, which sets or gets the port mapping for a single port.

**VB Syntax**
```
tset.OutputPorts(chNum) = portList
```

**Variable (Type) - Description**
- **tset** A TestsetControl object.
- **chNum** (Long) Channel number of the measurement.
- **portList** (String) A comma-separated list of port mappings. Spaces are ignored at the beginning and end of this text, and before or after commas. Space characters in other locations are not ignored.

**Return Type** String

**Default** Not Applicable

**Examples**
See External Testset Program

**C++ Syntax**
```
HRESULT get_OutputPorts(long channelNum, BSTR *outPorts);
HRESULT put_OutputPorts(long channelNum, BSTR outPorts);
```

**Interface** ITestsetControl
OutputRangeMode Property

Description
Sets or returns the Output sweep mode.
If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

Note: There is also a OutputRangeMode Property on the Converter Object.

VB Syntax

obj.OutputRangeMode = value

Variable (Type) - Description

obj A Mixer Interface pointer to the Measurement (object)

value (Enum as MixerRangeMode) - Output sweep mode. Choose from:
0 - mixSwept
1 - mixFixed

Return Type Enum

Default 0 - mixSwept

Examples mixer.OutputRangeMode = mixSwept

C++ Syntax

HRESULT get_OutputRangeMode(long *pVal)
HRESULT put_OutputRangeMode(long newVal)

Interface IMixer6

Last Modified:
4-Mar-2008  Added note.
OutputSideband Property

Description
Specify whether to select the sum (High) or difference (Low) products.

- When one LO is used: Input (+ or -) LO1 = Output frequency.
- When two LO’s are used: IF (+ or -) LO2 = Output frequency. See Also: IFSideband Property

If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

VB Syntax

```vbnet
obj.OutputSideband = value
```

Variable (Type) - Description

- **obj**: A Converter Object
- **value** (enum as ConverterSideBand) - Choose from:
  - 0 - or **naLowSide**: Minus (-) on the Mixer setup dialog
  - 1 - or **naHighSide**: Plus (+) on the Mixer setup dialog

Return Type
Enum as ConverterSideBand

Default
naLowSide

Examples

- `Print converter.OutputSideband 'prints the value of the OutputSideband`

C++ Syntax

```cpp
HRESULT get_OutputSideband(ConverterSideBand *pVal)
HRESULT put_OutputSideband(ConverterSideBand newVal)
```

Interface
IConverter

Last Modified:

26-Mar-2009   New topic
OutputSideband Property

**Description**
Specify whether to select the sum (High) or difference (Low) products.

- When one LO is used: Input + or - LO1 = Output frequency
- When two LOs are used: IF1 + or - LO2 = Output frequency

Use **IFSideband_Property** when two LOs are used to determine the IF1 frequency.

If you are changing several mixer configuration settings, you can make all the changes first and then issue the **Calculate** and **Apply** commands as you would do from the user interface.

**Note:** There is also an **OutputSideband_Property** on the Converter Object.

**VB Syntax**

```vbnet
mixer.OutputSideband = value
```

**Variable** *(Type) - Description*

- **mixer** A **Mixer (object)**
- **value** *(FCASideBand)* - Choose from:
  - 0 - **LOW** Minus (-) on the Mixer setup dialog
  - 1 - **HIGH** Plus (+) on the Mixer setup dialog

**Return Type**
Enum as FCASideBand

**Default**
LOW

**Examples**

```
Print mixer.OutputSideband 'prints the value of the OutputSideband
```

**C++ Syntax**

```
HRESULT get_OutputSideband(FCASideBand *pVal)
HRESULT put_OutputSideband(FCASideBand newVal)
```

**Interface**
IMixer

---

**Last Modified:**

- 2-Oct-2008 Clarified 1 vs 2 LOs
- 4-Mar-2008 Added note.
About Mixer Configuration

OutputStartFrequency Property

- **Description**: Sets or returns the mixer output start frequency.
  If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

- **VB Syntax**: `obj.OutputStartFrequency = value`

- **Variable**
  - **(Type)** - Description
  
  - **obj**: A Mixer Interface pointer to the Measurement (object)  
  Or  
  A Converter Object

  - **value**: (double) - Output Start Frequency in Hertz.

- **Return Type**: Double

- **Default**: Not Applicable

- **Examples**: Print mixer.OutputStartFrequency 'prints the value of the OutputStartFrequency

- **C++ Syntax**
  
  - HRESULT get_OutputStartFrequency(double *pVal)
  - HRESULT put_OutputStartFrequency(double newVal)

- **Interface**: IMixer
  IConverter

---

**Last Modified:**

- 2-Feb-2009  Added Converter

- 4-Mar-2008  Added note.
### OutputStopFrequency Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the mixer Output Stop frequency. If you are changing several mixer configuration settings, you can make all the changes first and then issue the <strong>Calculate</strong> and <strong>Apply</strong> commands as you would do from the user interface.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>obj.OutputStopFrequency = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>obj</code></td>
<td>A <strong>Mixer Interface</strong> pointer to the <strong>Measurement</strong> (object) Or A <strong>Converter Object</strong></td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(double)</strong> - Output stop frequency in Hertz.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Double</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td>Print mixer.OutputStopFrequency  <code>'prints the value of the OutputStopFrequency</code></td>
</tr>
</tbody>
</table>
| **C++ Syntax** | `HRESULT get_OutputStopFrequency(double *pVal)`  
`HRESULT put_OutputStopFrequency(double newVal)` |
| **Interface** | IMixer  
IConverter |

---

**Last Modified:**

- 2-Feb-2009   Added Converter
- 4-Mar-2008   Added note.
## Parameter Property

**Description**
Returns the measurement Parameter. To change the parameter, use `meas.ChangeParameter`.

**VB Syntax**
`measPar = meas.Parameter`

**Variable**
- **measPar** *(string)* - Variable to store Parameter string
- **meas** A Measurement *(object)*

**Return Type**
String

**Default**
Not applicable

**Examples**
`measPar = meas.Parameter` *Read*

**C++ Syntax**
`HRESULT get_Parameter(BSTR *pVal)`

**Interface**
IMeasurement
Parameter (Embedded LO) Property

**Description**
Returns the name of the parameter of the specified tuning sweep.

**VB Syntax**
```
value = embedLODiag.Parameter (n)
```

**Variable**
**Type** - Description
- **value** *(String)* Variable to store the returned data.
- **embedLODiag** *(EmbeddedLODiagnostic (object))*
- **n** *(Long)* Tuning sweep number. Use **NumberOfSweeps** to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**
```
data = embedLO.Parameter 3 ' read
```

**C++ Syntax**
```
HRESULT Parameter(long sweep, BSTR * param);
```

**Interface**
IEmbededLODiagnostic

Last Modified:
13-Apr-2007   MX New topic
**Read-only**

**Parent Property**

**Description**
Returns a handle to the parent object of the collection object being referred to in the statement. The parent property allows the user to traverse from an object back up the object hierarchy.

**VB Syntax**
`collection.Parent`

**Variable**
- **(Type)** - Description
  - `collection` - Description
    - `CalFactorSegments collection`
    - `CalFactorSegmentsPMAR Collection`
    - `Channels collection`
    - `E5091Testset Collection`
    - `ECalModules Collection`
    - `ExternalDevices Collection`
    - `ExternalTestsets Collection`
    - `Measurements collection`
    - `NaWindows collection`
    - `PowerLossSegments collection`
    - `PowerLossSegmentsPMAR Collection`
    - `PowerSensors collection`
    - `Segments collection`
    - `Traces collection`
    - `PathConfigurationManager`
    - `PowerMeterInterfaces Collection`

**Return Type**
Object

**Default**
Not Applicable

**Examples**
parentobj = chans.Parent 'returns a handle to the parent object (Application) of the chans collection. -Read

**C++ Syntax**
HRESULT get_Parent(IApplication* *pApplication); //IChannels, IChannel, IMeasurements, INAWindows, and IExternalDevices
HRESULT get_Parent(IChannel* *pChannel); //ITraces
HRESULT get_Parent(INAWindow* *pWindow); //ISegments
HRESULT get_Parent(IPowerSensor* *pSensor); //ICalFactorSegments(PMAR)
HRESULT get_Parent(ISourcePowerCalibrator* *pCalibrator); //IPowerSensors, IPowerLossSegments, and IPowerMeterInterfaces
<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>31-Jul-2009</td>
<td>Added ExternalDevices (9.0)</td>
</tr>
<tr>
<td>5-Mar-2009</td>
<td>Added PowerMeterInterfaces</td>
</tr>
</tbody>
</table>
**Read/Write**

*PassFailLogic Property*

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Sets and reads the logic of the PassFail line on the <strong>HANDLER IO connector</strong> (pin 33) and <strong>AUX IO connector</strong> (pin 12). Learn more about Global Pass/Fail. Note: This line is connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.</th>
</tr>
</thead>
</table>

**VB Syntax**  
`object.PassFailLogic = value`

**Variable**  
*(Type) - Description*

- `object` *(object)* - An Aux I/O or Handler I/O object
- `value` *(enum as NARearPanelIOLogic)* Choose from:
  - `0 - naPositiveLogic` - Causes the PassFail line to have positive logic (high = pass, low = fail).
  - `1 - naNegativeLogic` - Causes the PassFail line to have negative logic (high = fail, low = pass).

**Return Type**  
Long Integer

**Default**  
naPositiveLogic

**Examples**

```
aux.PassFailLogic = naNegativeLogic 'Write  
Text1.Text = aux.PassFailLogic 'Read
```

**C++ Syntax**

```
HRESULT put_PassFailLogic ( tagNARearPanelIOLogic Mode );
HRESULT get_PassFailLogic ( tagNARearPanelIOLogic* Mode );
```

**Interface**

- IHWAuxIO
- IHWMaterialHandlerIO
PassFailMode Property

Description
Sets and reads the mode of the PassFail line on the HANDLER IO connector (pin 33) and AUX IO connector (pin 12). Learn more about Global Pass/Fail.

Note: This line is connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.

VB Syntax

\[
\text{object.PassFailMode} = \text{value}
\]

Variable (Type) - Description

\text{object} (object) - An Aux I/O or Handler I/O object

\text{value} (enum as NAPassFailMode) - Choose from:

0 - \text{naDefaultPassNoWaitMode} - the line stays in PASS state. When a device fails, then the line goes to fail IMMEDIATELY.

1 - \text{naDefaultPassWaitMode} - the line stays in PASS state. When a device fails, then the line goes to fail after the Sweep End line is asserted.

2 - \text{naDefaultFailWaitMode} - the line stays in FAIL state. When a device passes, then the line goes to PASS state after the Sweep End line is asserted.

Return Type
Long Integer

Default
0 - \text{naDefaultPassNoWaitMode}

Examples

\text{HWAuxIO.PassFailMode} = \text{naDefaultPassNoWaitMode} \ 'Write

\text{mode} = \text{HWAuxIO.PassFailMode} \ 'Read

C++ Syntax

\[
\text{HRESULT put_PassFailMode ( tagNAPassFailMode Mode );}
\text{HRESULT get_PassFailMode ( tagNAPassFailMode* Mode );}
\]

Interface
IHWAuxIO
IHWMaterialHandlerIO
PassFailPolicy Property

**Description**
Sets the policy used to determine how global pass/fail is computed. [Learn more about Global Pass/Fail](#).

**VB Syntax**

```vbnet
object.PassFailPolicy = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>An Aux I/O or Handler I/O object</td>
</tr>
</tbody>
</table>

**value**

(enumer as NAPassFailPolicy) Choose from:

- 0 - `naPolicyAllTests` - Pass/Fail Status returns PASS if all tests on all measurements pass.
- 1 - `naPolicyAllMeas` - Pass/Fail Status returns PASS if all measurements have associated tests, and all tests pass. FAIL is returned if even one measurement has no associated limit test.

Only those measurements which are not in HOLD mode contribute to the pass/fail result.

**Return Type**

Long Integer

**Default**

`naPolicyAllTests`

**Examples**

```vbnet
matHndler.PassFailPolicy = naPolicyAllTests 'Write
policy = aux.PassFailPolicy 'Read
```

**C++ Syntax**

```cpp
HRESULT put_PassFailPolicy ( tagNAPassFailPolicy Policy);
HRESULT get_PassFailPolicy ( tagNARearPanelIOLogic* Policy);
```

**Interface**

IHWAuxI04

IHWMaterialHandlerI02
Read/Write

PassFailScope Property

Description
Sets and reads the Scope of the PassFail line on the HANDLER IO connector (pin 33) and AUX IO connector (pin 12). Learn more about Global Pass/Fail.

Note: The PassFail line is connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.

VB Syntax

```
object.PassFailScope = value
```

Variable (Type) - Description

- **object** (object) - An Aux I/O or Handler IO object
- **value** (enum NAPassFailScope) Choose from:
  - 0 - naChannelScope - The PassFail line returns to its default state before sweeps on the next channel start. (A channel measurement may require several sweeps.)
  - 1 - naGlobalScope - The PassFail line returns to its default state before the sweeps for the next triggerable channel start.

The default state of the PassFail line before a measurement occurs and after a failure occurs is set by the PassFailMode property.

Return Type
enum NAPassFailScope

Default
1 - naGlobalScope

Examples

```
HWAuxIO.PassFailScope = naGlobalScope 'Write

scope = HWAuxIO.PassFailScope 'Read
```

C++ Syntax

```
HRESULT put_PassFailScope ( tagNAPassFailScope Scope );
HRESULT get_PassFailScope ( tagNAPassFailScope* Scope );
```

Interface
IHWAuxIO
IHWMaterialHandlerIO
**Read-Only**

**PassFailStatus Property**

**Description** Returns the most recent pass/fail status value. Use this command as follows:

1. Set the PNA **trigger scope** to GLOBAL
2. Set the PNA **trigger source** to MANUAL or EXTERNAL.
3. Configure and enable **Limit Testing**
4. Trigger the PNA.
5. Use the **OPC?** (with **SCPIStringParser object**) to determine when the sweep is complete.
6. Use the **PassFailStatus** property to obtain the global pass/fail result.

Learn more about **Global Pass/Fail**.

**VB Syntax**

```
var = object.PassFailStatus
```

**Variable** *(Type) - Description*

<table>
<thead>
<tr>
<th>var</th>
<th>(enum as NAPassFailStatus) Variable to store returned status. One of the following will be returned:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td><strong>naStatusFail</strong> - all measurements not in HOLD mode have been swept, and one or more limit tests failed according to the specified Pass/Fail policy.</td>
</tr>
<tr>
<td>1</td>
<td><strong>naStatusPass</strong> - all measurements not in HOLD mode have been swept, and all associated limit tests have passed.</td>
</tr>
<tr>
<td>2</td>
<td><strong>naStatusNone</strong> - status cannot be determined because measurements are in progress.</td>
</tr>
</tbody>
</table>

| object | (object) - An **Aux I/O** or **Handler I/O** object |

**Return Type** Long Integer

**Default** Not Applicable

**Examples**

```
status = aux.PassFailStatus 'Read
```

**C++ Syntax**

```
HRESULT get_PassFailPolicy ( tagNAPassFailStatus* status);
```

**Interface**

- IHWAuxIO4
- IHWMaterialHandlerIO2

---

1300
### Path Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Specifies an interface to use for the power meter / sensor during a source power calibration.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>pwrMtrInterface.Path = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td><code>pwrMtrInterface</code> (object)</td>
<td>A <a href="#">PowerMeterInterface</a> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(enum as NACommunicationPath)</strong> Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - <strong>naGPIB</strong> - GPIB interface</td>
</tr>
<tr>
<td></td>
<td>1 - <strong>naUSB</strong> - USB interface</td>
</tr>
<tr>
<td></td>
<td>2 - <strong>naLAN</strong> - LAN interface</td>
</tr>
<tr>
<td>Return Type</td>
<td>Enum</td>
</tr>
<tr>
<td>Default</td>
<td><strong>naGPIB</strong></td>
</tr>
<tr>
<td>Examples</td>
<td><a href="#">See example</a></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT put_Path(tagNACommunicationPath pNewVal);</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT get_Path(tagNACommunicationPath *pVal);</code></td>
</tr>
<tr>
<td>Interface</td>
<td><a href="#">IPowerMeterInterface</a></td>
</tr>
</tbody>
</table>

Last Modified: 2-Aug-2007  MX New topic
PathCalMethod Property

Description

Note: This command replaces ThruCalMethod. (Read-Write) Specifies the calibration method for each port pair.

Note: Do NOT send this command to rely on SmartCal to determine the most accurate cal method for your connector settings and Cal Kits. You can send the query form of this command to learn the cal method determined by SmartCal.

Before using this command, first do the following:

- Set the connector types: ConnectorType
- Set cal kit: CalKitType
- Set or query the thru path pairs: ThruPortList

After sending or querying this command, send the Thru method: PathThruMethod

See an example of a 4-port guided calibration using COM.

VB Syntax

guidedCal.PathCalMethod (port1, port2) = "caltype1[,caltype2]"

Variable (Type) - Description

guidedCal GuidedCalibration (object)

port1 First port of the pair to be calibrated.

port2 Second port of the pair to be calibrated.

"caltype1[,caltype2]" (String) Cal types for 1st and 2nd ports of the pair, enclosed in a single pair of quotes. NOT case-sensitive.

caltype1 Cal type for the pair if caltype2 is not specified. Otherwise, Cal type for port 1. Choose from:

- “TRL”
- “SOLT”
- “QSOLTN”
- “EnhRespN”

For the last two arguments, replace N with the port to be used as the source port, which MUST be one of the port pair.

[caltype2] Optional argument. Use only when performing an adapter removal cal on the pair. This argument specifies the Cal Type on the second port; caltype1 then specifies the Cal Type of the first port.
Choose from the same arguments as `caltype1`.

**Return Type**  
**String** - Returns comma-separated cal types.

**Default**  
The most accurate cal method for the current cal.

**Example**  
```cpp
guidedCal.PathCalMethod(2,3) = "TRL"  'Write trl for port pair

guidedCal.PathCalMethod(1,4) = "TRL,SOLT"  'Write adapter removal cal, consisting of trl on port 1 and solt on port 4

calmethod = guided.PathCalMethod(1,4)  'Read previous example, returns: "TRL,SOLT"
```

**C++ Syntax**  
```cpp
HRESULT get_PathCalMethod(long firstport, long secondport, BSTR *calMethod);
HRESULT put_PathCalMethod(long firstport, long secondport, BSTR calMethod);
```

**Interface**  
`IGuidedCalibration3`

Last modified:

- 6-Apr-2011  
  Added default note.

- April 9, 2007  
  MX New topic
PathThruMethod Property

Description  

Note: This command replaces ThruCalMethod.  

(Read-Write) Specifies the calibration THRU method for each port pair.  

Note: Do NOT send this command to rely on SmartCal to determine the most accurate THRU method for your connector settings and Cal Kits. You can send the query form of this command to learn the THRU method determined by SmartCal.

Before sending this command, first do the following:

- Set the connector types: ConnectorType
- Set cal kit: CalKitType
- Set or query the thru path pairs: ThruPortList
- Set or query the Cal Type: PathCalMethod

To determine the default Thru method:

- Set the connector types: ConnectorType
- Set cal kit: CalKitType
- Send GenerateSteps
- Read the proposed port pairs: ThruPortList
- Read this command: (calmethod = guided.PathThruMethod(p1, p2)

See an example of a [4-port guided calibration using COM](https://example.com).

VB Syntax  

guidedCal.PathThruMethod (port1, port2) = "ThruType1[,ThruType2]"

Variable  

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GuidedCal</td>
<td>GuidedCalibration (object)</td>
</tr>
<tr>
<td>port1</td>
<td>First port of the pair to be calibrated.</td>
</tr>
<tr>
<td>port2</td>
<td>Second port of the pair to be calibrated.</td>
</tr>
</tbody>
</table>

"ThruType1[,ThruType2]"  

(String) Thru methods for 1st and 2nd ports of the pair, enclosed in a single pair of quotes. NOT case-sensitive.

thruType1  

Calibration thru method for the pair if thruType2 is not specified. Otherwise, thru method for port 1.

Choose from:

- “Defined Thru” A thru type for which there is a stored definition in the Cal Kit.
- “Zero Thru” Zero length thru, also known as flush-thru.
• “Undefined Thru” A thru type for which there is NOT a stored definition in the Cal Kit. Also known as Unknown Thru. Valid ONLY for SOLT cal type.

• “Undefined Thru using a Defined Thru” Using an ECal module, measure the internal thru using the “Undefined Thru” method.

**ThruType2** (String) Optional argument. Use only when performing an adapter removal cal on the pair as determined by **PathCalMethod**. The only valid arguments for ThruType1&2 (when specified) is “Defined Thru, Defined Thru”.

**Return Type**  **String** - Returns comma-separated ThruTypes.

**Default** The most accurate THRU method for the current cal.

**Example**
```
guidedCal.PathThruMethod(2,3) = "Zero Thru" 'Write for port pair

guidedCal.PathThruMethod(1,4) = "Defined Thru, Defined Thru" 'Write for adapter removal cal.

calmethod = guided.PathThruMethod(1,4) 'Read previous example,
return: "Defined Thru, Defined Thru"
```

**C++ Syntax**
```
HRESULT get_PathThruMethod(long firstport, long secondport, BSTR *thruMethod);
HRESULT put_PathThruMethod(long firstport, long secondport, BSTR thruMethod);
```

**Interface** IGuidedCalibration3

Last modified:

6-Apr-2011   Added default note

18-Feb-2011   Replacement command

9-Apr-2007    MX New topic
## PeakExcursion Property

**Description**
Sets and reads the peak excursion value for the specified marker. The Excursion value determines what is considered a "peak".

**VB Syntax**

```vbnet
mark.PeakExcursion = value
```

**Variable**

- **mark**
  
  A Marker (object)

- **value**
  
  (single) - Peak Excursion. Choose any number between -500 and 500

**Return Type**

Single

**Default**

3

**Examples**

```vbnet
mark.PeakExcursion = 1 'Write

PkExcur = mark.PeakExcursion 'Read
```

**C++ Syntax**

```csharp
HRESULT get_PeakExcursion(float *pVal)
HRESULT put_PeakExcursion(float newVal)
```

**Interface**

IMarker
**PeakThreshold Property**

**Description**
Sets peak threshold for the specified marker. If a peak (using the criteria set with PeakExcursion) is below this reference value, it will not be considered when searching for peaks.

**VB Syntax**

```
mark.PeakThreshold = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>mark</td>
<td>A Marker (object)</td>
</tr>
<tr>
<td>value</td>
<td>(single) - Peak Threshold. Choose any number between: -500 and 500</td>
</tr>
</tbody>
</table>

**Return Type**
Single

**Default**
-100db

**Examples**

```
mark.PeakThreshold = 1 'Write
PkThresh = mark.PeakThreshold 'Read
```

**C++ Syntax**

```
HRESULT get_PeakThreshold(float *pVal)
HRESULT put_PeakThreshold(float newVal)
```

**Interface**
IMarker
### PeakToPeak Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the Peak to Peak value of the measurement. To retrieve all 3 statistics value at the same time, use <code>meas.GetTraceStatistics</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>pp = meas.PeakToPeak</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td></td>
<td><code>pp</code> <em>(single)</em> - Variable to store peak-to-peak value</td>
</tr>
<tr>
<td></td>
<td><code>meas</code> A Measurement <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Single</td>
</tr>
<tr>
<td>Default</td>
<td>Not applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>pp = meas.PeakToPeak</code> 'Read'</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT get_PeakToPeak(float* pp)</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>
**Percent Property**

**Description**
Sets group delay aperture using a percent of the channel frequency span.

**VB Syntax**
`gdAperture.Percent = value`

**Variable (Type) - Description**

- **gdAperture**
  - Type: `GroupDelayAperture` (object)

- **value**
  - Type: `Double`
  - Description: Percent of frequency span to use for the aperture setting. Choose between the equivalent of 2 points and 100 percent of the channel frequency span.

**Return Type**
Double

**Default**
Percent range that equates to 11 points.

This can be changed to two points with a preference setting.

**Examples**
```
gdAperture.Percent = 25 'Write
aperture = gdAperture.Percent 'Read
```

**C++ Syntax**
```
HRESULT get_Percent(double Percent *pVal)
HRESULT put_Percent(double Percent newVal)
```

**Interface**
IGroupDelayAperture

---

**Last Modified:**
23-Feb-2010   MX New topic
PerformPowerCalibration Property

**Description**  
Enables Guided Power Cal and sets the source port to be calibrated.

**VB Syntax**  
guidedCal.**PerformPowerCalibration** (port) = value

**Variable**  
*(Type) - Description*

- **guidedCal**  
  GuidedCalibration (object)

- **port**  
  *(Long integer)*  
  Source port to be calibrated. ONLY one port may be calibrated with a Guided Power Cal.

- **value**  
  *(Boolean)*

  - **True**  
    Perform a Guided Power Calibration.

  - **False**  
    Do NOT perform a Guided Power Calibration

**Return Type**  
Boolean

**Default**  
False

**Example**  
guided.PerformPowerCalibration(1) = True

See example program

**C++ Syntax**  
HRESULT get_PerformPowerCalibration(long port,VARIANT_BOOL* val);
HRESULT put_PerformPowerCalibration(long port,VARIANT_BOOL newVal);

**Interface**  
IGuidedCalibration7

---

Last Modified:

9-Sep-2010   MX New topic
**Period Property**

**Description**
Sets the pulse-period (1/PRF) for ALL PNA-X internal pulse generators. The resolution of the period is 16.667nS.

**VB Syntax**
`pulse.Period = value`

**Variable**
- **Type** - Description
  - `pulse` A [PulseGenerator](#) (object)
  - `value` (Double) Pulse period in seconds. Choose a value from about 33ns to about 70 seconds.

**Return Type**
Double

**Default**
1e-3 sec

**Examples**
- `pulse.Period = 1ms 'Write`
- `value = pulse.Period 'Read`

**C++ Syntax**
```cpp
HRESULT get_Period(double* period);
HRESULT put_Period(double period);
```

**Interface**
IPulseGenerator

---

Last Modified:
1-Jan-2007    MX New topic
Write/Read

PhaseOffset Property

Description
Sets the Phase Offset for the active channel.

VB Syntax
meas.PhaseOffset = value

Variable
(meas - Description

(meas - A Measurement (object)

(value - (double) - PhaseOffset in degrees. Choose any number between:
-360 and +360

Return Type
Double

Default
0

Examples
meas.PhaseOffset = 25 'Write
poffset = meas.PhaseOffset 'Read

C++ Syntax
HRESULT get_PhaseOffset(double *pVal)
HRESULT put_PhaseOffset(double newVal)

Interface
IMeasurement
Write/Read

PhaseAsFixture Property

**Description**
Sets and reads the state of phase offset as a fixture with True Mode balanced measurements. Learn more about iTMSA phase and power offset.

**VB Syntax**
```vbnet
balStim.PhaseAsFixture = value
```

**Variable**
- **Type** - Description
  - `balStim` A `BalancedStimulus` (object)
  - `value` (Boolean) State of phase offset as a fixture.
    - **False** Offset is applied but is NOT included as a fixture in the output calculations.
    - **True** Offset is applied and included as a fixture in the output calculations.

**Return Type**
Boolean

**Default**
False

**Examples**
```
balStim.PhaseAsFixture = False 'Write
var = balStim.PhaseAsFixture 'Read
```

**C++ Syntax**
```cpp
HRESULT get_PhaseAsFixture (VARIANT_BOOL *bVal)
HRESULT put_PhaseAsFixture (VARIANT_BOOL bVal)
```

**Interface**
`IBalancedStimulus`

---

Last Modified:
27-May-2008    MX New topic
PhaseControlMode Property

**Description**  
Write and read the Phase Control mode for the specified port.

**VB Syntax**  
`phase.PhaseControlMode(srcPort) = value`

**Variable**  
**(Type)** - Description

- **phase**: A `PhaseControl` Object
- **srcPort**: (Long Integer) Source port for which to make phase control settings.
  
  **Note**: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**: (Enum as NAPhaseControlMode) Choose from:
  
  0 - `naPhaseControlOff`  
  Do NOT control the phase of <port>.

  1 - `naPhaseControlParameter`  
  Control the phase of <port>.

  2 - `naPhaseControlPaired`  
  (READ-only) - reference port for a 'controlled' (parameter) Port.

  3 - `naPhaseControlOpenLoop`  
  Sets a raw phase value for either swept phase or fixed phase, but no receivers are used to control the phase.

**Return Type**  
Enum

**Default**  
0 - `naPhaseControlOff`

**Examples**  
```vbnet
phase.PhaseControlMode 1 = naPhaseControlParameter ' Write

value = phase.PhaseControlMode 2' Read
```

**C++ Syntax**  
```cpp
HRESULT get_PhaseControlMode(long port, enum NAPhaseControlMode* pVal);
HRESULT put_PhaseControlMode(long port, enum NAPhaseControlMode newVal);
```

**Interface**  
`IPhaseControl`

---

Last Modified:

8-Dec-2010    MX New topic
**PhaseCorrectionData Property**

**Description**
Write and read an array of phase offsets. Each phase offset is summed with each phase point to get new target value. The PNA attempts to set phase for each target value. The number of phase offset values must be the same as the number of data points.

**VB Syntax**
```
phase.PhaseCorrectionData(srcPort) = value
```

**Variable**
(Description)

- **phase**
  A PhaseControl Object

- **srcPort**
  (Long Integer) Source port for which to make phase control settings.

  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**
  (Long values) Phase offset data array.

**Return Type**
Long data array

**Default**
Not Applicable

**Examples**
```
phase.PhaseCorrectionData 1 = 10,11,12 ' Write 3 phase offset values
value = phase.PhaseCorrectionData 2 ' Read
```

**C++ Syntax**
```
HRESULT get_PhaseCorrectionData(long port, long* pVals);
HRESULT put_PhaseCorrectionData(long port, long newVals);
```

**Interface**
IPhaseControl

---

**Last Modified:**
8-Dec-2010  MX New topic
Write/Read

PhaseCorrectionEnabled Property

**Description**
Write and read whether to use the phase correction offset array. Use `PhaseCorrectionData` to write or read the offset data.

**VB Syntax**
```
phase.PhaseCorrectionEnabled(srcPort) = value
```

**Variable**
(Length) - Description

- `phase` (A `PhaseControl` Object)
- `srcPort` (Long Integer) Source port for which to make phase control settings.
  
  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- `value` (Boolean) Phase correction array state.
  - `True` – Apply phase correction offset array.
  - `False` – Do NOT apply phase correction offset array.

**Return Type**
Boolean

**Default**
False

**Examples**
```
phase.PhaseCorrectionEnabled 1 = True ' Write
value = phase.PhaseCorrectionEnabled 2 ' Read
```

**C++ Syntax**
```
HRESULT get_PhaseCorrectionEnabled(long port, VARIANT_BOOL* pVal);
HRESULT put_PhaseCorrectionEnabled(long port, VARIANT_BOOL newVal);
```

**Interface**
`IPhaseControl`

---

Last Modified:
8-Dec-2010   MX New topic
PhaseIterationNumber Property

**Description**  Write and read the maximum number of background phase sweeps to perform.

**VB Syntax**  

phase.PhaseIterationNumber(srcPort) = value

**Variable (Type) - Description**

*phase*  A PhaseControl Object

*srcPort*  (Long Integer) Source port for which to make phase control settings.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port

*value*  (Double) Number of background sweep iterations. Choose a value between 1 and 25.

**Return Type**  Double

**Default**  5

**Examples**

phase.PhaseIterationNumber 1 = 15  ' Write

value = phase.PhaseIterationNumber 2' Read

**C++ Syntax**

HRESULT get_PhaseIterationNumber(long port, double* pVal);
HRESULT put_PhaseIterationNumber(long port, double newVal);

**Interface**  IPhaseControl

Last Modified:

8-Dec-2010   MX New topic
**PhaseParameter Property**

**Description**
Write and read the ratioed receivers (parameter) and paired port to use for phase control.

**VB Syntax**

```vbnet
phase.PhaseParameter(srcPort) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>phase</td>
<td>A PhaseControl Object</td>
</tr>
<tr>
<td>srcPort</td>
<td>(Long Integer) Source port for which to make phase control settings.</td>
</tr>
<tr>
<td>Note:</td>
<td>If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use <code>chan.getPortNumber</code> to translate the string into a port number. To learn more see <a href="#">Remotely Specifying a Source Port</a>.</td>
</tr>
<tr>
<td>value</td>
<td>(String) Ratioed parameter. Choose any two PNA physical receivers. Use either standard receiver notation (&quot;R/R3&quot;) or logical receiver notation (&quot;a1/a3&quot;). Separate the two receiver names by a forward slash '/'. For example: &quot;a1/a3&quot;.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
"a1/b1"

**Examples**

```vbnet
phase.PhaseParameter 1 = "a1/a3" ' Write
value = phase.PhaseParameter 2' Read
```

**C++ Syntax**

```c++
HRESULT get_PhaseParameter(long port, BSTR* pVal);
HRESULT put_PhaseParameter(long port, BSTR newVal);
```

**Interface**
IPhaseControl

---

Last Modified:

8-Dec-2010  MX New topic
# PhaseParameterModes Property

**Description**  
Returns the available phase control modes for the specified port. Use `PhaseControlMode` to set the Phase Control mode.

**VB Syntax**  
```vbnet
phase.PhaseParameterModes(srcPort) = value
```

**Variable**  
**Type** - Description

- `phase`  
  A `PhaseControl` Object

- `srcPort`  
  (Long Integer) Source port for which to make phase control settings.
  
  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- `value`  
  (String) Choose from:
  
  - "Off" - Turn phase control OFF
  - "Openloop" - Sets a raw phase value for either swept phase or fixed phase, but no receivers are used to control the phase.
  - "Parameter" - Sets and controls the phase of the signal at <port>.
  - "Reference" - Reference port for a controlled (parameter) port. Use `PhaseReferencePort` to set the reference port.

**Return Type**  
String

**Default**  
"Off"

**Example**  
```vbnet
value = phase.PhaseParameterModes 2' Read
```

**C++ Syntax**  
```c++
HRESULT get_PhaseParameterModes(long port, BSTR* pVal);
```

**Interface**  
IPhaseControl

---

**Last Modified:**

- 8-Dec-2010  
  MX New topic
PhaseReferencePort Property

Description
Sets and returns the reference port for the Phase Control measurement.

VB Syntax
`phase.PhaseReferencePort(srcPort) = value`

Variable (Type) - Description

`phase` A PhaseControl Object

`srcPort` (Long Integer) Source port for which to make phase control settings.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

`value` (Long Integer) Reference port number. ONLY specific ports are available to be a reference for each source port. Learn more.

Return Type
Long Integer

Default
Depends on the controlled port.

Examples
```
phase.PhaseReferencePort 1 = 3 ' Write
```

```
value = phase.PhaseReferencePort 2 ' Read
```

C++ Syntax
```
HRESULT get_PhaseReferencePort(long port, long* pVal);

HRESULT put_PhaseReferencePort(long port, long newVal);
```

Interface
IPhaseControl

Last Modified:
8-Dec-2010    MX New topic
**PhaseSwpAsFixture Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and reads the state of phase offset as a fixture with True Mode balanced measurements.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>balStim.PhaseAsFixture = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><code>(Type) - Description</code></td>
</tr>
<tr>
<td><code>balStim</code></td>
<td>A <code>BalancedStimulus</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Boolean) State of phase offset as a fixture.</td>
</tr>
<tr>
<td>False</td>
<td>Offset is applied but is NOT included as a fixture in the output calculations.</td>
</tr>
<tr>
<td>True</td>
<td>Offset is applied and included as a fixture in the output calculations.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Boolean</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>False</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>balStim.PhaseAsFixture = False</code> 'Write'</td>
</tr>
<tr>
<td></td>
<td><code>var = balStim.PhaseAsFixture</code> 'Read'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_PhaseAsFixture (VARIANT_BOOL *bVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_PhaseAsFixture (VARIANT_BOOL bVal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IBalancedStimulus2</td>
</tr>
</tbody>
</table>

Last Modified:

3-Mar-2009    MX New topic (A.08.50)
PhaseSwpState Property

Description  Sets and reads the state of phase sweep. Sweep type must be set to CWTime.

VB Syntax  

```
balStim.PhaseSwpState  = value
```

Variable  

- **balStim**  
  A BalancedStimulus (object)
- **value**  
  (Boolean) State of phase sweep.  
  - **False**  
    Phase sweep disabled.  
  - **True**  
    Phase sweep enabled.

Return Type  Boolean

Default  False

Examples  

```
balStim.PhaseSwpState = False  'Write
var = balStim.PhaseSwpState  'Read
```

C++ Syntax  

```
HRESULT get_PhaseSwpState (VARIANT_BOOL *bVal)
HRESULT put_PhaseSwpState (VARIANT_BOOL bVal)
```

Interface  IBalancedStimulus2

Last Modified:  

3-Mar-2009  MX New topic (A.08.50)
PhaseTolerance Property

**Description**  Write and read the tolerance value to be used for background phase sweeps.

**VB Syntax**  
```
phase.PhaseTolerance(srcPort) = value
```

**Variable**  
- **(Type)** - Description
  - **phase**  A `PhaseControl` Object
  - **srcPort**  (Long Integer) Source port for which to make phase control settings.
    - **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).
  - **value**  (Double) Tolerance for background sweeps in degrees. Choose a value between 1 and 5.

**Return Type**  Double

**Default**  5 degrees

**Examples**  
```
phase.PhaseTolerance 1 = 3 ' Write
value = phase.PhaseTolerance 2' Read
```

**C++ Syntax**  
```
HRESULT get_PhaseTolerance(long port, double* pVal);
HRESULT put_PhaseTolerance(long port, double newVal);
```

**Interface**  IPhaseControl

Last Modified: 8-Dec-2010  MX New topic
# Pin Property

**Description**
Returns the Pin result of a PNOP or PSAT marker search.

- PNOP In = Marker 4 X-axis value
- PSAT In = Marker 2 X-axis value

**VB Syntax**
```vbnet
pln = pMark.Pin
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>pln</code></td>
<td>(double) Variable to store returned value</td>
</tr>
<tr>
<td><code>pMark</code></td>
<td>A <a href="#">PNOP</a> (object) or <a href="#">PSaturation</a> (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Not applicable

**Examples**
```vbnet
pln = pMark.Pin 'Read
```

See example program

**C++ Syntax**
```cpp
HRESULT get_Pin(double * pNewVal)
```

**Interface**
IPNOP or IPSaturation

---

Last Modified:

19-Feb-2010   MX New topic
PinOffset Property

Description
Sets and returns the PinOffset value used to calculate various PNOP parameters. Also set BackOff Property.

A sweep must be executed (single or continuous) and SearchPowerNormalOperatingPoint Method must be sent before reading marker results.

To turn off the PNOP markers, either turn them off individually or DeleteAllMarkers.

To search a User Range with the PNOP search, first activate marker 1. The user range used with the PNOP search only applies to marker 1 searching for the linear gain value. The other markers may fall outside the user range.

VB Syntax
```vbnet
pnop.PinOffset = value
```

Variable
(Type) - Description

- **pnop** (A PNOP (object))
- **value** (double) - PinOffset value in dB. Choose any number between: -500 and 500

Return Type
Double

Default
0 dB

Examples
```vbnet
pinOffs = pnop.PinOffset 'Read
```

See example program

C++ Syntax
```cpp
HRESULT put_PinOffset(double newVal);
HRESULT get_PinOffset(double* pNewVal)
```

Interface
IPNOP
PMaxBackOff Property

**Description**
Sets and returns the backoff value used to calculate various PSAT parameters. A sweep must be executed (single or continuous) and SearchPowerSaturation Method must be sent before reading marker results.

To turn off the PSAT markers, either turn them off individually or DeleteAllMarkers.

To search a User Range with the PSAT search, first activate marker 1. The user range used with the PNOP search only applies to marker 1 searching for the linear gain value. The other markers may fall outside the user range.

**VB Syntax**

```
pSat.PMaxBackOff = value
```  

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pSat</td>
<td>A PSaturation (object)</td>
</tr>
<tr>
<td>value</td>
<td>(double) - Backoff value in dB. Choose any number between: -500 and 500</td>
</tr>
</tbody>
</table>

**Return Type**

Double

**Default**

0 dB

**Examples**

```
backoff = pSat.PMaxBackOff 'Read
```

**C++ Syntax**

```
HRESULT put_PMaxBackOff(double newVal);
HRESULT get_PMaxBackOff(double* pNewVal)
```

**Interface**

IPSaturation

---

Last Modified:

22-Feb-2010  MX New topic
**Description**

Returns the PMaxIn result of a PNOP and PSAT marker search.

\[ \text{PMax In} = \text{Marker 3 X-axis value} \]

**VB Syntax**

\[ p\text{MaxIn} = p\text{Mark}.\text{PMaxIn} \]

**Variable (Type) - Description**

- \( p\text{MaxIn} \) (double) - Variable to store returned value
- \( p\text{Mark} \) A PNOP (object) or PSaturation (object)

**Return Type**

Double

**Default**

Not applicable

**Examples**

\[ p\text{MaxIn} = p\text{Mark}.\text{PMaxIn} \]  
See example program

**C++ Syntax**

HRESULT get_PMaxIn(double* pNewVal)

**Interface**

IPNOP or IPSAT

---

Last Modified:

- 19-Feb-2010  MX New topic
### PMaxOut Property

**Description**
Returns the PMaxOut result of the PNOP and PSAT marker search.  
\[ \text{PMaxOut} = \text{Marker 3 Y-axis value} \]

**VB Syntax**
```vbnet
pMaxOut = pMark.PMaxOut
```

**Variable (Type) - Description**
- `pMaxOut` (double) - Variable to store returned value
- `pMark` A [PNOP](#) (object) or [PSaturation](#) (object)

**Return Type**
Double

**Default**
Not applicable

**Examples**
```vbnet
pMaxOut = pMark.PMaxOut 'Read
```

**C++ Syntax**
```cpp
HRESULT get_PMaxOut(double* pNewVal)
```

**Interface**
IPNOP or IPSaturation

---

**Last Modified:** 19-Feb-2010

MX New topic
PointAveragingState Property

**Description**
Turns point averaging ON or OFF for all measurements on the channel.

**VB Syntax**
```
chan.PointAveragingState = state
```

**Variable**
(Type) - Description

- **chan**
  A Channel (object)

- **state**
  (Enun)
  - **False** - Turns point averaging OFF
  - **True** - Turns point averaging ON

**Return Type**
Boolean

**Default**
False

**Examples**
```
chan.PointAveragingState = True 'Write
aver = chan.PointAveragingState 'Read
```

**C++ Syntax**
```
HRESULT get_PointAveragingState(BOOL *pVal)
HRESULT put_PointAveragingState(BOOL newVal)
```

**Interface**
IChannel16

---

Last Modified: 7-Oct-2008  MX New topic

---

1329
### Points Property

**Description**  
Sets group delay aperture using a fixed number of data points.

**VB Syntax**  
`gdAperture.Points = value`

**Variable**  
**Type** - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>gdAperture</code></td>
<td>A <code>GroupDelayAperture</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Number of data points to use for the aperture setting. Choose between two points and the number of points in the channel.</td>
</tr>
</tbody>
</table>

**Return Type**  
Double

**Default**  
Points range that equates to 11 points.  
This can be changed to two points with a preference setting.

**Examples**  
```
gdAperture.Points = 25 'Write
aperture = gdAperture.Points 'Read
```

**C++ Syntax**  
```cpp
HRESULT get_Points(double Points *pVal)
HRESULT put_Points(double Points newVal)
```

**Interface**  
IGroupDelayAperture

---

Last Modified:  
23-Feb-2010   MX New topic
**PointSweepState Property**

**Description**
Turns point sweep ON or OFF for all measurements on the channel. Point sweep measures both the forward and reverse parameters at each frequency point before stepping to the next frequency. The display trace is updated after the forward and reverse parameters are measured at that frequency point.

**VB Syntax**

```
chan.PointSweepState = state
```

**Variable**

- **chan**  
  A Channel (object)

- **state**  
  boolean

  - **False** - Turns point sweep OFF
  - **True** - Turns point sweep ON

**Return Type**
Boolean

**Default**
False

**Examples**

```
chan.PointSweepState = True 'Write
averg = chan.PointSweepState 'Read
```

**C++ Syntax**

```
HRESULT get_PointSweepState(VARIANT_BOOL *pVal)
HRESULT put_PointSweepState(VARIANT_BOOL newVal)
```

**Interface**
IChannel16

---

Last Modified:
3-Nov-2008  MX New topic
**Description**  
This command is replaced by PortDelay property.  
Sets a Port Extension value for Port 1

**VB Syntax**  
```
portExt.Port1 = value
```

**Variable (Type) - Description**

- **portExt**  
  A Port Extension (object)

- **value**  
  (double) - Port Extension value in seconds. Choose any number between -10 and 10

**Return Type**  
Double

**Default**  
0

**Examples**  
```
portExt.Port1 = 10e-6 'Write
prt1 = portExt.Port1 'Read
```

**C++ Syntax**
```
HRESULT get_Port1(double *pVal)
HRESULT put_Port1(double newVal)
```

**Interface**  
IPortExtension
## Port2 Property  Superseded

<table>
<thead>
<tr>
<th>Description</th>
<th>This command is replaced by <a href="#">PortDelay property</a>. Sets a Port Extension value for Port 2</th>
</tr>
</thead>
</table>

### VB Syntax

```vbnet
    portExt.Port2 = value
```

### Variable  **(Type)** - Description

- **portExt**: A Port Extension **(object)**
- **value**: **(double)** - Port Extension value in seconds. Choose any number between **-10** and **10**

### Return Type

Double

### Default

0

### Examples

```vbnet
    portExt.Port2 = 10e-6  'Write
    prt2 = portExt.Port2     'Read
```

### C++ Syntax

```cpp
    HRESULT get_Port2(double *pVal)
    HRESULT put_Port2(double newVal)
```

### Interface

IPortExtension
Port3 Property  Superseded

Description  This command is replaced by PortDelay property.
Sets a Port Extension value for Port 3

VB Syntax  
\[ \text{portExt.Port3} = \text{value} \]

Variable  
\- \textbf{(Type)} - \textbf{Description}

portExt  A Port Extension (\textbf{object})

value  (\textbf{double}) - Port Extension value in seconds. Choose any number between \textbf{-10} and \textbf{10}

Return Type  Double

Default  0

Examples  
\text{portExt.Port3} = 10e-6  \textbf{'Write}

\text{prt3} = \text{portExt.Port3}  \textbf{'Read}

C++ Syntax  
\text{HRESULT get\_Port3(double *pVal)}
\text{HRESULT put\_Port3(double new\_Val)}

Interface  IPortExtension
Port1NoiseTunerSwitchPresetsToExternal Property

Description
Sets the default setting for the Noise Tuner switch. Learn more.

VB Syntax
pref.Port1NoiseTunerSwitchPresetsToExternal = bool

Variable
(Type) - Description
pref (Preference (object))
bool (Boolean) - Choose from:
False - Sets the default (preset) to INTERNAL
True - Sets the default (preset) to EXTERNAL

Return Type
Boolean

Default
True

Examples
pref.Port1NoiseTunerSwitchPresetsToExternal = False 'Write
prefer = pref.Port1NoiseTunerSwitchPresetsToExternal 'Read

C++ Syntax
HRESULT put_Port1NoiseTunerSwitchPresetsToExternal(VARIANT_BOOL bValue)
HRESULT get_Port1NoiseTunerSwitchPresetsToExternal(VARIANT_BOOL *bValue)

Interface
IPreferences8

Last Modified:
1-Feb-2008 MX New topic
### Port2PdeembedCktModel Property

**Description**  Select whether or not to load a 2-port De-embedding circuit model for the specified port number. Circuit model is applied when both "USER" is selected and the filename is specified. To set the filename, use strPort2Pdeembed_S2PFile Property.

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**  
```vb
fixture.Port2PdeembedCktModel(port) = value
```

**Variable**  
- **fixture** *(Type)*: A Fixturing (object)
- **port** *(Integer)*: Port number to receive circuit model.
- **value** *(Enum as NAFixturing2PdeembedCkt)*:  
  0 naFix2PD_USER: load a 2-port De-embedding circuit model  
  1 naFix2PD_NONE: no model

**Return Type**  Long Integer

**Default**  naFix2PD_NONE

**Examples**  
```vb
fixture.Port2PdeembedCktModel(2) = naFix2PD_USER 'Write
value = fixture.Port2PdeembedCktModel(1) 'Read
```

**C++ Syntax**  
```c++
HRESULT get_Port2PdeembedCktModel(short port tagNAFixturing2PdeembedCkt *pVal)
HRESULT put_Port2PdeembedCktModel(short port tagNAFixturing2PdeembedCkt newVal)
```

**Interface**  IFixturing
### Port2PdeembedState Property

**Description**
Turns de-embedding ON or OFF for all ports on the channel.

**VB Syntax**

```vbnet
fixture.Port2PdeembedState = value
```

**Variable**

- **Type** - Description
- **fixture** - A Fixturing (object)
- **value** - (Boolean)
  - **False** - Turns De-embedding OFF
  - **True** - Turns De-embedding ON

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
fixture.Port2PdeembedState = False 'Write
value = fixture.Port2PdeembedState 'Read
```

**C++ Syntax**

```csharp
HRESULT get_Port2PdeembedState(VARIANT_BOOL *pVal)
HRESULT put_Port2PdeembedState(VARIANT_BOOL newVal)
```

**Interface**
IFixturing
### PortArbzImag Property

**Description**
Sets and returns the Imaginary portion of the impedance value for the specified single-ended port. Use `PortArbzReal` to set the real value. Or use `PortArbzZ0` to set both values together.

**VB Syntax**
```
fixture.PortArbzImag(portNum) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing object</td>
<td></td>
</tr>
<tr>
<td>portNum</td>
<td>Integer</td>
<td>Single-ended port number to receive impedance value.</td>
</tr>
<tr>
<td>value</td>
<td>Double</td>
<td>Real Impedance value. Choose a value between -1E18 and 1E18</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**
```
fixture.PortArbzImag(2) = 75 'Write
value = fixture.PortArbzImag(1) 'Read
```

**C++ Syntax**
```
HRESULT get_PortArbzImag( short portNum, double *pVal)
HRESULT put_PortArbzImag( short portNum, double newVal)
```

**Interface**
IFixturing3
# PortArbzReal Property

**Description** Sets and returns the Real portion of the impedance value for the specified single-ended port. Use `PortArbzImag` to set the imaginary value. Or use `PortArbzZ0` to set both values together.

**VB Syntax**

```vb
fixture.PortArbzReal(portNum) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>fixture</code></td>
<td>A <code>Fixturing</code> (object)</td>
</tr>
<tr>
<td><code>portNum</code></td>
<td>(Integer) Single-ended port number to receive impedance value.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Real Impedance value. Choose a value between 0 to 1E7</td>
</tr>
</tbody>
</table>

**Return Type**

Double

**Default**

50

**Examples**

```vb
fixture.PortArbzReal(2) = 75 'Write
value = fixture.PortArbzReal(1) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_PortArbzReal( short portNum, double *pVal)
HRESULT put_PortArbzReal( short portNum, double newVal)
```

**Interface**

`IFixturing3`
### PortArbzState Property

**Description**  
Turns Port Impedance ON or OFF for all ports on the channel.

**VB Syntax**  
```
fixture.PortArbzState = value
```

**Variable**  
(Type) - Description

- **fixture**  
  A [Fixturing](#) (object)

- **value**  
  (Boolean)
  - **False** - Turns Port Impedance OFF
  - **True** - Turns Port Impedance ON

**Return Type**  
Boolean

**Default**  
False

**Examples**  
```
fixture.PortArbzState = False 'Write

value = fixture.PortArbzState 'Read
```

**C++ Syntax**  
```
HRESULT get_PortArbzState(VARIANT_BOOL *pVal)
HRESULT put_PortArbzState(VARIANT_BOOL newVal)
```

**Interface**  
IFixturing
**Write/Read**

**PortArbzZ0 Property**

**Description**
Sets and returns the Real portion of the impedance value for the specified single-ended port. The imaginary portion is automatically set to 0.0.

To set both values separately, use `PortArbzReal` and `PortArbzImag`.

**VB Syntax**

```vb
fixture.PortArbzZ0(portNum) = value
```

**Variable**

- `fixture` *(Type)* - Description
  - A `Fixturing` *(object)*

- `portNum` *(Integer)* - Single-ended port number to receive impedance value.

- `value` *(Double)* - Impedance value. Choose a value between 0 to 1E7

**Return Type**

- `Double`

**Default**

- `50`

**Examples**

```vb
fixture.PortArbzZ0(2) = 75 'Write
value = fixture.PortArbzZ0(1) 'Read
```

**C++ Syntax**

```c++
HRESULT get_PortArbzZ0( short portNum, double *pVal)
HRESULT put_PortArbzZ0( short portNum, double newVal)
```

**Interface**

- `IFixturing3`
### PortCatalog Property

**Description**
Returns a comma-separated list of the Output port selections that are available for a given logical input port.

Read the number of input ports for the test set using `NumberOfPorts` Property.

**VB Syntax**
```vbnet
value = tset.PortCatalog(logPort)
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>value</code></td>
<td>(String) Variable to store the returned information.</td>
</tr>
<tr>
<td><code>tset</code></td>
<td>A <code>TestsetControl</code> object.</td>
</tr>
<tr>
<td><code>logPort</code></td>
<td>(Long) Logical Input port number for which to return valid output ports.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Not Applicable

**Examples**

```vbnet
value = testset1.PortCatalog 2
```

**C++ Syntax**

```cpp
HRESULT get_PortCatalog(long inputPort, BSTR *outPort);
```

**Interface**
`ITestsetControl`

---

**Last Modified:**

18-Jan-2007  Fixed argument
### PortCLogic Property

**Description**
Sets and reads the logic mode of Port C on the AUX IO connector and the Handler IO connector.

*Note:* Port C lines are connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.

**VB Syntax**
```
AuxIO.PortCLogic = value
```

**Variable**

- **AuxIO** *(object)* - A Hardware Aux I/O object

- **value** *(Enum as NaRearPanelIOLogic)* - Choose from:
  - **0 - naPositiveLogic** - The associated data line goes **HIGH** when writing a 1 to a PortC bit.
  - **1 - naNegativeLogic** - The associated data line goes **LOW** when writing a 1 to a PortC bit.

When Port C is in Output/Write mode, a change in logic causes the output lines to change state immediately. For example, Low levels change to High levels.

When Port C is in Input/Read mode, a change in logic will not cause the lines to change, but data read from Port C will reflect the change in logic.

**Return Type**
Enum

**Default**
1 - naNegativeLogic

**Examples**
```
auxIO.PortCLogic = value  'Write
value = auxIo.PortCLogic  'Read
```

**C++ Syntax**
```
HRESULT put_PortCLogic ( tagNARearPanelIOLogic Mode );
HRESULT get_PortCLogic ( tagNARearPanelIOLogic* Mode );
```

**Interface**
IHWAuxIO
PortCMode Property

**Description**
Sets and reads whether Port C is setup for writing or reading data on the AUX IO connector and the Handler IO connector.

**Note:** Port C lines are connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.

**VB Syntax**

```
AuxIO.PortCMode = value
```

**Variable**

* AuxIO *(object) - A Hardware Aux I/O object
* value *(enum as NaPortMode) - Choose from:
  0 - naInput - set the port for reading
  1 - naOutput - set the port for writing

**Return Type**

Enum as NaPortMode

**Default**

1 - naInput

**Examples**

```vbnet
auxIo.get_PortCMode = naInput 'Write
value = auxIo.get_PortCMode 'Read
```

**C++ Syntax**

```cpp
HRESULT get_PordCMode( tagNAPortMode* pMode );
HRESULT put_PordCMode( tagNAPortMode pMode );
```

**Interface**

IHWAuxIO
## PortCoupleToSystemMedia Property

**Description**  
Sets and returns the state of coupling with the system Media type.

**VB Syntax**  
`fixture.PortCoupleToSystemMedia(port)= value`

**Variable** (Type) - Description

- **fixture**  
  A `Fixture` Object

- **port**  
  (Integer) Port Number that will receive the coupling change.

- **value**  
  (Boolean) Coupling state. Choose from:
  - **True** - Media type is coupled with the system setting.
  - **False** - Media type is NOT coupled with the system setting.

**Return Type**  
Boolean

**Default**  
True

**Examples**

- `fixture.PortCoupleToSystemMedia(2)= True`

- `bool = fixture.PortCoupleToSystemMedia(1)`

**C++ Syntax**

- `HRESULT get_PortCoupleToSystemMedia(short portNum, VARIANT_BOOL *pVal);`
- `HRESULT put_PortCoupleToSystemMedia(short portNum, VARIANT_BOOL newVal);`

**Interface**  
`IFixturing4`

---

**Last Modified:**

- 6-Feb-2009
- MX New topic
PortCoupleToSystemVelocity Property

**Description**
Sets and returns the state of coupling with the system Velocity Factor.

**VB Syntax**
```
fixture.PortCoupleToSystemVelocity (port) = value
```

**Variable** *(Type) - Description*
- **fixture** A Fixturing Object
- **port** (Integer) Port Number that will receive the coupling change.
- **value** (Boolean) Coupling state. Choose from:
  - **True** - Velocity Factor is coupled with the system setting.
  - **False** - Velocity Factor is NOT coupled with the system setting.

**Return Type**
Boolean

**Default**
True

**Examples**
```
fixture.PortCoupleToSystemVelocity(2) = True

bool = fixture.PortCoupleToSystemVelocity(1)
```

**C++ Syntax**
```
HRESULT get_PortCoupleToSystemVelocity(short portNum, VARIANT_BOOL *pVal);
HRESULT put_PortCoupleToSystemVelocity(short portNum, VARIANT_BOOL newVal);
```

**Interface**
IFixturing4

---

*Last Modified:*
6-Feb-2009   MX New topic
## PortDelay Property

**Description**
Sets and returns the Port Extensions Delay value for the specified port number.

*Note:* This command affects ALL measurements on the channel. This command replaces Port 1, Port 2, Port 3 Properties.

**VB Syntax**
```
fixture.PortDelay(port) = value
```

**Variable (Type) - Description**
- `fixture` A Fixturing (object)
- `port` (Integer) Port number to receive delay value.
- `value` (Double) Delay value in seconds. Choose a value between -1E18 and 1E18.

**Return Type**
Double

**Default**
0

**Examples**
```
fixture.PortDelay(2) = .002 'Write
value = fixture.PortDelay(1) 'Read
```

**C++ Syntax**
```cpp
HRESULT get_PortDelay(short port double *pVal)
HRESULT put_PortDelay(short port double newVal)
```

**Interface**
IFixturing
PortDescription Property

Description  For each port of the ECal module that is going to be characterized, sets and reads the description of the adapters, cable, or fixture to be included in the user characterization. This description is stored with the characterization in the ECal module.

Set this description before sending Initialize or the default (empty string) will be used.

VB Syntax  
userChar.PortDescription (port)= value

Variable  (Type) - Description

userChar  An ECalUserCharacterizer Object

port  (Enum) ECal port for which description is to be set. Choose from:

1 or naECalPort_A
2 or naECalPort_B
3 or naECalPort_C
4 or naECalPort_D

value  (String) Descriptive text, limited to 24 characters maximum.

Return Type  String

Default  "" (Empty String)

Examples  
userChar.PortDescription (naECalPort_C)= "3.5 mm adapter, SN 00001"

C++ Syntax  
HRESULT get_PortDescription(NAECalPort port, *BSTR description);
HRESULT put_PortDescription(NAECalPort port, BSTR description);

Interface  IECalUserCharacterizer
PortDistance Property

**Description**  Sets and returns the port extension delay in physical length (distance).

**VB Syntax**  
```
fixture.PortDistance (port) = value
```

**Variable (Type) - Description**
- **fixture**  A Fixturing Object
- **port**  (Integer) Port Number that will receive the delay setting.
- **value**  (Double) Physical length in distance. First specify units with `PortDistanceUnit`.

**Return Type**  Double

**Default**  0

**Examples**
```
fixture.PortDistance(2) = .01 'Write
value = fixture.PortDistance(2) 'Read
```

**C++ Syntax**
```
HRESULT get_PortDistance(short portNum, double *pVal);
HRESULT put_PortDistance(short portNum, double newVal);
```

**Interface**  IFixturing4

---

Last Modified:
6-Feb-2009    MX New topic
**PortDistanceUnit Property**

**Description**
Sets and returns the units for specifying port extension delay in physical length (distance).

**VB Syntax**
`fixture.PortDistanceUnit = value`

**Variable**
(Type) - Description

- **fixture**
  A Fixturing Object

- **value**
  (enum naDistanceUnit) Choose from:
  - 0 – naDistanceUnit_Meter
  - 1 – naDistanceUnit_Feet
  - 2 – naDistanceUnit_Inch

**Return Type**
Enum

**Default**
naDistanceUnit_Meter

**Examples**

```vbnet
fixture.PortDistanceUnit(2) = naDistanceUnit_Meter 'Write
value = fixture.PortDistanceUnit(2) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_PortDistanceUnit(short portNum, tagNADistanceUnit* pVal);
HRESULT put_PortDistanceUnit(short portNum, tagNADistanceUnit newVal);
```

**Interface**
IFixturing4

---

**Last Modified:**
6-Feb-2009    MX New topic
## PortExtState Property

### Description
Turns Port Extension ON or OFF for all ports on the channel.

### VB Syntax
```
fixture.PortExtState = value
```

### Variable
- **Type:** Description

  - `fixture` A [Fixturing](#) object
  
  - `value` (Boolean)
    
    - **False** - Turns Port Extensions OFF
    - **True** - Turns Port Extensions ON

### Return Type
Boolean

### Default
False

### Examples
```
fixture.PortExtState = 0 'Write

value = fixture.PortExtState 'Read
```

### C++ Syntax
```
HRESULT get_PortExtState(VARIANT_BOOL *pVal)
HRESULT put_PortExtState(VARIANT_BOOL newVal)
```

### Interface
IFixturing
## PortExtUse1 Property

**Description**
Sets and returns the Use1 ON/OFF state for the use of the PortLoss1 and PortFreq1 values for the specified port number.

*Note:* This command affects ALL measurements on the channel.

**VB Syntax**
```
fixture.PortExtUse1(port) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>fixture</code></td>
<td>A <em>Fixturing</em> object</td>
</tr>
<tr>
<td><code>port</code></td>
<td>Port number to receive Use1 ON / OFF state. (Integer)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Boolean) False - Turns Use1 OFF True - Turns Use1 ON</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**
```
fixture.PortExtUse1(2) = False 'Write
```
```
value = fixture.PortExtUse1(1) 'Read
```

**C++ Syntax**
```
HRESULT get_PortExtUse1(short port VARIANT_BOOL *pVal)
HRESULT put_PortExtUse1(short port VARIANT_BOOL newVal)
```

**Interface**
`IFixturing`
PortExtUse2 Property

**Description**
Sets and returns the Use2 ON/OFF state for the use of the PortLoss2 and PortFreq2 values for the specified port number.

*Note:* This command affects ALL measurements on the channel.

**VB Syntax**
```
fixture.PortExtUse2(port) = value
```

**Variable - Description**

- **fixture**  A **Fixturing** (object)
- **port** (Integer) Port number to receive Use2 ON / OFF state.
- **value** (Boolean)
  - False - Turns Use1 OFF
  - True - Turns Use1 ON

**Return Type**
Boolean

**Default**
False

**Examples**
```
fixture.PortExtUse2(2) = False 'Write
value = fixture.PortExtUse2(1) 'Read
```

**C++ Syntax**
```
HRESULT get_PortExtUse2(short port VARIANT_BOOL *pVal)
HRESULT put_PortExtUse2(short port VARIANT_BOOL newVal)
```

**Interface**
IFixturing
**PortFreq1 Property**

**Description**
Sets and returns Frequency1 value for the specified port number.

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**
```vbnet
fixture.PortFreq1(port) = value
```

**Variable**
- **(Type) - Description**
  - `fixture` A **Fixturing** (object)
  - `port` (Integer) Port number to receive extrapolated loss.
  - `value` (Double) Frequency1 value. Choose a frequency within the frequency span of the PNA.

**Return Type**
Double

**Default**
1 GHz

**Examples**
```vbnet
fixture.PortFreq1(2) = naFix2PD_USER 'Write
value = fixture.PortFreq1(1) 'Read
```

**C++ Syntax**
```c++
HRESULT get_PortFreq1(short port double *pVal)
HRESULT put_PortFreq1(short port double newVal)
```

**Interface**
IFixturing
## PortFreq2 Property

**Description**
Sets and returns Frequency2 value for the specified port number.

*Note: This command affects ALL measurements on the channel.*

**VB Syntax**

```vbnet
fixture.PortFreq2(port) = value
```

**Variable**

- **fixture** (*Fixturing* object)
- **port** (*Integer*) Port number to receive extrapolated loss.
- **value** (*Double*) Frequency2 value. Choose a frequency within the frequency span of the PNA.

**Return Type**
Double

**Default**
1 GHz

**Examples**

```vbnet
fixture.PortFreq2(2) = 10E9 'Write
value = fixture.PortFreq2(1) 'Read
```

### C++ Syntax

- HRESULT get_PortFreq2(short port double *pVal)
- HRESULT put_PortFreq2(short port double newVal)

**Interface**
IFixturing
### PortLabel Property

**Description**
Sets and returns the label on the calibration kit Port for the calibration wizard.

**VB Syntax**

```vbnet
calKit.PortLabel (portNum) = value
```

**Variable**

- **Type** - Description
  - **calKit** - A CalKit (object)
  - **portNum** - (long integer) - number of the port to be labeled. Choose either 1 or 2
  - **value** - (string) - Label that is visible in the calibration wizard.

**Return Type**
String

**Default**
Depends on the Cal Kit.

**Examples**

```vbnet
calKit.PortLabel = "MyCalKit" 'Write
kitLabel = calKit.PortLabel 'Read
```

**C++ Syntax**

```
HRESULT get_PortLabel(long port, BSTR *pVal)
HRESULT put_PortLabel(long port, BSTR newVal)
```

**Interface**
ICalKit
# PortLogic Property

**Description**
Sets and returns the logic mode of data ports A-H on the Handler I/O connector. Port C of the Handler I/O is connected internally to the Port C of the Aux I/O connector. Therefore, it will have the same logic mode.

**VB Syntax**
```
handler.PortLogic = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>handler</td>
<td>(object) - A Handler I/O object</td>
</tr>
<tr>
<td>value</td>
<td>(enum as NaRearPanelIOLogic) - Choose from:</td>
</tr>
<tr>
<td>0</td>
<td>naPositiveLogic - When a value of one is written, the associated line goes High</td>
</tr>
<tr>
<td>1</td>
<td>naNegativeLogic - When a value of one is written, the associated line goes Low</td>
</tr>
</tbody>
</table>

For ports that are in output (write) mode, a change in logic causes the output lines to change state immediately. For example, Low levels change immediately to High levels.

For ports that are in input (read) mode (C,D,E only), a change in logic will be reflected when data is read from that port. For example, if a line read 0, the next read after a logic change will read 1.

**Return Type**
Long Integer

**Default**
1 - naNegativeLogic

**Examples**
```
handler.PortLogic = value 'Write
value = handler.PortLogic 'Read
```

**C++ Syntax**
```
HRESULT put_PortLogic( tagNARearPanelIOLogic Mode );
HRESULT get_PortLogic( tagNARearPanelIOLogic* Mode );
```

**Interface**
IHWMaterialHandlerIO
**PortLoss1 Property**

**Description**
Sets and returns the Loss1 value for the specified port number.

*Note:* This command affects ALL measurements on the channel.

**VB Syntax**
`fixture.PortLoss1(port) = value`

**Variable**
- **(fixture)** - A Fixturing (object)
- **(port)** - Port number to receive Loss value
- **(value)** - Loss1 value in dB. Choose a value between -90 and 90.

**Return Type**
Double

**Default**
0

**Examples**
`fixture.PortLoss1(2) = .002 'Write`

`value = fixture.PortLoss1(1) 'Read`

**C++ Syntax**

- `HRESULT get_PortLoss1(short port double *pVal)`
- `HRESULT put_PortLoss1(short port double newVal)`

**Interface**
IFxturing
## PortLoss2 Property

**Description**  
Sets and returns the Loss2 value for the specified port number.  

*Note:* This command affects ALL measurements on the channel.

**VB Syntax**  
```vbnet  
fixture.PortLoss2(port) = value  
```

**Variable**  
*Type* - Description  

- `fixture`  
  - **Type:** A `Fixturing` object  
- `port`  
  - **Type:** Integer  
  - Port number to receive Loss value  
- `value`  
  - **Type:** Double  
  - Loss2 value in dB. Choose a value between -90 and 90.

**Return Type**  
Double

**Default**  
0

**Examples**  
```vbnet  
fixture.PortLoss2(2) = .002 'Write  
value = fixture.PortLoss2(1) 'Read  
```

**C++ Syntax**  
```c++  
HRESULT get_PortLoss2(short port double *pVal)  
HRESULT put_PortLoss2(short port double newVal)  
```

**Interface**  
IFixturing
PortLossDC Property

Description
Sets and returns the Loss value at DC for the specified port number.

Learn about Loss compensation values.

Note: This command affects ALL measurements on the channel.

VB Syntax
```
fixture.PortLossDC(port) = value
```

Variable (Type) - Description

| fixture  | A Fixturing (object) |
| port     | (Integer) Port number to receive Loss value at DC. |
| value    | (Double) Loss value in ohms. Choose a value between -90 and 90 |

Return Type
Double

Default
0

Examples
```
fixture.PortLossDC(2) = .002 'Write

value = fixture.PortLossDC(1) 'Read
```

C++ Syntax
```
HRESULT get_PortLossDC(short port double *pVal)
HRESULT put_PortLossDC(short port double newVal)
```

Interface
IFixturing
Write/Read

PortMatching_C Property

Description
Sets and returns the Capacitance value for the specified port number.

Note: This command affects ALL measurements on the channel.

VB Syntax

`fixture.PortMatching_C(port) = value`

Variable

(Type) - Description

*fixture* A Fixturing (object)

*port* (Integer) Port number to receive capacitance value

*value* (Double) Capacitance value in farads. Choose a value between -1E18 to 1E18.

Return Type
Double

Default
0

Examples

`fixture.PortMatching_C(2) = .00002 'Write`

`value = fixture.PortMatching_C(1) 'Read`

C++ Syntax

HRESULT get_PortMatching_C(short port double *pVal)

HRESULT put_PortMatching_C(short port double newVal)

Interface
IFixturing
PortMatching_G Property

Description
Sets and returns the Conductance value for the specified port number.

Note: This command affects ALL measurements on the channel.

VB Syntax
   fixture.PortMatching_G(port) = value

Variable (Type) - Description

   fixture (A Fixturing (object))
   port (an Integer) Port number to receive conductance value.
   value (a Double) Conductance value in siemens. Choose a value between -1E18 and 1E18.

Return Type
Double

Default
0

Examples
   fixture.PortMatching_G = .002 'Write
   value = fixture.PortMatching_G 'Read

C++ Syntax
   HRESULT get_PortMatching_G(short port double *pVal)
   HRESULT put_PortMatching_G(short port double newVal)

Interface
IFixturing
PortMatching_L Property

Description
Sets and returns the Inductance value for the specified port number.

Note: This command affects ALL measurements on the channel.

VB Syntax
fixture.PortMatching_L(port) = value

Variable (Type) - Description
fixture A Fixturing (object)
port (Integer) Port number to receive inductance value
value (Double) Inductance value in henries. Choose a value between -1E18 and 1E18.

Return Type
Double

Default
0

Examples
fixture.PortMatching_L = .002 'Write
value = fixture.PortMatching_L 'Read

C++ Syntax
HRESULT get_PortMatching_L(short port double *pVal)
HRESULT put_PortMatching_L(short port double newVal)

Interface
IFixturing
## PortMatching_R Property

### Description
Sets and returns the Resistance value for the specified port number.

**Note:** This command affects ALL measurements on the channel.

### VB Syntax
`fixture.PortMatching_R(port) = value`

### Variable

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>fixture</code></td>
<td>Fixturing (object)</td>
<td></td>
</tr>
<tr>
<td><code>port</code></td>
<td>Integer</td>
<td>Port number to receive resistance value.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>Double</td>
<td>Resistance value in ohms. Choose a value between -1E18 and 1E18.</td>
</tr>
</tbody>
</table>

### Return Type
Double

### Default
0

### Examples
```vbnet
fixture.PortMatching_R = .1 'Write
value = fixture.PortMatching_R 'Read
```

### C++ Syntax

```cpp
HRESULT get_PortMatching_R(short port double *pVal)
HRESULT put_PortMatching_R(short port double newVal)
```

### Interface
`IFixturing`
**PortMatchingCktModel Property**

**Description**
Sets and returns the Port Matching circuit model for the specified port number.

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**
```vbnet
fixture.PortMatchingCktModel(port) = value
```

**Variable**
- **Type** - Description
  - `fixture` A **Fixturing** (object)
  - `port` (Integer) Port number to receive circuit model.
  - `value` (Enum as **NAFixturingPortMatchCkt**) Circuit model. Choose from
    - 0 `naFixPMC_SLPC` Series L - Parallel C
    - 1 `naFixPMC_PCSL` Parallel C - Series L
    - 2 `naFixPMC_PLSC` Parallel L - Series C
    - 3 `naFixPMC_SCPL` Series C - Parallel L
    - 4 `naFixPMC_PLPC` Parallel L - Parallel C
    - 5 `naFixPMC_USER` Load S2P file - also set filename to load with **strPortMatch_S2PFile Property**
    - 6 `naFixPMC_NONE` No circuit model

**Return Type**
Long Integer

**Default**
`naFixPMC_NONE`

**Examples**
```vbnet
fixture.PortMatchingCktModel(2) = naFixPMC_PLSC 'Write
value = fixture.PortMatchingCktModel(1) 'Read
```

**C++ Syntax**
```c
HRESULT get_PortMatchingCktModel(short port tagNAFixturingPortMatchCkt *pVal)
HRESULT put_PortMatchingCktModel(short port  tagNAFixturingPortMatchCkt newVal)
```

**Interface**
IFixturing
PortMatchingState Property

Description
Sets and returns the Port Matching State on the channel.

VB Syntax
```
fixture.PortMatchingState = value
```

Variable (Type) - Description

- `fixture` A Fixturing (object)
- `value` (boolean)
  - `True` - Turns Port Matching ON
  - `False` - Turns Port Matching OFF

Return Type
Boolean

Default
False

Examples
```
fixture.PortMatchingState = True 'Write
value = fixture.PortMatchingState 'Read
```

C++ Syntax
```
HRESULT get_PortMatchingState(VARIANT_BOOL *pVal)
HRESULT put_PortMatchingState(VARIANT_BOOL newVal)
```

Interface
IFixturing
PortMedium Property

**Description**  Sets and returns the media type of the added fixture or transmission line.

**VB Syntax**  

```vbnet
fixture.PortMedium (port)= value
```

**Variable**  

- **Type**  - Description

  - **fixture**  - A Fixturing Object
  - **port**  - (Integer) Port Number for which media type is being set.
  - **value**  - (enum NACalStandardMedium) - Medium of the transmission line of the standard. Choose from:
    - 0 - naCoax - Coaxial Cable
    - 1 - naWaveGuide - Waveguide

**Return Type**  Enum

**Default**  0 - naCoax

**Examples**  

```vbnet
fixture.PortMedium(2)= naCoax 'Write
value = fixture.PortMedium(2) 'Read
```

**C++ Syntax**  

```cpp
HRESULT get_PortMedium(short portNum, tagNACalStandardMedium* pVal);
HRESULT put_PortMedium(short portNum, tagNACalStandardMedium newVal);
```

**Interface**  IFixturing4

---

Last Modified:

6-Feb-2009    MX New topic
PortMode Property

Description
Sets and returns whether Port C or Port D is used for writing or reading data on the Handler IO connector. The Handler IO Port C is connected internally to the Port C of the Aux IO connector. Therefore, the Aux IO connector will have the same input/output mode.

VB Syntax
`handler.PortMode (port) = value`

Variable (Type) - Description
- `handler` (object) - A Handler I/O object
- `port` (enum as NAMatHandlerPort) Port to be changed. Choose from:
  - 2 - naPortC
  - 3- naPortD
- `value` (enum as NaPortMode) - Choose from:
  - 0 - naInput - set the port for reading
  - 1 - naOutput - set the port for writing

Return Type
Long Integer

Default
1 - naInput

Examples
```vbnet
handler.PortMode(naPortC) = naInput 'Write
value = handler.PortMode(naPortD) 'Read
```

C++ Syntax
```cpp
HRESULT put_PortMode ( tagNAMatHandlerPort Port, tagNAPortMode Mode );
HRESULT get_PortMode ( tagNAMatHandlerPort Port, tagNAPortMode* Mode );
```

Interface
IHWMaterialHandlerIO
PortsNeedingDeltaMatch Property

Description
Returns the port numbers for which delta match correction is required. 0 (zero) is returned if the Cal does NOT require Delta Match correction for one of the following reasons:

- The Cal does NOT involve Unknown Thru or TRL. You specify this using ThruCalMethod Property.
- The Cal DOES involve Unknown Thru or TRL, but the delta match data can be calculated by the Unknown Thru or TRL Cal. Learn how this is possible. However, you can force the Cal to use the Delta Match data from a Cal Set.

VB Syntax
value = guided.PortsNeedingDeltaMatch

Variable (Type) - Description
value (Variant) Variable to store the returned list of port numbers.

guided GuidedCalibration (object)

Return Type Variant

Default Not Applicable

Examples
Dim ports As Variant
ports = guided.PortsNeedingDeltaMatch

C++ Syntax
HRESULT get_PortsNeedingDeltaMatch (VARIANT* portList);

Interface IGuidedCalibration2
## PortVelocityFactor Property

**Description** Sets and returns the Port Extensions Velocity Factor value for the specified port number.  

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**

```
fixture.PortVelocityFactor(port) = value
```

**Variable**

- **(Type) - Description**
  - `fixture` A Fixturing (object)
  - `port` (Integer) Port number to receive velocity factor value.
  - `value` (Double) Velocity Factor value. Choose a number between: 0 and 10  
  
  - (.66 polyethylene dielectric; .7 PTFE dielectric)

**Return Type** Double

**Default** 1

**Examples**

```
fixture.PortVelocityFactor(2) = .6 'Write
value = fixture.PortVelocityFactor(1) 'Read
```

**C++ Syntax**

```
HRESULT get_PorVelocityFactor(short port double *pVal)
HRESULT put_PorVelocityFactor(short port double newVal)
```

**Interface** IFixturing4

---

**Last Modified:**  
12-Feb-2009   MX New topic
Write / Read

About Port Extensions

PortWGCT cutoffFreqProperty

Description
Sets and returns the cutoff (minimum) frequency of the added waveguide fixture or transmission line.

VB Syntax

`fixture.PortWGCT cutoffFreq(port) = value`

Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing Object</td>
<td></td>
</tr>
<tr>
<td>port</td>
<td>(Integer) Port Number for which media type is being set.</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>(Double) Cutoff frequency in Hz.</td>
<td></td>
</tr>
</tbody>
</table>

This value is ignored when PortMedium Property is set to COAX for the same port.

Return Type
Double

Default
System Media Cutoff Frequency

Examples

`fixture.PortWGCT cutoffFreq(2) = 1e9`  'Write

`value = fixture.PortWGCT cutoffFreq(2)`  'Read

C++ Syntax

`HRESULT get_PortWGCT cutoffFreq(short portNum, double *pVal);`
`HRESULT put_PortWGCT cutoffFreq(short portNum, double newVal);`

Interface IFixturing4

Last Modified:

6-Feb-2009  MX New topic
### POut Property

**Description**
Returns the POut result of the PNOP or PSAT marker search.

- PSAT Out = Marker 2 Y-axis value
- PNOPOut = Marker 4 Y-axis value

**VB Syntax**

```vb
pOut = pMark.POut
```

**Variable**

- **Type**
- **Description**
  - `pOut` (double) - Variable to store returned value
  - `pMark` - A PNOP (Object) or PSaturation (Object)

**Return Type**
Double

**Default**
Not applicable

**Examples**

```
'Read
pOut = pMark.POut

See example program
```

**C++ Syntax**

```cpp
HRESULT get_POut(double* pNewVal)
```

**Interface**
IPNOP or IPSaturation

---

Last Modified:

- 19-Feb-2010
- MX New topic
## PowerSlope Property

**Description**
Sets or returns the Power Slope value. Power Slope function increases or decreases the output power over frequency. Units are db/GHz. For example: PowerSlope = 2 will increase the power 2db/1GHz.

**VB Syntax**

```
object.PowerSlope = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>object</code></td>
<td>Channel (object)</td>
</tr>
<tr>
<td></td>
<td>or</td>
</tr>
<tr>
<td></td>
<td>CalSet (object) - Read-only property</td>
</tr>
</tbody>
</table>

**Value**

(double) - Power Slope. Choose any number between -2 and 2. 
No slope = 0

**Return Type**

Double

**Default**

0

**Examples**

```
chan.PowerSlope = 2  'Write
pwrslp = chan.PowerSlope  'Read
```

**C++ Syntax**

```
HRESULT get_PowerSlope(double *pVal)
HRESULT put_PowerSlope(double newVal)
```

**Interface**

IChannel
CalSet3
PowerAcquisitionDevice Property

**Description**
Returns the power sensor channel (A or B) that is currently selected for use at a specific frequency.

If **UsePowerSensorFrequencyLimits** is set to False, this property will return the sensor channel last used for a source power calibration. This setting corresponds to the **Use this sensor only** checkbox in the **Power Sensor Settings** dialog.

When performing an SMC calibration, use **SetPowerAcquisitionDevice Method** to set the power sensor channel.

**VB Syntax**
```
sensor = pwrCal.PowerAcquisitionDevice(dFreq)
```

**Variable** *(Type)* - Description

**sensor** *(enum NAPowerAcquisitionDevice)* The currently selected sensor channel for the specified frequency. Choose from:
- 0 – naPowerSensor_A
- 1 – naPowerSensor_B

**pwrCal** A SourcePowerCalibrator (object)

**dFreq** (double) Frequency (Hz) for the power reading of interest.

**Return Type** enum NAPowerAcquisitionDevice

**Default** Not Applicable

**Examples**
```
selectedSensor = pwrCal.PowerAcquisitionDevice(1.E9) 'Write
```

**C++ Syntax**
```
HRESULT get_PowerAcquisitionDevice(double dFreq, tagNAPowerAcquisitionDevice* enumAcqDevice);
```

**Interface** ISourcePowerCalibrator2
**PowerAsFixture Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and reads the state of power offset as a fixture with True Mode balanced measurements. <a href="#">Learn more about iTMSA power and power offset.</a></th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>balStim.PowerAsFixture = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>balStim</code></td>
<td>A <code>BalancedStimulus</code> (object)</td>
</tr>
</tbody>
</table>
| `value`     | **(Boolean)** State of power offset as a fixture.  
**False** Offset is applied but is NOT included as a fixture in the output calculations.  
**True** Offset is applied and included as a fixture in the output calculations. |
| Return Type | Boolean                                                                                                                         |
| Default     | False                                                                                                                          |
| Examples    | `balStim.PowerAsFixture = False` **Write**                                                                                      |
|             | `var = balStim.PowerAsFixture` **Read**                                                                                         |
| C++ Syntax  | HRESULT `get_PowerAsFixture (VARIANT_BOOL *bVal)`  
HRESULT `put_PowerAsFixture (VARIANT_BOOL bVal)`                                                                 |
| Interface   | IBalancedStimulus                                                                                                               |

Last Modified:

27-May-2008   MX New topic
**PowerCalibrationLevel Property**

**Description**  
Set and read the power level at which to perform the source cal during an Enhanced Power Cal or during the power cal portion of some Noise Figure Cals.

**VB Syntax**  
`object.PowerCalibrationLevel (port) = value`

**Variable (Type) - Description**

*object A GuidedCalibration (Object)*

*port (Long) PNA Port number to connect the power sensor.*

*value (Double) - Power level in dB. Choose a value from +30 to (-30).*

**Return Type**  
Double

**Default**  
0

**Examples**

```vbnet
cal.PowerCalibrationLevel(1) = -5 'Write
pLevel = nfx.PowerCalibrationLevel(1) 'Read
```

See enhanced power cal example

**C++ Syntax**

```c++
HRESULT get_PowerCalibrationLevel(long port, double* pVal)
HRESULT put_PowerCalibrationLevel(long port, double* pVal)
```

**Interface**

IGuidedCalibration6

Last Modified:

- 4-Oct-2010  Added enhanced power cal
- 12-Oct-2009  MX New topic
**PowerCalibrationPowerLevel Property**

**Description**
Set and read the power level at which to perform the source cal during an Guided Power Cal or during an NFX Cal.

When used with Guided Power Cal, first enable a power cal using PerformPowerCalibration Property.

**VB Syntax**
```
object.PowerCalibrationPowerLevel (port) = value
```

**Variable (Type) - Description**
- **object** (GuidedCalibration (Object))
- **port** (Long) - PNA Port number to connect the power sensor.
- **value** (Double) - Power level in dB. Choose a value from +30 to (-30).

**Return Type**
Double

**Default**
0

**Examples**
```
cal.PowerCalibrationPowerLevel(1) = -5 'Write
pLevel = nfx.PowerCalibrationPowerLevel(1) 'Read
```

**C++ Syntax**
```
HRESULT get_PowerCalibrationPowerLevel(long port, double* pVal)
HRESULT put_PowerCalibrationPowerLevel(long port, double* pVal)
```

**Interface**
IGuidedCalibration6

---

**Last Modified:**
12-Oct-2009  MX New topic
PowerLevel Property

**Description**
Set and read the power level at which to perform the Source Power Cal portion of a Gain Compression Calibration or a SweptIMD Cal.

**VB Syntax**

```
object.PowerLevel = value
```

**Variable**

*object* - Description

object A GainCompressionCal (object)

A SweptIMDCal (object)

*value* - (Double) - Power level in dB. Choose a value from +30 to (-30). [Learn about choosing a power level](#).

**Return Type**

Double

**Default**

0

**Examples**

```
gca.PowerLevel = -5 'Write
```

```
pLevel = imd.PowerLevel 'Read
```

**C++ Syntax**

```cpp
HRESULT get_PowerLevel(double* pVal)
HRESULT put_PowerLevel(double newVal)
```

**Interface**

IGainCompressionCal

ISweptIMDCal

---

**Last Modified:**

- 9-Sep-2008 Added IMD
- 11-Apr-2008 MX New topic
PowerMax Property

**Description**
Sets and returns the maximum power level for Safe Mode.

**VB Syntax**
```
RxLevel.PowerMax(srcPort) = value
```

**Variable**
- **RxLevel**: A `ReceiverLeveling` Object
- **srcPort**: (Long Integer) Source port for which to set the max power for Receiver Leveling.
- **value**: (Double) Maximum power level in dB.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

**Return Type**
(Double)

**Default**
+ 30

**Examples**
```
rxLevel.PowerMax (1) = 25 ' Write

value = rxLevel.PowerMax 2' Read
```

**C++ Syntax**
```
HRESULT get_PowerMax(long port, double* pVal);
HRESULT put_PowerMax(long port, double newVal);
```

**Interface**
`IReceiverLevelingConfiguration`

---

Last Modified:

13-Feb-2009    MX New topic
PowerMaxIn
PowerMaxOut
PowerMeterChannel Property

Description  Identifies which channel of the power meter the power sensor is connected to.

VB Syntax  

\[
\text{chan} = \text{powerSensor}.\text{PowerMeterChannel}
\]

Variable  (Type) - Description

\[
\text{chan} \quad \text{(enum NAPowerAcquisitionDevice)}
\]

– Power meter channel identifier for sensor.

Choose from:

\[
\begin{align*}
0 & \quad \text{naPowerSensor}_A \\
1 & \quad \text{naPowerSensor}_B
\end{align*}
\]

\[
\text{pwrSensor} \quad \text{A PowerSensor (Object) or A PowerSensorAsReceiver (Object)}
\]

Return Type  NAPowerAcquisitionDevice

Default  Not Applicable

Examples  

\[
\text{meterChannel} = \text{pwrSensor}.\text{PowerMeterChannel}
\]

C++ Syntax  

\[
\text{HRESULT PowerMeterChannel(tagNAPowerAcquisitionDevice *pSensor)};
\]

Interface  

IPowerSensor

IPowerSensorAsReceiver

Last Modified:

25-Aug-2009  Added PMAR
PowerMeterGPIBAddress Property  Superseded

Description  This command is replaced with PowerMeterInterface Object. Specifies the GPIB address of the power meter that will be referenced by the SourcePowerCalibrator object.

When performing a source power cal, the PNA will search VISA interfaces that are configured in the Agilent IO Libraries on the PNA.

VB Syntax

`powerCalibrator.PowerMeterGPIBAddress = value`

Variable (Type) - Description

`powerCalibrator (object) - A SourcePowerCalibrator (object)`

`value (long integer) – Power meter GPIB address. Choose any number between 0 and 30.`

Return Type  Long integer

Default  13

Examples

Set `powerCalibrator = pna.SourcePowerCalibrator`
`powerCalibrator.PowerMeterGPIBAddress = 13` 'Write

`pwrMtrAddress = powerCalibrator.PowerMeterGPIBAddress` 'Read

C++ Syntax

HRESULT put_PowerMeterGPIBAddress(long newVal);
HRESULT get_PowerMeterGPIBAddress(long *pVal);

Interface  ISourcePowerCalibrator

Last Modified:

9-Jul-2007  Superseded
PowerMin Property

**Description**
Sets and returns the minimum power level for Safe Mode.

**VB Syntax**
```
RxLevel.PowerMin(srcPort) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RxLevel</td>
<td>A ReceiverLeveling Object</td>
</tr>
<tr>
<td>srcPort</td>
<td>(Long Integer) Source port for which to set the min power for Receiver Leveling.</td>
</tr>
</tbody>
</table>

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Double) Minimum power level in dB.</td>
</tr>
</tbody>
</table>

**Return Type**
(Double)

**Default**
-95 dB

**Examples**
```
rxLevel.PowerMin (1) = -50 ' Write
value = rxLevel.PowerMin 2' Read
```

**C++ Syntax**
```
HRESULT get_PowerMin(long port, double* pVal);
HRESULT put_PowerMin(long port, double newVal);
```

**Interface**
IReceiverLevelingConfiguration

---

Last Modified:

13-Feb-2009   MX New topic
### PowerOffset Property

**Description**
Sets and returns the power level offset value.

**VB Syntax**
```
RxLevel.PowerOffset(srcPort) = value
```

**Variable (Type) - Description**

- **RxLevel**: A `ReceiverLeveling` Object
- **srcPort**: (Long Integer) Source port for which to set the offset power for Receiver Leveling.
  - **Note**: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).
- **value**: (Double) Power level Offset in dB.

**Return Type**
(Double)

**Default**
0 dB

**Examples**
```
rxLevel.PowerOffset (1) = 10 ' Write
value = rxLevel.PowerOffset 2' Read
```

**C++ Syntax**
```
HRESULT get_PowerOffset(long port, double* pVal);
HRESULT put_PowerOffset(long port, double newVal);
```

**Interface**
`IReceiverLevelingConfiguration`

---

Last Modified:
13-Feb-2009   MX New topic
# PowerOnDuringRetraceMode Property

## Description
For single-band frequency or segment sweeps ONLY, specify whether to turn RF power ON or OFF during a retrace.

This setting remains until changed using this command, or until the hard drive is changed or reformatted.

## VB Syntax
```
pref.PowerOnDuringRetraceMode = value
```

## Variable
- **Type**: A Preferences (object)
- **Description**:

## Value
- **Type**: Enum
- **Choose from**:
  - **0 - naRetracePowerMode_Auto**: Power is left ON during retrace of single-band frequency or segment sweeps ONLY.
  - **1 - naRetracePowerMode_OFF**: Power is turned OFF during retrace of single-band frequency or segment sweeps ONLY.

## Return Type
- **Type**: Enum
- **Default**: 0 - naRetracePowerMode_Auto

## Examples
```
pref.PowerOnDuringRetraceMode = naPowerSweepRetraceMode_Start 'Write
psMode = pref.naPowerOnDuringRetraceMode_Start 'Read
```

## C++ Syntax
```
HRESULT get_PowerOnDuringRetraceMode (tagNARetracePowerMode* preference);
HRESULT put_PowerOnDuringRetraceMode (tagNARetracePowerMode val)
```

## Interface
- **IPreferences4**

---

Last modified:
- **Nov. 16, 2006**
- **New command**

### PowerSensorCalKitType Property

**Description**
Set and read the cal kit to be used for calibrating at the reference plane when the power sensor connector is different from the DUT port.

When used with Guided Power Cal, first enable a power cal using `PerformPowerCalibration` Property.

**VB Syntax**

```vbnet
object.PowerSensorCalKitType (n) = value
```

**Variable**

- **(Type)** - Description

  - **object**
    - A GuidedCalibration (object)
  - **n** (Long) PNA port number for which cal kit is specified.
  - **value** (String) - Cal Kit. Use `CompatibleCalKits Property` to return a list of valid cal kits.

**Return Type**

String

**Default**

Not Applicable

**Examples**

```vbnet
gguided.PowerSensorCalKitType(1) = "85052B"  'Write
cType = guided.PowerSensorCalKitType(1)  'Read
```

**C++ Syntax**

```cpp
HRESULT get_PowerSensorCalkitType(long port, BSTR* Val)
HRESULT put_PowerSensorCalkitType(long port, BSTR  newVal)
```

**Interface**

IGuidedCalibration6

---

_Last Modified:

- 30-Nov-2010  Fixed example
- 12-Oct-2009  MX New topic_
PowerSensorCalkitType Property

Description: Set and read the cal kit to be used for calibrating at the adapter when the power sensor connector is different from the DUT. Use PowerSensorConnectorType to specify the connector type of the adapter.

VB Syntax: `object.PowerSensorCalkitType = value`

Variable (Type) - Description:

- `object`: A GainCompressionCal (object)
- A SweptIMD (object)
- A GuidedCalibrationPowerSensor (object)

Value (String) - Cal Kit. Use CompatibleCalKits Property to return a list of valid cal kits.

Return Type: String

Default: Not Applicable

Examples:
```
gca.PowerSensorCalkitType = "85052B" 'Write
ctype = imd.PowerSensorCalkitType 'Read
```

C++ Syntax:
```
HRESULT get_PowerSensorCalkitType(BSTR* Val)
HRESULT put_PowerSensorCalkitType(BSTR newVal)
```

Interface: IGainCompressionCal2
- ISweptIMD
- IGuidedCalibrationPowerSensor

Last Modified:
- 8-Feb-2011 Added Guided
- 9-Sep-2008 Added Swept IMD
- 11-Apr-2008 MX New topic
PowerSensorConnectorType Property

**Description**
Set and read the power sensor connector type which is used to perform the Source Power Cal. Use **PowerSensorCalKitType** to specify the Cal Kit to use for the cal.

**VB Syntax**

```vbnet
object.PowerSensorConnectorType = value
```

**Variable**

- **(Type)** - Description

  - **object.** A **GainCompressionCal** *(object)*
  - A **SweptIMDCal** *(object)*
  - A **GuidedCalibrationPowerSensor** *(object)*

- **(String)** - Power sensor connector type. Use **ValidConnectorType Property** to return a list of valid connector types.
  - Select "Ignored" to NOT compensate for the adapter.

**Return Type**

- **String**

**Default**

- Not applicable

**Examples**

```vbnet
(gca.PowerSensorConnectorType = "APC3.5 male" 'Write
ctype = imd.PowerSensorConnectorType 'Read
```

**C++ Syntax**

- HRESULT get_PowerSensorConnectorType(BSTR* Val)
- HRESULT put_PowerSensorConnectorType(BSTR newVal)

**Interface**

- IGainCompressionCal2
- ISweptIMDCal
- IGuidedPowerCalPowerSensor

**Last Modified:**

- 8-Feb-2011   Added Guided
- 9-Sep-2008   Added Swept IMD
- 11-Apr-2008  MX New topic
### PowerSensorConnectorType Property

**Description**
Set and read the power sensor connector type which is used to perform a Power Cal during an S-parameter calibration or during an NFX Cal.

When used with [Guided Power Cal](About Guided Power Cal), first enable a power cal using [PerformPowerCalibration Property](About Guided Power Cal).

**VB Syntax**
```
guided.PowerSensorConnectorType (n) = value
```

**Variable**

<table>
<thead>
<tr>
<th><strong>(Type)</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>guided</code></td>
<td>A <a href="#">GuidedCalibration</a> (object)</td>
</tr>
<tr>
<td><code>n</code></td>
<td>(Long) PNA port number to connect power sensor to.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(String) - Power sensor connector type. Use [ValidConnectorType Property](About Guided Power Cal) to return a list of valid connector types. Set to &quot;Ignored&quot; to NOT compensate for the adapter.</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Not applicable.

**Examples**
```
guided.PowerSensorConnectorType(1) = "APC3.5 male" 'Write
ctype = guided.PowerSensorConnectorType(1) 'Read
```

**C++ Syntax**
```
HRESULT get_PowerSensorConnectorType (long port, BSTR* val);
HRESULT put_PowerSensorConnectorType (long port, BSTR newVal);
```

**Interface**
IGuidedCalibration6

---

**Last Modified:**
30-Nov-2010 Added index to example
12-Oct-2009 MX New topic
**PowerStep Property**

**Description**
Sets and returns the safe mode power step value.

**VB Syntax**
\[ RxLevel.PowerStep(srcPort) = value \]

**Variable**
*(Type)* - Description

- **RxLevel**
  A ReceiverLeveling Object

- **srcPort**
  (Long Integer) Source port for which to set the power step for Receiver Leveling.

  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**
  (Double) Power step in dB.

**Return Type**
(Double)

**Default**
1 dB

**Examples**

```
rxLevel.PowerStep (1) = 2 ' Write

value = rxLevel.PowerStep 2 ' Read
```

**C++ Syntax**

```cpp
HRESULT get_PowerStep(long port, double* pVal);
HRESULT put_PowerStep(long port, double newVal);
```

**Interface**
IRxReceiverLevelingConfiguration

---

Last Modified:

13-Feb-2009   MX New topic
# PowerSweepRetracePowerMode Property

**Description**  
At the end of a power sweep, while waiting to trigger the next sweep, maintain source power at either the start power level or at the stop power level.

**VB Syntax**  
`pref.PowerSweepRetracePowerMode = value`

**Variable**  
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>pref</code></td>
<td>A Preferences (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Enum) - Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - <code>naPowerSweepRetraceMode_Start</code> - maintain source at start power level.</td>
</tr>
<tr>
<td></td>
<td>1 - <code>naPowerSweepRetraceMode_Stop</code> - maintain source at stop power level.</td>
</tr>
</tbody>
</table>

**Return Type**  
Enum

**Default**  
0 - `naPowerSweepRetraceMode_Start`

**Examples**  
```vbnet
pref.PowerSweepRetracePowerMode = naPowerSweepRetraceMode_Start 'Write
psMode = pref.PowerSweepRetracePowerMode 'Read
```

**C++ Syntax**  
`HRESULT get_PowerSweepRetracePowerMode (tagNAPowerSweepRetraceMode* preference);`  
`HRESULT put_PowerSweepRetracePowerMode (tagNAPowerSweepRetraceMode val)`

**Interface**  
`IPreferences3`

---

**Last modified:**
- 30-Aug-2010  
  Fixed Read example
- Oct. 25, 2006  
  New command
### PreciseTuningTolerance Property

**Description**
Sets and returns the tuning tolerance for precise tuning.

**VB Syntax**
```
obj.PreciseTuningTolerance = value
```

**Variable**
- **(Type)** - Description
  - `obj` An [EmbeddedLO](#) (object) or a [ConverterEmbeddedLO](#) (object)
  - `value` (Double) Tuning tolerance in Hz.

**Return Type**
(Double)

**Default**
1 Hz

**Examples**
- `embedLO.PreciseTuningTolerance = .5` 'write
- `value = embedLO.PreciseTuningTolerance` 'read

**C++ Syntax**
```
HRESULT get_PreciseTuningTolerance(double* tolerance);
HRESULT put_PreciseTuningTolerance(double tolerance);
```

**Interface**
IEembeddedLO

---

**Last Modified:**
- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
**PreferInternalTriggerOnChannelSingle Property**

**Description**
Set and read the preference for the `chan.Single` trigger behavior. This setting persists until changed.

These preferences are important when performing a Guided calibration, as the PNA uses the `chan.Single` trigger command to measure standards.

- set `PreferInternalTriggerOnChannelSingle = False` to use an External trigger sweep to measure a cal standard.
- set `PreferInternalTriggerOnChannelSingle = True` to use an External sweep for the measurement, but rely on the PNA to send Internal trigger signals for calibrating.

To set this preference for an **Unguided** Calibration, use `PreferInternalTriggerOnUnguidedCal Property`.

The `chan.Single` trigger command NEVER respects the Trigger Source = Manual setting. It always switches to Internal for one trigger, then back to Manual, regardless of this preference command.

**VB Syntax**

```
pref.PreferInternalTriggerOnChannelSingle = bool
```

**Variable (Type) - Description**

- **pref**
  A Preferences (object)

- **bool** (Boolean) - Choose from:
  
  - 0 - False - the Single trigger property does respect the Trigger Source = External setting. For example, if Trigger source = External, the single trigger method will wait for the External trigger signal and then allow only one sweep.

  - 1 - True - the Single trigger command does NOT respect the Trigger Source = External setting. For example, when the Single method is sent, the PNA immediately switches to Internal sweep, responds to one trigger signal, then switches back to External.

**Return Type**
Boolean

**Default**
0 - False

**Examples**

```
pref.PreferInternalTriggerOnChannelSingle = False 'Write
prefer = pref.PreferInternalTriggerOnChannelSingle 'Read
```

**C++ Syntax**

```
HRESULT put_PreferInternalTriggerOnChannelSingle( VARIANT_BOOL bprefSingle)
HRESULT get_PreferInternalTriggerOnChannelSingle( VARIANT_BOOL *bprefSingle)
```

**Interface**
IPreferences2
### PreferInternalTriggerOnUnguidedCal Property

**Description**  
Set and read the preference for the trigger behavior when performing an Unguided calibration.

**VB Syntax**  
`pref.PreferInternalTriggerOnUnguidedCal = bool`

**Variable (Type) - Description**

- **pref**  
  A `Preferences` (object)

- **bool**  
  (Boolean) - Choose from:

  **0 - False** - The trigger behavior during an Unguided calibration DOES respect the setting of the `Trigger source` command. For example, during an Unguided Cal, when Trigger source = External, the PNA will wait for the External trigger signal and then allow only one sweep.

  **1 - True** - The trigger behavior during an Unguided calibration does NOT respect the `Trigger Source = External` setting. For example, during an Unguided Cal, when Trigger source = External, the PNA immediately switches to Internal sweep, measures the standard with one trigger signal, then switches back to External trigger.

**Note:** When Trigger Source = Manual during a calibration, the PNA ALWAYS switches to Internal for one trigger to measure a standard, then back to Manual, regardless of this preference command.

**Return Type**  
Boolean

**Default**  
0 - False

**Examples**

- `pref.PreferInternalTriggerOnUnguidedCal = False` 'Write

- `prefer = pref.PreferInternalTriggerOnUnguidedCal` 'Read

**C++ Syntax**

- `HRESULT put_PreferInternalTriggerOnUnguidedCal( VARIANT_BOOL bprefUnguided)`
- `HRESULT get_PreferInternalTriggerOnUnguidedCal( VARIANT_BOOL *bprefUnguided)`

**Interface**  
`IPreferences2`
PresetPowerState Property

Description  Set and return the Preset Power ON/OFF state.

VB Syntax  \texttt{pref.PresetPowerState = value}

Variable  
\begin{itemize}
  \item \texttt{pref} A Preferences (object)
  \item \texttt{value} (Enum as NAPowerStates) - Choose from:
    \begin{itemize}
      \item \texttt{naPowerON (0)} - Instrument Preset always turns RF power ON.
      \item \texttt{naPowerAUTO (1)}: \hspace{1cm}
        \begin{itemize}
          \item When the current power setting is OFF, leave power OFF at Preset.
          \item When the current power setting is ON, turn power ON at Preset.
        \end{itemize}
    \end{itemize}
\end{itemize}

Return Type  enum

Default  naPowerON (0)

Examples  \texttt{pref.PresetPowerState = naPowerON } \texttt{ 'Write}
\texttt{pwrState = pref.PresetPowerState } \texttt{ 'Read}

C++ Syntax  HRESULT get_PresetPowerState(tagNAPowerStates* pVal);
HRESULT put_PresetPowerState(tagNAPowerStates pVal);

Interface  IPreferences11

Last Modified:

9-Mar-2010   MX New topic
## PulseMeasMode Property

**Description**
Sets the pulse measurement state for the channel.

**VB Syntax**

\[
pulseMeas.PulseMeasMode = value
\]

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pulseMeas</td>
<td>A PulseMeasurementControl (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0) naPulseMeasurementOff</td>
<td>Turn OFF pulse measurements.</td>
</tr>
<tr>
<td>(1) naPulseStandardMeasurement</td>
<td>Turn ON standard pulse measurements.</td>
</tr>
<tr>
<td>(2) naPulseProfileMeasurement</td>
<td>Turn ON pulse profile measurements.</td>
</tr>
</tbody>
</table>

**Return Type**

Enum

**Default**

(0) naPulseMeasurementOff

**Examples**

\[
pulse.PulseMeasMode = naPulseProfileMeasurement \text{ 'Write}
\]

\[
value = pulse.PulseMeasMode \text{ 'Read}
\]

**C++ Syntax**

```c++
HRESULT get_PulseMeasMode(tagNAPulseMeasurementMode* pVal);
HRESULT put_PulseMeasMode(tagNAPulseMeasurementMode newVal);
```

**Interface**

IPulseMeasurementControl

---

Last Modified:

11-Mar-2010  New topic
RangeCount Property

**Description**: Returns the number of ranges that are available in the PNA. To see the range names, query the Name property of each range in the FOM collection.

**VB Syntax**: `value = FOM.RangeCount`

**Variable**
- **Object**
  - `object` An **FOM** (collection object)
- **Value**
  - `value` (long) - Variable to store the returned number of ranges.

**Return Type**: Long Integer

**Default**: Not Applicable

**Examples**: `NumRanges = fom.RangeCount`  'Read

**C++ Syntax**: `HRESULT get_RangeCount(long *count)`

**Interface**: IFOM

---

**Last Modified:**

8-Mar-2007   Added link to Name property
Read-only

rangeNumber Property

Description
Returns the index number of the range within the FOM collection.

VB Syntax
value = FOMRange.rangeNumber

Variable (Type) - Description
value (Long) - Variable to store the returned range number.

object An FOMRange (object)

Return Type
Long

Default
Not Applicable

Examples
num = fomRange.rangeNumber 'Read

C++ Syntax
HRESULT get_rangeNumber(long *pVal)

Interface
IFOMRange
RatioedPowerCorrectionData Property

**Description**  
Write and read an array of ratioed power offsets. This allows the setting of arbitrary impedance, which is used for active load applications. The number of offset values must be the same as the number of data points. Use [RatioedPowerCorrectionEnabled](#) to use the offset values.

**VB Syntax**  
`phase.RatioedPowerCorrectionData(srcPort) = value`

**Variable**  
- **(Type)** - Description
  - `phase`  
    A [PhaseControl](#) Object
  - `srcPort`  
    (Long Integer) Source port for which to make phase control settings.
    **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#)
  - `value`  
    (Long values) Ratioed amplitude offset data array.

**Return Type**  
Long data array

**Default**  
Not Applicable

**Examples**  
```
phase.RatioedPowerCorrectionData 1 = .1,.2,.3 ' Write 3 power offset values
value = phase.RatioedPowerCorrectionData 2' Read
```

**C++ Syntax**  
```c++
HRESULT get_RatioedPowerCorrectionData(long port, long* pVals);
HRESULT put_RatioedPowerCorrectionData(long port, long newVals);
```

**Interface**  
IPhaseControl

Last Modified:
8-Dec-2010    MX New topic
Write/Read

RatioedPowerCorrectionEnabled Property

Description
Write and read whether to use the ratioed power offset array. Use RatioedPowerCorrectionData Property to write or read the offset data.

VB Syntax
\[ \text{phase.RatioedPowerCorrectionEnabled}(\text{srcPort}) = \text{value} \]

Variable (Type) - Description

\textit{phase} \hspace{1cm} \text{A PhaseControl Object}

\textit{srcPort} \hspace{1cm} \text{(Long Integer) Source port for which to make phase control settings.}

\textbf{Note:} If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use \textit{chan.getPortNumber} to translate the string into a port number. To learn more see \text{Remotely Specifying a Source Port}

\textit{value} \hspace{1cm} \text{(Boolean) Ratioed power offset array state.}

\textbf{True} – Apply offset array.

\textbf{False} – Do NOT apply offset array.

Return Type
Boolean

Default
False

Examples
\textit{phase.RatioedPowerCorrectionEnabled 1 = True} ' Write

\textit{value = phase.RatioedPowerCorrectionEnabled 2} ' Read

C++ Syntax
\begin{verbatim}
HRESULT get_RatioedPowerCorrectionEnabled(long port, VARIANT_BOOL* pVal);
HRESULT put_RatioedPowerCorrectionEnabled(long port, VARIANT_BOOL newVal);
\end{verbatim}

Interface
IPhaseControl

Last Modified:
8-Dec-2010  MX New topic
R1InputPath Property

**Description**

PNA models with option 081, and all PNA-X models, have a switch in the test set that allows access to the port 1 reference receiver through the front panel Reference 1 connectors. This command throws that switch between the internal path to the receiver, or through the external connectors.

See other Frequency Offset properties.

**VB Syntax**

```vbnet
chan.R1InputPath = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chan</td>
<td>A Channel (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Enum as naInputPath) - Choose from:naPathInternal - (0) - internal path to the reference receiver naPathExternal (1) - path through external connectors</td>
</tr>
</tbody>
</table>

**Return Type**

Enum

**Default**

naPathInternal - (0)

**Examples**

```vbnet
chan.R1InputPath = naPathInternal 'Write
```

```vbnet
Inpath = chan.R1InputPath 'Read
```

**C++ Syntax**

```cpp
HRESULT get_R1InputPath (tag NAInputPath *pPath);
HRESULT put_R1InputPath (tag NAInputPath newPath);
```

**Interface**

IChannel2

---

Last Modified:

30-Apr-2009  Added PNA-X models
ReadingsPerPoint Property

This command, along with ReadingsTolerance, allows for settling of the power sensor READINGS. Specifies the maximum number of power readings that are taken at each stimulus point to allow for power meter settling. Each reading is averaged with the previous readings at that stimulus point. When this average meets the ReadingsTolerance value or this number of readings has been made, the average is returned as the valid reading.

**VB Syntax**

\[ \text{pwrSensor.ReadingsPerPoint} = \text{value} \]

**Variable**

* (Type) - Description

- \( pwrCal \) A SourcePowerCalibrator (Object) or A PowerSensorAsReceiver (Object)
- \( \text{value} \) (long integer) – Number of power readings. Choose any number between 3 and 100.

**Return Type**

Long Integer

**Default**

3

**Examples**

\[ \text{pwrSensor.ReadingsPerPoint} = 3 \] 'Write
\[ \text{numReadings} = \text{pwrSensor.ReadingsPerPoint} \] 'Read

**C++ Syntax**

HRESULT put_ReadingsPerPoint(long newVal);
HRESULT get_ReadingsPerPoint(long *pVal);

**Interface**

ISourcePowerCalibrator
IPowerSensorAsReceiver

Last Modified:

25-Aug-2009   Added PMAR
17-Apr-2007   Clarified verbage
ReadingsTolerance Property

**Description**  
This command, along with ReadingsPerPoint Property allows for settling of the power sensor READINGS. Each power reading is averaged with the previous readings at each stimulus point. When the average meets this tolerance value or the maximum ReadingsPerPoint has been made, the average is returned as the valid reading.

**VB Syntax**  
pwrSens.ReadingsTolerance = value

**Variable**  
**Type** - Description

- **pwrCal**  
  A SourcePowerCalibrator (Object) or A PowerSensorAsReceiver (Object)

- **value**  
  **(Double)** – Power meter settling tolerance value in dB. Choose any number between 0 and 5.

**Return Type**  
Double

**Default**  
.05 dB

**Examples**  
pwrSens.ReadingsTolerance = .1 'Write
ReadTol = pwrSensor.ReadingsTolerance 'Read

**C++ Syntax**  
HRESULT get_ReadingsTolerance( double *pVal);
HRESULT put_ReadingsTolerance( double newVal);

**Interface**  
ISourcePowerCalibrator3
IPowerSensorAsReceiver

---

Last Modified:

- 25-Aug-2009  Added PMAR
- 17-Apr-2007  Clarified verbage
## ReadyForTriggerPolarity Property

**Description**
Specifies the polarity of Ready for Trigger output.

All existing Ready for Trigger outputs for PNA-X and PNA-L models are configured simultaneously with this command. [See Capabilities Summary.](#)

The Ready for Trigger polarity can NOT be configured for E836x models.

**VB Syntax**

```vbnet
trigsetup.ReadyForTriggerPolarity = value
```

**Variable**

- **Type**: Description

  - `trigsetup` A [TriggerSetup](object) (object)
  - `value` [Enum as NALevel](#)

  Choose from:

  - **0 - naLow**: Outputs a TTL low when the PNA is ready for trigger.
  - **1 - naHigh**: Outputs a TTL high when the PNA is ready for trigger.

**Return Type**
Enum

**Default**

- **0 - naLow**

**Examples**

```vbnet
trigsetup.ReadyForTriggerPolarity = naLow  'Write
pol = trigsetup.ReadyForTriggerPolarity  'Read
```

**C++ Syntax**

```cpp
HRESULT get_ReadyForTriggerPolarity(tagNALevel *pVal);
HRESULT put_ReadyForTriggerPolarity(tagNALevel newVal);
```

**Interface**

ITriggerSetup3

---

**Last Modified:**

- 14-Mar-2008  MX New topic
About the Handler I/O Connector

ReadyForTriggerState Property

Description: Determines the control of Material Handler connector Pin 21.

VB Syntax: `handler.ReadyForTriggerState = value`

Variable: (Type) - Description

`handler` (object) - A Handler I/O object

`value` (boolean)
False - Pin 21 is controlled by Output Port B7
True - Pin 21 is controlled by the Ready for Trigger signal

Return Type: Boolean

Default: False

Examples: `handler.ReadyForTriggerState = False` 'Write
`bState = handler.ReadyForTriggerState` 'Read

C++ Syntax: `HRESULT put_ReadyForTriggerState (BOOL *pVal);`
`HRESULT get_ReadyForTriggerState (BOOL newVal);`

Interface: IHWMaterialHandlerIO2
## ReceiverAttenuator Property

**Description**
Sets or returns the value of the specified receiver attenuator control.

**VB Syntax**

```vbnet
object.ReceiverAttenuator(rec) = value
```

**Variable**

- **object**
  - Channel *(object)*
  - or
  - CalSet *(object)* - Read-only property

- **rec** *(long integer)* - Receiver with attenuator control to be changed. Choose from any of the available receivers in your PNA
  1 - Receiver A
  2 - Receiver B

Receiver attenuation can not be set using *logical receiver notation*.

- **value** *(double)* - Attenuator value in dB. Choose any Long Integer between 0 and 35 in 5dB steps:
  If an invalid value is entered, the analyzer will select the next lower valid value. For example, if 19.9 is entered the analyzer will select 15 dB attenuation.

**Return Type**

Double

**Default**

0 db

**Examples**

```vbnet
chan.ReceiverAttenuator(1) = 5 'Write

attn = chan.ReceiverAttenuator(2) 'Read
```

**C++ Syntax**

```c++
HRESULT get_ReceiverAttenuator(long lport, double *pVal)
HRESULT put_ReceiverAttenuator(long lport, double newVal)
```

**Interface**

IChannel
|CalSet3
Description
Returns the number of receivers in the remote PNA. The returned number includes both test port receivers and reference receivers. See the number of reference receivers in your PNA.

VB Syntax
value = cap.ReceiverCount

Variable (Type) - Description
value (Long) - Variable to store the returned number of receivers.
cap A Capabilities (object)

Return Type
Long

Default
Not Applicable

Examples
value = cap.ReceiverCount 'Read

C++ Syntax
HRESULT get_ReceiverCount(long * receiverCount);

Interface ICapabilities
About Receiver Leveling

ReceiverRatio Property

**Description**
Sets and returns the receiver ratio pair to be used with Receiver Leveling.
To perform receiver leveling with a ratioed receiver, use `ReceiverRatio Property`.

**VB Syntax**
```
RxLevel.ReceiverRatio(srcPort) = value
```

**Variable**
- **RxLevel** (Type) - Description
  A `ReceiverLeveling` Object

- **srcPort** (Long Integer) Source port for which to set the receiver for Receiver Leveling.
  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see `Remotely Specifying a Source Port`.

- **value** (String) Receiver to use for the leveling sweeps. (Not case sensitive). Choose any receiver in your PNA. See the block diagram of your PNA, located at the bottom of all `Specs documents`.
  Receivers can also be referred to using logical receiver notation. This notation makes it easy to refer to receivers with an external test set connected to the PNA. You do not need to know which physical receiver is used for each test port. Learn more.

**Return Type**
String

**Default**
Not applicable

**Examples**
```
rxLevel.ReceiverRatio (1) = "R1"  ' Write
rxLevel.ReceiverRatio (1) = "b2"  ' Write
value = rxLevel.ReceiverRatio 2  ' Read
```

**C++ Syntax**
```
HRESULT get_ReceiverRatio( long port, BSTR* pVal);
HRESULT put_ReceiverRatio( long port, BSTR newVal);
```

**Interface**
`IReceiverLevelingConfiguration`

---

Last Modified:

- 22-Dec-2010 Added link to ratioed rec.
- 13-Feb-2009 MX New topic
## ReceiverStepAttenuatorStepSize Property

**Description**
Returns a value indicating the step size of the attenuator.

**VB Syntax**
value = cap.ReceiverStepAttenuatorStepSize(n)

**Variable**
- **value** (Double) - Variable to store the returned value of the attenuator step size.
- **cap** A Capabilities (object)
- **n** (Long) - port number to query for the value of the attenuator step size.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
value = cap.ReceiverStepAttenuatorStepSize(1) 'Read

**C++ Syntax**
HRESULT get_ReceiverStepAttenuatorStepSize(long portNumber, double * stepSize);

**Interface**
ICapabilities
Read-only

ReceivePort Property

Description
Returns the receiver (response) port number of measurement. To understand how this property is useful, see IMeasurement2 Interface.

Note: Returning a receiver port is only supported for S-Parameter measurements. If the measurement is not an S-Parameter, then E_NA_BAD_PARAMETER is returned.

VB Syntax
value = meas.ReceivePort

Variable (Type) - Description
value (Long) - Variable to store the returned value
meas A Measurement (object)

Return Type
Long Integer

Default
Not Applicable

Examples
rp = meas.ReceivePort

C++ Syntax
HRESULT ReceivePort(Long* rcvPort);

Interface
IMeasurement2
## RedTraceOnFail Property

**Description**
Set and return whether to display limit line failures as red trace segments or red data points (dots).

**VB Syntax**
```
pref.RedTraceOnFail = bool
```

### Variable (Type) - Description
- **pref**: A `Preferences` (object)
- **bool** (Boolean) - Choose from:
  - **False**: Display failures as red data points (dots).
  - **True**: Display failures as red trace segments. (Red Trace On Fail).

**Return Type**
Boolean

**Default**
False

**Examples**
```
pref.RedTraceOnFail = False 'Write
prefer = pref.RedTraceOnFail 'Read
```

### C++ Syntax
```
HRESULT put_RedTraceOnFail( VARIANT_BOOL bValue)
HRESULT get_RedTraceOnFail( VARIANT_BOOL *bValue)
```

**Interface**
IPreferences10

---

**Last Modified:** 11-Aug-2009

MX New topic
**ReduceIFBandwidth Property**

**Description**
Sets or returns the state of the Reduced IF Bandwidth at Low Frequencies setting.

**VB Syntax**
```
chan.ReduceIFBandwidth = state
```

**Variable**
(Type) - Description
- **chan** A Channel (object)
- **state** (boolean)
  - False - Turns Reduce IFBW OFF
  - True - Turns Reduce IFBW ON

**Return Type**
Boolean

**Default**
True

**Examples**
- `chan.ReduceIFBandwidth = False 'Write`
- `reduce = chan.ReduceIFBandwidth 'Read`

**C++ Syntax**
- `HRESULT get_ReduceIFBandwidth(BOOL *pVal)`
- `HRESULT put_ReduceIFBandwidth(BOOL newVal)`

**Interface**
IChannel5

---

Last Modified:
- 16-Aug-2007 Corrected Interface number
Write/Read

About Source Power Cal

About PMAR

ReferenceCalFactor Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the Cal Factor (%) for the 50 MHz reference signal associated with this power sensor. Use this property only if the power sensor does not contain cal factors in EPROM (for example, HP/Agilent 848x sensors).</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>pwrSensor.ReferenceCalFactor = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><em>(Type) - Description</em></td>
</tr>
<tr>
<td>pwrSensor</td>
<td>A <code>PowerSensor</code> (Object)</td>
</tr>
<tr>
<td></td>
<td>A <code>PowerSensorAsReceiver</code> (Object)</td>
</tr>
<tr>
<td>value</td>
<td><em>(double)</em> – Cal factor in units of percent. This can be any value between 1 and 150.</td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>100</td>
</tr>
<tr>
<td>Examples</td>
<td><code>pwrSensor.ReferenceCalFactor = 98 ' R</code></td>
</tr>
<tr>
<td></td>
<td><code>RefFact = pwrSensor.ReferenceCalFactor ' Read</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT put_ReferenceCalFactor(double newVal);</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT get_ReferenceCalFactor(double *pVal);</code></td>
</tr>
<tr>
<td>Interface</td>
<td><code>IPowerSensor</code></td>
</tr>
<tr>
<td></td>
<td><code>IPowerSensorAsReceiver</code></td>
</tr>
</tbody>
</table>

Last Modified:

25-Aug-2009    Added PMAR
**ReferenceMarkerState Property**

**Description**  
Turn ON or OFF the reference marker.

**VB Syntax**  
`meas.ReferenceMarkerState = state`

**Variable**  
*(Type) - Description*

- **app**  
  A Measurement *(object)*

- **state**  
  (boolean) -  
  *True* - turns the reference marker ON  
  *False* - turns the reference marker OFF

**Return Type**  
Boolean

**Default**  
False

**Examples**  
`meas.ReferenceMarkerState = True`

`reference = meas.ReferenceMarkerState`

**C++ Syntax**  
HRRESULT get_ReferenceMarkerState(VARIANT_BOOL bState)  
HRRESULT put_ReferenceMarkerState(VARIANT_BOOL* bState)

**Interface**  
IMeasurement
ReferenceReceiver Property

**Description**
Sets and returns the receiver to be used with Receiver Leveling.
To perform receiver leveling with a ratioed receiver, use `ReceiverRatio Property`.

**VB Syntax**
```vbnet
RxLevel.ReferenceReceiver(srcPort) = value
```

**Variable (Type) - Description**

- **RxLevel**
  A `ReceiverLeveling` Object

- **srcPort**
  (Long Integer) Source port for which to set the receiver for Receiver Leveling.
  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **value**
  (String) Receiver to use for the leveling sweeps. (Not case sensitive). Choose any receiver in your PNA. see the block diagram of your PNA, located at the bottom of all `Specs documents`.
  Receivers can also be referred to using logical receiver notation. This notation makes it easy to refer to receivers with an external test set connected to the PNA. You do not need to know which physical receiver is used for each test port. [Learn more](#).

**Return Type**
String

**Default**
Not applicable

**Examples**
```vbnet
rxLevel.ReferenceReceiver (1) = "R1" ' Write
rxLevel.ReferenceReceiver (1) = "b2" ' Write
value = rxLevel.ReferenceReceiver 2' Read
```

**C++ Syntax**
```cpp
HRESULT get_ReferenceReceiver( long port, BSTR* pVal);
HRESULT put_ReferenceReceiver( long port, BSTR newVal);
```

**Interface**
IRxReceiverLevelingConfiguration

---

Last Modified:

- 22-Dec-2010  Added link to ratioed rec.
- 13-Feb-2009  MX New topic
ReferenceValue Property

Description
Sets or returns the value of the Y-axis Reference Level of the active trace.

VB Syntax
\[ trce.ReferenceValue = value \]

Variable
(Type) - Description

\( trce \) A Trace (object)

\( value \) (double) - Reference Value. Units and range depend on the current data format.

Return Type
Double

Default
Not applicable

Examples
\[ meas.ReferenceValue = 0 \ 'Write \]
\[ rlev = meas.ReferenceValue \ 'Read \]

C++ Syntax
HRESULT get_ReferenceValue(double *pVal)
HRESULT put_ReferenceValue(double newVal)

Interface
ITrace
## ReferencePosition Property

**Description**
Sets or returns the Reference Position of the active trace.

**VB Syntax**

```
trce.ReferencePosition = value
```

**Variable**

<table>
<thead>
<tr>
<th><strong>(Type)</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>trce</code></td>
<td>A Trace (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) - Reference position on the screen measured in horizontal graticules from the bottom of the screen. Choose from any number between: 0 and 10.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0

**Examples**

```
meas.ReferencePosition = 5 'Middle of the screen -Write
rpos = meas.ReferencePosition -Read
```

**C++ Syntax**

```
HRESULT get_ReferencePosition(double *pVal)
HRESULT put_ReferencePosition(double newVal)
```

**Interface**
ITrace
RemoteCalStoragePreference Property

Description
Specifies the default manner in which calibrations performed using COM or SCPI are to be stored. Cal data is always stored to the channel’s Cal Register regardless of this setting. This setting survives instrument preset and reboot. It remains until changed by another invocation of this property.

VB Syntax
`pref.RemoteCalStoragePreference = value`

Variable (Type) - Description
- `cal` (A Preferences (object))
- `value` (Enum) - Choose from:
  - 0 - `naPreferCalRegister` - Cal is saved ONLY to the channel’s Cal Register.
  - 1 - `naPreferNewUserCalSet` - Cal is automatically saved to a new User Cal Set file when performing a calibration using COM. The Cal Set name is automatically generated. This corresponds to pre-6.0 behavior. Use the `Name` property to change the name after the cal is complete.
  - 2 - `naPreferReuseCurrentCalSet` - The cal is saved to the Cal Set that is currently selected on the specific channel. This could be the channel’s Cal Register. If the channel does not yet have a selected Cal Set, the cal will be saved to a new User Cal Set with an automatically-generated name.

Return Type
Enum

Default
0 - `naPreferCalRegister`

Examples
`pref.RemoteCalStoragePreference = naPreferNewUserCalSet`  `'Write`
`calStorageMode = pref.RemoteCalStoragePreference`  `'Read`

C++ Syntax
`HRESULT get_RemoteCalStoragePreference(enum NARemoteCalStoragePreference* preference);`
`HRESULT put_RemoteCalStoragePreference(enum NARemoteCalStoragePreference val);`

Interface
IPreferences7

Last Modified:
16-Apr-2007  MX New topic
# ReportReceiverOverload Property

**Description**  
Set and return whether to display receiver overload warnings.

**VB Syntax**  
`pref.ReportReceiverOverload = value`

**Variable**  
*pref* A [Preferences](object) (object)

*value* (Boolean) - Choose from:
- **True**  Display overload warnings.
- **False**  Do NOT display overload warnings.

**Return Type**  
Boolean

**Default**  
True

**Examples**  
```vba  
pref.ReportReceiverOverload = True 'Write  
value = pref.ReportReceiverOverload 'Read  
```

**C++ Syntax**  
```cpp  
HRESULT get_ReportReceiverOverload (VARIANT_BOOL PowerSweepRetraceMode* preference);  
HRESULT put_ReportReceiverOverload (VARIANT_BOOL PowerSweepRetraceMode val)  
```

**Interface**  
IPreferences12

---

**Last Modified:**  
30-Aug-2010  
MX New topic
### ResolutionBW Property

**Description**  
Sets and returns the Resolution Bandwidth for the IM Spectrum measurement.

**VB Syntax**  
```vbnet
ims.ResolutionBW = value
```

**Variable**  
*ims* - An `IMSpectrum` Object

*value* - (Double) Resolution BW in Hz. Choose from:
- 60k  | 100k  | 150k  | 300k  | 600k  | 1.0M  | 3.0M

If an invalid number is specified, the PNA will round up to the closest valid number.

**Return Type**  
Double

**Default**  
600 kHz

**Examples**  
```vbnet
ims.ResolutionBW = 150e3 'Write
```

```vbnet
value = ims.ResolutionBW 'Read
```

**C++ Syntax**  
```c++
HRESULT get_ResolutionBW(double *pVal)
HRESULT put_ResolutionBW(double newVal)
```

**Interface**  
`IIMSpectrum`

---

Last Modified:

19-Aug-2008  
MX New topic
Write/Read

Reverse2PortAdapter Property

Description  Set and read whether or not to reverse ports on a 2-port fixture or adapter to be de-embedded.

VB Syntax  `fixture.Reverse2PortAdapter (port) = bool`

Variable  (Type) - Description

`fixture` A Fixturing (object)

`port` PNA port number for which SNP file is to be de-embedded.

`bool` True - Reverse ports.
    False - Do NOT reverse ports.

Return Type  Boolean

Default  False

Examples  `fixture.Reverse2PortAdapter = True`

    `value = fixture.Reverse2PortAdapter 'Read`

C++ Syntax  HRESULT get_Reverse2PortAdapter(short portNum, VARIANT_BOOL *pRev);
    HRESULT put_Reverse2PortAdapter(short portNum, VARIANT_BOOL bRev);

Interface  IFixturing6

Last Modified:

16-Nov-2010  MX New topic
Write/Read

ReverseLinearPowerLevel Property

Description
Set and read the reverse power level to the DUT. This is applied to the DUT output port when making reverse measurements like S22.

VB Syntax
\[ \text{gca.ReverseLinearPowerLevel} = \text{value} \]

Variable (Type) - Description
- \textit{gca} A \textbf{GainCompression} (object)
- \textit{value} (double) Reverse power level in dBm. Choose a value from +30 to -30.
- \textit{Return Type} Double
- \textit{Default} -5

Examples
\[ \text{gca.ReverseLinearPowerLevel} = -10 \quad \text{\texttt{Write}} \]
\[ \text{LinPwr} = \text{gca.ReverseLinearPowerLevel} \quad \text{\texttt{Read}} \]

C++ Syntax
- \textbf{HRESULT get_ReverseLinearPowerLevel(double* pVal)}
- \textbf{HRESULT put_ReverseLinearPowerLevel(double newVal)}

Interface
\textbf{IGainCompression}

Last Modified:
21-Nov-2007   MX New topic
## Write/Read

### About Receiver Overload

#### RFOffOnReceiverOverload Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Set and return whether to turn source power OFF when a receiver is overloaded.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>pref.RFOffOnReceiverOverload = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>Type</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td></td>
<td><code>pref</code> (A <strong>Preferences</strong> (object))</td>
</tr>
<tr>
<td></td>
<td><code>value</code> (Boolean)  - Choose from:</td>
</tr>
<tr>
<td></td>
<td><strong>True</strong>  Turn OFF source power to ALL ports when a receiver is overloaded.</td>
</tr>
<tr>
<td></td>
<td><strong>False</strong> Power remains ON when a receiver is overloaded.</td>
</tr>
<tr>
<td>Return Type</td>
<td>boolean</td>
</tr>
<tr>
<td>Default</td>
<td>False</td>
</tr>
<tr>
<td>Examples</td>
<td><code>pref.RFOffOnReceiverOverload = True</code> 'Write'</td>
</tr>
<tr>
<td></td>
<td><code>value = pref.RFOffOnReceiverOverload</code> 'Read'</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_RFOffOnReceiverOverload (VARIANT_BOOL* preference);</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_RFOffOnReceiverOverload (VARIANT_BOOL val)</td>
</tr>
<tr>
<td>Interface</td>
<td>IPreferences12</td>
</tr>
</tbody>
</table>

Last Modified:

- 30-Aug-2010  MX New topic
SafeMode Property

Description  Sets and returns the state of Safe Mode.

VB Syntax  

\[
RxLevel.SafeMode(srcPort) = value
\]

Variable  

\( RxLevel \)  A \( ReceiverLeveling \) Object

\( value \)  (Boolean) Choose from:

- \( True \) - Safe mode ON
- \( False \) - Safe mode OFF

\( srcPort \)  (Long Integer) Source port for which to set the Safe Mode state for Receiver Leveling.

Note: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use \( chan.getPortNumber \) to translate the string into a port number. To learn more see Remotely Specifying a Source Port

Return Type  Variant Boolean

Default  False

Examples  

\[
rxLevel.SafeMode (1) = True ' Write
\]

\[
value = rxLevel.SafeMode 2' Read
\]

C++ Syntax  

\[
HRESULT get_SafeMode(long port, VARIANT_BOOL* pLevelingSafeMode);
\]

\[
HRESULT put_SafeMode(long port, VARIANT_BOOL LevelingSafeMode);
\]

Interface  \( IReceiverLevelingConfiguration \)

Last Modified:  
13-Feb-2009  MX New topic
### SafeSweepCoarsePowerAdjustment Property

**Description**  Set and read the Safe Sweep COARSE power adjustment.

**VB Syntax**  

```vbnet
gca.SafeSweepCoarsePowerAdjustment = value
```

**Variable**  

- **Type**
  - `gca`  A `GainCompression` object
  - `value`  (Double) Coarse power adjustment setting in dBm. Choose a value from +30 to (-30).

**Return Type**  Double

**Default**  3.0

**Examples**  

```vbnet
gca.SafeSweepCoarsePowerAdjustment = 2.0 'Write

SSCourse = gca.SafeSweepCoarsePowerAdjustment 'Read
```

**C++ Syntax**  

- `HRESULT get_SafeSweepCoarsePowerAdjustment(double* value)`
- `HRESULT put_SafeSweepCoarsePowerAdjustment(double value)`

**Interface**  `IGainCompression`

---

**Last Modified:**  1-Dec-2007  MX New topic
**SafeSweepEnable Property**

**Description**  
Set and read the (ON | OFF) state of Safe Sweep mode.

**VB Syntax**  
gca.SafeSweepEnable = value

**Variable**  
*Type* - Description

- **gca** A **GainCompression** (object)
- **value** (Boolean) Safe Sweep state. Choose from:
  - **False** - Disable Safe Sweep
  - **True** - Enable Safe Sweep

**Return Type**  
Boolean

**Default**  
False

**Examples**  
gca.SafeSweepEnable = True 'Write
SSEnable = gca.SafeSweepEnable 'Read

**C++ Syntax**  
HRESULT get_SafeSweepEnable(VARIANT_BOOL* value)
HRESULT put_SafeSweepEnable(VARIANT_BOOL value)

**Interface**  
IGainCompression

---

Last Modified:

- 1-Dec-2007  MX New topic
### SafeSweepFinePowerAdjustment Property

**Description**  
Set and read the Safe Sweep FINE power adjustment.

**VB Syntax**  
gca.SafeSweepFinePowerAdjustment = value

**Variable**  
- **Type**  
  - `gca` A `GainCompression` (object)
  
- **value** (Double) Fine power adjustment setting in dBm. Choose a value from +30 to (-30).

**Return Type**  
Double

**Default**  
1.0

**Examples**  
gca.SafeSweepFinePowerAdjustment = 0.1 'Write

SSfine = gca.SafeSweepFinePowerAdjustment 'Read

**C++ Syntax**  
HRESULT get_SafeSweepFinePowerAdjustment(double* value)  
HRESULT put_SafeSweepFinePowerAdjustment(double value)

**Interface**  
IGainCompression

---

Last Modified:

1-Dec-2007  
MX New topic
Write/Read

SafeSweepFineThreshold Property

Description
Set and read the compression level at which Safe Sweep changes from the COARSE power adjustment to the FINE power adjustment.

VB Syntax
`gca.SafeSweepFineThreshold = value`

Variable (Type) - Description
- `gca` A `GainCompression` (object)
- `value` (Double) Threshold setting in dBm. Choose a value from +30 to (-30).

Return Type
Double

Default
0.5 dBm

Examples
`gca.SafeSweepFineThreshold = 0.1` 'Write
`SSThresh = gca.SafeSweepFineThreshold` 'Read

C++ Syntax
HRESULT get_SafeSweepFineThreshold(double* value)
HRESULT put_SafeSweepFineThreshold(double value)

Interface
IGainCompression

Last Modified:
1-Dec-2007  MX New topic
### SafeSweepMaximumLimit Property

**Description**  
When the PNA port that is connected to the DUT Output measures the specified value, the input power to the DUT is no longer incremented at that frequency.

**VB Syntax**  
`gca.SafeSweepMaximumLimit = value`

**Variable**  
*Type* - Description  
- **gca** A [GainCompression](#) (object)
- **value** (Double) Maximum power level in dBm. Choose a value from -100 to +100.

**Return Type**  
Double

**Default**  
100

**Examples**  
```vbnet  
gca.SafeSweepMaximumLimit = 23 'Write
maxPwr = gca.SafeSweepMaximumLimit 'Read
```

**C++ Syntax**  
```cpp  
HRESULT get_SafeSweepMaximumLimit(double* value)  
HRESULT put_SafeSweepMaximumLimit(double value)
```

**Interface**  
IGainCompression

---

**Last Modified:**  
16-Mar-2010  
MX New topic
### SaturationLevel Property

**Description**
Set and read the deviation dB from the maximum Pout. This is the point of saturation. This value is used for Compression Method: Compression from Saturation.

**VB Syntax**
```
gca.SaturationLevel = value
```

**Variable**
- **(Type)** - Description
  - `gca` A [GainCompression](#) (object)
  - `value` (Double) - Saturation level in dB. Choose a value greater than 0.01 dB.

**Return Type**
Double

**Default**
.1 dB

**Examples**
```
gca.SaturationLevel = .5 'Write
satLevel = gca.SaturationLevel 'Read
```

**C++ Syntax**
```
HRESULT get_SaturationLevel(double* pVal)
HRESULT put_SaturationLevel(double newVal)
```

**Interface**
IGainCompression3

---

**Last Modified:**
3-Sep-2009 MX New topic

---
Read-only

SB_BalPortNegative Property

Description
With a Single-ended - Balanced topology, returns the PNA port number that is connected to the Negative side of the DUT's Balanced Port.
Use SetSBPorts Method to set the port mapping for a Single-Ended - Balanced topology.

VB Syntax
var = balTopology.SB_BalPortNegative

Variable
(Type) - Description
var (Long Integer) Variable to store the returned value.

balTopology A BalancedTopology (object)

Return Type
Long Integer

Default
Not Applicable

Examples
variable = balTop.SB_BalPortNegative 'Read

C++ Syntax
HRESULT get_SB_BalPortNegative(long *bVal)

Interface
IBalancedTopology
### SB_BalPortPositive Property

**Description**  
With a Single-ended - Balanced topology, returns the PNA port number that is connected to the Positive side of the DUT's Balanced Port. Use [SetSBPorts Method](#) to set the port mapping for a Single-Ended - Balanced topology.

**VB Syntax**  
```
var = balTopology.SB_BalPortPositive
```

**Variable** *(Type) - Description*
- **var** *(Long Integer)* Variable to store the returned value.

**balTopology**  
A [BalancedTopology](#) *(object)*

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**  
```
variable = balTop.SB_BalPortPositive  'Read
```

**C++ Syntax**  
```
HRESULT get_SB_BalPortPositive(long *bVal)
```

**Interface**  
IBalancedTopology
### SB_SEPort Property

**Description**
With a Single-ended - Balanced topology, returns the PNA port number that is connected to the DUT's Single-ended port.

Use [SetSBPorts Method](#) to set the port mapping for a Single-Ended - Balanced topology.

**VB Syntax**

```vbnet
var = balTopology.SB_SEPort
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>var</td>
<td>(Long Integer) Variable to store the returned value.</td>
</tr>
</tbody>
</table>

**balTopology**
A [BalancedTopology](#) (object)

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**

```
variable = balTopology.SB_SEPort  'Read
```

**C++ Syntax**

```c++
HRESULT get_SB_SEPort(long *bVal)
```

**Interface**
IBalancedTopology
### SBalMeasurement Property

**Description**
Sets and returns the measurement for the Single-Ended - Balanced topology.

**VB Syntax**

```vbnet
balMeas.SBalMeasurement = value
```

**Variable (Type) - Description**

- **balMeas**
  - A `BalancedMeasurement` (object)

- **value**
  - (String) - Single-ended - Balanced Measurement parameter. Not case-sensitive. Choose from:

<table>
<thead>
<tr>
<th>Sss11</th>
<th>Ssd12</th>
<th>Ssc12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sds21</td>
<td>Sdd22</td>
<td>Sdc22</td>
</tr>
<tr>
<td>Scs21</td>
<td>Scd22</td>
<td>Scc22</td>
</tr>
<tr>
<td>Imb</td>
<td>CMRR1</td>
<td>CMRR2</td>
</tr>
<tr>
<td></td>
<td>(Sds21/Scs21)</td>
<td>(Ssd12/Scs12)</td>
</tr>
</tbody>
</table>

**Return Type**

Sss11

**Default**
Not Applicable

**Examples**

```
balMeas.SBalMeasurement = "Ssd12"  'Write
variable = balMeas.SBalMeasurement  'Read
```

**C++ Syntax**

```c++
HRESULT get_SBalMeasurement(BSTR *pVal)
HRESULT put_SBalMeasurement(BSTR newVal)
```

**Interface**

`IBalancedMeasurement`
## ScaleCouplingMethod Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the method of scale coupling.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>win.ScaleCouplingMethod = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>win</code></td>
<td>An <a href="object">NAWindow</a> (object) Any window object can be used to set this global property.</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(enum NAScaleCouplingMethod)</td>
</tr>
<tr>
<td>0</td>
<td><code>naScaleCouplingOff</code> - Scale Coupling is Off</td>
</tr>
<tr>
<td>1</td>
<td><code>naScaleCouplingWindow</code> - Traces within selected windows share scaling</td>
</tr>
<tr>
<td>2</td>
<td><code>naScaleCouplingAll</code> - Scaling is shared among traces in all selected windows Select windows using <a href="#">ScaleCouplingState Property</a></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Enum</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>0 – <code>naScaleCouplingOff</code></td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>win.ScaleCouplingMethod = naScaleCouplingWindow</code> 'Write method = app.ActiveNAWindow.ScaleCouplingMethod' 'Read'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_ScaleCouplingMethod(tagNAScaleCouplingMethod* couplingMethod);</code> <code>HRESULT put_ScaleCouplingMethod(tagNAScaleCouplingMethod couplingMethod);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>INAWWindow2</td>
</tr>
</tbody>
</table>
## ScaleCouplingState Property

**Description**
Enables and disables scale coupling for the window. Use `ScaleCouplingMethod` to select the coupling method.

**VB Syntax**
```
win.ScaleCouplingState = bool
```

**Variable**
<table>
<thead>
<tr>
<th><strong>Type</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>win</code></td>
<td>An <code>NAWindow</code> (object).</td>
</tr>
<tr>
<td><code>bool</code></td>
<td>(Boolean)</td>
</tr>
<tr>
<td><code>False</code></td>
<td>NO scale coupling for this window.</td>
</tr>
<tr>
<td><code>True</code></td>
<td>Scale coupling enabled for this window.</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
True

**Examples**
```
win.ScaleCouplingState = false 'Write
```
```
coupled = app.ActiveNAWindow.ScaleCouplingState 'Read
```

**C++ Syntax**
```
HRESULT get_ScaleCouplingState(VARIANT_BOOL *pVal);
HRESULT put_ScaleCouplingState(VARIANT_BOOL newVal)
```

**Interface**
`INAWindow2`

---

**Last Modified:**
6-Aug-2009  MX New topic
## Scope Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the scope of a trigger signal. This determines whether a trigger signal affects a single channel or all channels in the PNA.</th>
</tr>
</thead>
</table>

**Note:** Trigger Modes Point and EverySweep require that Trigger.Scope be set to `naChannelTrigger`.

### VB Syntax

```vbnet
trigsetup.Scope = value
```

### Variable (Type) - Description

- **trigsetup** A `TriggerSetup (object)`
- **value** (enum `NATriggerType`) - Trigger type. Choose from:
  - 0 - `naGlobalTrigger` - a trigger signal is applied to all triggerable channels
  - 1 - `naChannelTrigger` - a trigger signal is applied to the current channel. The next trigger signal will be applied to the next channel; not necessarily the next channel in numeric sequence (1-2-3-4 and so forth).

### Return Type

Long Integer

### Default

`naGlobalTrigger`

### Examples

```vbnet
trigsetup.Scope = naGlobalTrigger 'Write
trigtyp = trigsetup.Scope 'Read
```

### C++ Syntax

```cpp
HRESULT get_Scope(tagNATriggerType *pTrigger)
HRESULT put_Scope(tagNATriggerType trigger)
```

### Interface

`ITriggerSetup`

---

Last Modified:

6-Nov-2007 Updated for new sweep mode
Write/Read

About Marker Search

SearchFunction Property

Description
Emulates the Tracking function in the marker search dialog box. The value you choose for SearchFunction will determine the type of search that takes place when the Tracking property is set true.

The tracking function finds the selected search function every sweep. In effect, turning Tracking ON is the same as executing one of the "Search..." methods (such as SearchMin, SearchMax) for every sweep.

VB Syntax
(mark.SearchFunction = value)

Variable
(Type) - Description

mark A Marker (object)
value (enum NAMarkerFunction) - search function. Choose from:
0 - naMarkerFunction_None
1 - naMarkerFunction_Min
2 - naMarkerFunction_Max
3 - naMarkerFunction_Target
4 - naMarkerFunction_NextPeak
5 - naMarkerFunction_PeakRight
6 - naMarkerFunction_PeakLeft
7 - naMarkerFunction_Compression

Return Type
Long Integer

Default
0 - naMarkerFunction_None

Examples
mark.SearchFunction = naMarkerFunction_Target 'When this marker is set to track, it will track the Target value.

searchfunction = mark.SearchFunction 'Read

C++ Syntax
HRESULT get_SearchFunction(tagNAMarkerFunction *pVal)
HRESULT put_SearchFunction(tagNAMarkerFunction newVal)

Interface IMarker

1441
**SearchFailures Property**

**Description**
Returns a comma-separated list of the frequency indexes that were out of tolerance for SMART Sweep mode, or at the power limit for 2D Sweep mode. Zero (0) is the first frequency data point.

Must be Single triggered. Invalid results occur if the GCA channel is continuously sweeping.

**VB Syntax**
```vbnet
value = gca.SearchFailures
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>Double</td>
<td>Variable to store the returned data.</td>
</tr>
<tr>
<td>gca</td>
<td>GainCompression</td>
<td>A GainCompression (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Returns a comma-separated list of frequency indexes.

**Default**
Not applicable

**Examples**
```vbnet
SFA = gca.SearchFailures 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SearchFailures(VARIANT* value)
```

**Interface**
IGainCompression

---

Last Modified:

11-Dec-2007    MX New topic
SearchSummary Property

**Description**
Returns the status of a compression search.
This command can be used to indicate when a GCA search is complete.

**Note:** The returned value reflects the current state of the GCA compression search which can vary when in continuous sweep. This command is intended to be used with `chan.Single` (trigger).

**VB Syntax**

```vb
value = gca.SearchSummary
```

**Variable** *(Type) - Description*

`value` *(Enum)* Variable to store the returned value.

- **0** - `naSearchNotDone` - Acquisition is still in process.
- **1** - `naSearchSucceeded` - Acquisition is complete and compression value found for all frequency points.
- **2** - `naSearchFailed` - Acquisition is complete and unable to find the compression value at one or more frequency points.

**gca** A `GainCompression` *(object)*

**Return Type**
Enum

**Default**
Not Applicable

**Examples**

```vb
sum = gca.SearchSummary
```

**C++ Syntax**

```c++
HRESULT get_SearchSummary(enum naGCASearchSummary* value)
```

**Interface**
`IGainCompression`

---

Last Modified:
10-Jun-2009   MX New topic
SecurityLevel Property

Description
Controls the display of frequency information on the PNA screen and printouts.

VB Syntax
```
app.SecurityLevel value
```

Variable (Type) - Description

*app*  An *Application* (object)

*value* (enum NASecurityLevel) - Choose from:

0 - *naNoSecurity*  ALL frequency information is displayed.

1 - *naLowSecurity*  NO frequency information is displayed. Frequency information can be redisplayed using the Security Setting dialog box or this command.

2 - *naHighSecurity*  LOW setting plus GPIB console is disabled. Frequency information can be redisplayed ONLY by performing a Preset, recalling an instrument state with None or Low security settings, or using this command.

3 - *naExtraSecurity*  HIGH setting plus:

- **ASCII data saving** is disabled. Same method to redisplay frequency information as HIGH setting.
- **Mixer setup files** (*.mxr) can NOT be saved.

Return Type
Long Integer

Default
0 - None

Examples
```
app.SecurityLevel = naLowSecurity  'Write

level = app.SecurityLevel  'Read
```

C++ Syntax
```
HRESULT get_NASecurityLevel(tagNASecurityLevel *level);
HRESULT put_NASecurityLevel(tagNASecurityLevel level);
```

Interface
IApplication4

Last Modified:
17-Jul-2007  Add Extra level
SegmentNumber Property

**Description**
Returns the number of the current segment, PowerSensorCalFactorSegment or PowerLossSegment object.

**VB Syntax**
```
seg.SegmentNumber
```

**Variable**
(Type) - Description

- `seg` Any of the following objects:
  - A `Segment` (Object)
  - A `PowerSensorCalFactorSegment` (Object)
  - A `PowerSensorCalFactorSegmentPMAR` Object (Object)
  - A `PowerLossSegment` (Object)
  - A `PowerLossSegmentPMAR` (Object)

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```
segNum = seg.SegmentNumber 'returns the segment number -Read
```

**C++ Syntax**
```
HRESULT get_SegmentNumber(long *pVal)
```

**Interface**
ISegment
- IPowerSensorCalFactorSegment
- IPowerSensorCalFactorSegmentPMAR
- IPowerLossSegment
- IPowerLossSegmentPMAR

**Last Modified:**
27-Aug-2009  Added PMAR (9.0)
## SegmentCount Property

**Description**
Returns the number of segments on the Applied mixer.

**VB Syntax**

```vbnet
value = conv.SegmentCount
```

**Variable**

- **value** (Long integer) Variable in which to store the returned segment count.
- **conv** A [Converter Object](#)

**Return Type**
Long integer

**Default**
1 Segment is created on new converter objects.

**Examples**

```vbnet
count = mxr.SegmentCount
```

[See example program](#)

**C++ Syntax**

```c
HRESULT get_SegmentCount(long *value);
```

**Interface**
IConverter5

---

Last Modified:

- 26-Oct-2010  
  New command (A.09.33)
SegmentFixedFrequency Property

**Description**  
Set and read the CW Frequency for mixer segments. The specified SegmentRangeMode must be set to Fixed. Send Apply before sending a query (read). Learn more.

**VB Syntax**  
`conv.SegmentFixedFrequency(index, range) = value`

**Variable (Type) - Description**

- **conv**  
  A Converter Object

- **index**  
  (Long integer) Segment for which fixed frequency is being set. Choose a segment between 1 and the current segment count. Use SegmentCount Property to read the current count in the Applied Mixer.

- **range**  
  (Enum as ConverterFrequencyRange) Range for which fixed frequency is being set. Choose from:
  - 0 - naInputFrequencies - set input frequency
  - 1 - naOutputFrequencies - set output frequency
  - 2 - naLO1Frequencies - set LO1 frequency
  - 3 - naLO2Frequencies - set LO2 frequency
  - 4 - naIFFrequencies - set IF frequency

- **value**  
  (Double) CW Frequency. Choose a value within the frequency range of the PNA.

**Return Type**  
Double

**Default**  
Center frequency of the PNA

**Examples**

```vbnet
mxr.SegmentFixedFrequency(1, 0) = 1e9 'sets the input frequency to 1 GHz
```

See example program

**C++ Syntax**

```csharp
HRESULT get_SegmentFixedFrequency(long index, tagConverterFrequencyRange range, double *value);
HRESULT put_SegmentFixedFrequency(long index, tagConverterFrequencyRange value, double value);
```

**Interface**  
IConverter5

Last Modified:
26-Oct-2010  New command (A.09.33)
### SegmentFixedPower Property

#### Description
Set and read the fixed power level for the mixer segment.

#### VB Syntax
```
conv.SegmentFixedPower(index,range) = value
```

#### Variable
<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>conv</code></td>
<td>A Converter Object</td>
</tr>
<tr>
<td><code>index</code></td>
<td>(Long integer) Segment for which power is being set. Choose a segment between 1 and the current segment count. Use <code>SegmentCount Property</code> to read the current count in the Applied Mixer.</td>
</tr>
</tbody>
</table>
| `range` | (Enum as ConverterFrequencyRange) Range for which power is being set. Choose from: 
0 - `nalInputFrequencies` - set input power
1 - `nalOutputFrequencies` - set output power
2 - `nalLO1Frequencies` - set LO1 power
3 - `nalLO2Frequencies` - set LO2 power
4 - `nalIFrequencies` - set IF power |
| `value` | (Double) Power level in dBm. Choose a value within the power/attenuation range of the PNA. |

#### Return Type
Double

#### Default

#### Examples
```
mxr.SegmentFixedPower(1,0)=0 'sets the input power level to 0 dBm
```

#### C++ Syntax
```
HRESULT get_SegmentFixedPower(long index, tagConverterFrequencyRange range, double *value);
HRESULT put_SegmentFixedPower(long index, tagConverterFrequencyRange value, double value);
```

#### Interface
IConverter5

---

Last Modified:

26-Oct-2010  New command (A.09.33)
SegmentIFBandwidth Property

**Description**
Sets and returns the IF Bandwidth for the sweep segment. Send **Apply** before sending a query (read). Learn more.

**VB Syntax**
```vbnet
conv.SegmentIFBandwidth(index, value)
```

**Variable** *(Type) - Description*

- `conv` A **Converter Object**
- `index` (Long integer) Segment for which IF Bandwidth is to be set. Choose a segment between 1 and the current segment count. Use **SegmentCount Property** to read the current count in the **Applied Mixer**.
- `value` (Long integer) IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. See the lists. If an invalid number is specified, the analyzer will round up to the closest valid number.

**Return Type** Long integer

**Default** 100 kHz

**Examples**
```vba
mxr.SegmentIFBandwidth(1,3)=10e3 'Sets IFBW to 10 kHz
```

**C++ Syntax**
```
HRESULT get_SegmentIFBandwidth(long index, long * val);
HRESULT put_SegmentIFBandwidth(long index, long val);
```

**Interface** IConverter5

---

Last Modified:

26-Oct-2010 New command (A.09.33)
**SegmentIsInputGreaterThanLO Property**

**Description**  
Set and read whether to use the Input frequency that is greater than the LO or less than the LO.

Send **Apply** before sending a query (read). [Learn more](#).

**VB Syntax**  
`conv.SegmentIsInputGreaterThanLO(index, range) = value`

**Variable**  
**Variable (Type) - Description**

- **conv**  
  A **Converter Object**

- **index**  
  (Long integer) Segment which is being set. Choose a segment between 1 and the current segment count. Use **SegmentCount Property** to read the current count in the Applied Mixer.

- **range**  
  (Enum as `ConverterFrequencyRange`) Range for which value is being set. Choose from:
  - 0 - `naInputFrequencies` - set input power
  - 1 - `naOutputFrequencies` - set output power
  - 2 - `naLO1Frequencies` - set LO1 power
  - 3 - `naLO2Frequencies` - set LO2 power
  - 4 - `naIFrequencies` - set IF power

- **value**  
  (Boolean) - Choose from the following:
  - **True** - Use the Input that is Greater than the specified LO.
  - **False** - Use the Input that is Less than the specified LO.

**Return Type**  
Boolean

**Default**  
True

**Examples**  
```vbnet
mxr.SegmentIsInputGreaterThanLO(1, 2=1 'sets segment 1, LO1 to Input greater
```

See example program

**C++ Syntax**  
```c++
HRESULT get_SegmentIsInputGreaterThanLO(long index,  
tagConverterFrequencyRange range, VARIANT_BOOL *value);

HRESULT put_SegmentIsInputGreaterThanLO(long index,  
tagConverterFrequencyRange value, VARIANT_BOOL value);
```

**Interface**  
IConverter5
**SegmentMixingMode Property**

**Description**
Set and read whether the mixing mode (high side or low side) for the mixer segment. Send **Apply** before sending a query (read). [Learn more](#).

**VB Syntax**
```vbnet
conv.SegmentMixingMode(index, range) = value
```

**Variable**
- **(Type) - Description**
  - **conv** A [Converter Object](#)
  - **index** (Long integer) Segment for which mixing mode being set. Choose a segment between 1 and the current segment count. Use **SegmentCount Property** to read the current count in the [Applied Mixer](#).
  - **range** (Enum as ConverterFrequencyRange) Range for which mixing mode is being set. Choose from:
    - **1 - naOutputFrequencies** - sets output frequencies for 1-stage or 2-stage mixers.
    - **4 - naIFFrequencies** - sets IF frequencies for the first LO in 2-stage mixers.
  - **value** (Enum as ConverterSideBand) Choose from:
    - **0 - naLowSide** Input minus LO
    - **1 - naHighSide** Input plus LO

**Return Type**
Enum as ConverterSideBand

**Default**
0 - **naLowSide**

**Examples**
```vbnet
mxr.SegmentMixingMode(1,1)=1 'sets segment 1 output frequencies to Highside mixing.
```

**C++ Syntax**
```cpp
HRESULT get_SegmentMixingMode(long index, tagConverterFrequencyRange range, tagConverterSideBand *value);
HRESULT put_SegmentMixingMode(long index, tagConverterFrequencyRange value, tagConverterSideBand value);
```

**Interface**
IConverter5

---

**Last Modified:**
26-Oct-2010   New command (A.09.33)
SegmentPoints Property

**Description**  
Sets and returns the number of data points to be measured in the sweep segment.  
Send **Apply** before sending a query (read). [Learn more](#).

**VB Syntax**  
`conv.SegmentPoints(index) = value`

**Variable**  
* **(Type)** - **Description**

* `conv`  
  A [Converter Object](#)

* `index`  
  (Long integer) Segment for which points is to be set. Choose a segment between 1 and the current segment count. Use [SegmentCount Property](#) to read the current count in the [Applied Mixer](#).

* `value`  
  (Long integer) - Choose a value between 1 and the [maximum number of data points allowed in the PNA](#). This is also the total number of points allowed for ALL segments.

**Return Type**  
Long integer

**Default**  
21

**Examples**  
```vbnet
mxr.SegmentPoints(1)=3 'Sets 3 points for segment 1.
```

See example program

**C++ Syntax**  
```cpp
HRESULT get_SegmentPoints(long index, long * val);
HRESULT put_SegmentPoints(long index, long val);
```

**Interface**  
IConverter5

---

Last Modified:

26-Oct-2010  
New command (A.09.33)
**SegmentRangeMode Property**

**Description**
Sets or returns the segment sweep mode (Swept or Fixed) for the specified range (Input/LO/Output).

Send **Apply** before sending a query (read). Learn more.

**VB Syntax**
```
conv.SegmentRangeMode(index,range) = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>conv</td>
<td>A Converter Object</td>
</tr>
<tr>
<td>index</td>
<td>(Long integer) Segment for which mixing mode being set. Choose a segment between 1 and the current segment count. Use <strong>SegmentCount Property</strong> to read the current count in the <strong>Applied Mixer</strong>.</td>
</tr>
<tr>
<td>range</td>
<td>(Enum as ConverterFrequencyRange) Range for which sweep mode is being set. Choose from: 0 - <strong>naInputFrequencies</strong> - set input sweep mode. 1 - <strong>naOutputFrequencies</strong> - set output sweep mode. 2 - <strong>naLO1Frequencies</strong> - set LO1 sweep mode. 3 - <strong>naLO2Frequencies</strong> - set LO2 sweep mode. 4 - <strong>naIFFrequencies</strong> - set IF sweep mode.</td>
</tr>
<tr>
<td>value</td>
<td>(Enum as NARangeMode) Choose from: 0 - <strong>naSwept</strong> Range is swept 1 - <strong>naFixed</strong> Range is fixed.</td>
</tr>
</tbody>
</table>

**Return Type**
Enum as NARangeMode

**Default**
Input and Output - **naSwept**
LO (1 and 2) - **naFixed**

**Examples**
```
mxr.SegmentRangeMode(1,1)=0  'sets segment 1 output range to swept.
```

**C++ Syntax**
```
HRESULT get_SegmentRangeMode(long index, tagConverterFrequencyRange range, tagNARangeMode *value);
HRESULT put_SegmentRangeMode(long index, tagConverterFrequencyRange value, tagNARangeMode value);
```
Interface  IConverter5

Last Modified:

26-Oct-2010  New command (A.09.33)
## SegmentStartFrequency Property

**Description**  
Set and read the start frequency for the mixer segment. The specified `SegmentRangeMode` must be set to Swept. Send `Apply` before sending a query (read). Learn more.

**VB Syntax**  
`conv.SegmentStartFrequency(index,range) = value`

**Variable**  

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>conv</code></td>
<td>A Converter Object</td>
</tr>
<tr>
<td><code>index</code></td>
<td>(Long integer) Segment for which start frequency is being set. Choose a segment between 1 and the current segment count. Use <code>SegmentCount Property</code> to read the current count in the Applied Mixer.</td>
</tr>
</tbody>
</table>
| `range` | (Enum as ConverterFrequencyRange) Range for which start frequency is being set. Choose from:  
  
  0 - `naInputFrequencies` - set input frequency  
  1 - `naOutputFrequencies` - set output frequency  
  2 - `naLO1Frequencies` - set LO1 frequency  
  3 - `naLO2Frequencies` - set LO2 frequency  
  4 - `naIFrequencies` - set IF frequency |
| `value` | (Double) Start frequency. Choose a value within the frequency range of the PNA. |

**Return Type**  
Double

**Default**  
Start frequency of the PNA

**Examples**  
```vbnet
mxr.SegmentStartFrequency(1,0)=1e9 'sets the input start frequency to 1 GHz
```

See example program

**C++ Syntax**  
```c++
HRESULT get_SegmentStartFrequency(long index, tagConverterFrequencyRange range, double *value);
HRESULT put_SegmentStartFrequency(long index, tagConverterFrequencyRange value, double value);
```

**Interface**  
IConverter5

---

Last Modified:
26-Oct-2010   New command (A.09.33)
**SegmentState Property**

**Description**  
Sets and returns the ON/OFF state of a sweep segment. Off segments are not included if a segment sweep.

Send **Apply** before sending a query (read). Learn more.

**VB Syntax**  
`conv.SegmentState(index) = state`

**Variable**  
**Type** - Description

- `conv`  
  A **Converter Object**

- `index`  
  (Long integer) Segment to set ON or OFF. Choose a segment between 1 and the current segment count. Use **SegmentCount Property** to read the current count in the **Applied Mixer**.

- `state`  
  (Boolean) - Choose from:
  - **True** - Segment ON
  - **False** - Segment OFF

**Return Type**  
Boolean

**Default**  
ON when added.

**Examples**

```vbnet
mxr.SegmentState(1)=False 'Turns segment 1 OFF.
```

See example program

**C++ Syntax**

```cpp
HRESULT get_SegmentState(long index, VARIANT_BOOL * val);
HRESULT put_SegmentState(long index, VARIANT_BOOL val);
```

**Interface**  
IConverter5

---

**Last Modified:**

26-Oct-2010  
New command (A.09.33)
**SegmentStopFrequency Property**

**Description**
Set and read the Stop frequency for the mixer segment. The specified `SegmentRangeMode` must be set to Swept.

Send **Apply** before sending a query (read). [Learn more.](#)

**VB Syntax**

```vbnet
conv.SegmentStopFrequency(index,range) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>conv</code></td>
<td>A <strong>Converter Object</strong></td>
</tr>
<tr>
<td><code>index</code></td>
<td>(Long integer) Segment for which Stop frequency is being set. Choose a segment between 1 and the current segment count. Use <code>SegmentCount Property</code> to read the current count in the <strong>Applied Mixer</strong>.</td>
</tr>
<tr>
<td><code>range</code></td>
<td>(Enum as ConverterFrequencyRange) Range for which Stop frequency is being set. Choose from:</td>
</tr>
<tr>
<td>0</td>
<td>naInputFrequencies - set input frequency</td>
</tr>
<tr>
<td>1</td>
<td>naOutputFrequencies - set output frequency</td>
</tr>
<tr>
<td>2</td>
<td>naLO1Frequencies - set LO1 frequency</td>
</tr>
<tr>
<td>3</td>
<td>naLO2Frequencies - set LO2 frequency</td>
</tr>
<tr>
<td>4</td>
<td>naIFFrequencies - set IF frequency</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Stop frequency. Choose a value within the frequency range of the PNA.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
Stop frequency of the PNA

**Examples**

```vbnet
mxr.SegmentStopFrequency(1,0)=1e9 'sets the input Stop frequency to 1 GHz
```

See example program

**C++ Syntax**

```csharp
HRESULT get_SegmentStopFrequency(long index, tagConverterFrequencyRange range, double *value);
HRESULT put_SegmentStopFrequency(long index, tagConverterFrequencyRange value, double value);
```

**Interface**

| `IConverter5` |

---

Last Modified:
26-Oct-2010  New command (A.09.33)
SelectPort Property

**Description**
Sets and returns a port mapping for a single port. If this command creates a conflict with an existing port, the PNA will resolve the conflict.

**Note:** This command is currently not supported for the Z5623AK44.

**VB Syntax**
```vbnet
tset.SelectPort(chNum, portNum) = portValue
```

**Variable**
- **Type** - Description
  - `tset` A `TestsetControl` object.
  - `chNum` *(Long)* Channel number of the measurement.
  - `portNum` *(Long)* Physical port number to map.
  - `portValue` *(Long)* Logical port value to assign

**Return Type**
Long

**Default**
Not Applicable

**Examples**
See External Testset Program

**C++ Syntax**
```c++
HRESULT get_SelectPort(long channelNum, long PortNum long *outPort);
HRESULT put_SelectPort(long channelNum, long PortNum long outPort);
```

**Interface**
`ITestsetControl`
SensorIndex Property

**Description**
For dual sensor power meters, sets and returns the power sensor channel (1 or 2) to be used.

**VB Syntax**
```vbnet
pwrSensor.SensorIndex = value
```

**Variable**

- **pwrSensor** (Type): PowerSensorAsReceiver (Object)
- **value** (Long) - Power Meter channel. Choose from:
  1 - Sensor A
  2 - Sensor B

**Return Type**
Long

**Default**
1

**Examples**
```vbnet
pwrSensor.SensorIndex = False 'Write
sensor = pwrSensor.SensorIndex 'Read
```

**C++ Syntax**
```c++
HRESULT put_SensorIndex(long newVal);
HRESULT get_SensorIndex(long* pVal);
```

**Interface**
IPowerSensorAsReceiver

---

**Last Modified:**
25-Aug-2009  MX New topic
Write/Read

SeparatePowerCal Property

**Description**
Specifies whether to use a Thru standard or to use two power sensor connections during the power cal of an SMC calibration. [Learn more.](#)

This command must be sent immediately after the Initialize command, but before all other calibration properties.

**VB Syntax**
```
smc.SeparatePowerCal = bool
```

**Variable**
(Type) - Description

smc An SMCType (object)

bool (Boolean)

**True** - Do NOT use a Thru, but instead perform separate power cals on Input and Output reference planes.

**False** - Perform Cal with Thru standard.

**Return Type**
Boolean

**Default**
False

**Example**
```
FCAppLib.ISMCType4 SMC =
  (FCAppLib.ISMCType4)CalMgr.CreateCustomCal("SMC");
SMC.Initialize(chan, true);
if (separatePowerCalIsDesired)
  SMC.SeparatePowerCal = true;
```

**C++ Syntax**
```
HRESULT put_SeparatePowerCal(VARIANT_BOOL bValue);
HRESULT get_SeparatePowerCal(VARIANT_BOOL *bValue);
```

**Interface**
SMCType4

---

Last Modified:

16-Sep-2009   MX New topic
**ShowStatistics Property**

**Description**
Displays and hides the measurement (Trace) statistics (peak-to-peak, mean, standard deviation) on the screen. To display measurement statistics for a narrower band of the X-axis, use `StatisticsRange`.

The analyzer will display either measurement statistics or Filter Bandwidth statistics; not both.

**VB Syntax**

```
meas.ShowStatistics = value
```

**Variable**

- **meas**
  - Type: A Measurement *(object)*
- **value**
  - Type: Boolean value:
    - True - Show statistics
    - False - Hide statistics

**Return Type**

Boolean

**Default**

False

**Examples**

```
meas.ShowStatistics = True 'Write
showstats = meas.ShowStatistics 'Read
```

**C++ Syntax**

```
HRESULT put_ShowStatistics(VARIANT_BOOL bState)
```

**Interface**

IMeasurement
### ShowProperties Property

**Description**  
Turns ON and OFF the display of the test set control status bar. This status bar indicates the test set that is being controlled and the current port mappings. This setting is turned ON and OFF automatically when the test set is enabled or disabled.

**VB Syntax**  
`tset.ShowProperties = value`

**Variable**  
(***Type***) - Description

- `tset`  
  A `TestsetControl` object.  
  OR  
  An `E5091Testset` object.

- `value`  
  (***Boolean***)
  
  - **True** - Turns display of testset properties ON.  
  - **False** - Turns display of testset properties OFF.

**Return Type**  
Boolean

**Default**  
**False** (True when test set control is enabled.)

**Examples**  
See E5091A Example Program
See External Testset Program

**C++ Syntax**  
`HRESULT get_ShowProperties(VARIANT_BOOL *state);`
`HRESULT put_ShowProperties(VARIANT_BOOL state);`

**Interface**  
IE5091Testsets  
ITestsetControl
## SICL Property

### Description
Allows you to control the PNA via SICL (standard instrument control library). In this mode, the analyzer can receive SCPI commands from the LAN interface or from a program residing on the PNA itself. This command performs the same function as the SICL / GPIB dialog box - SICL Enabled checkbox. See Configuring the analyzer for SICL/VISA.

With this method you can augment a test program written using SICL that resides on the PNA so that it will run unattended. An automation script can be written to start the PNA, enable SICL (using the SICL property), and then start the SICL based program.

### VB Syntax

```
app.SICL value
```

### Variable (Type) - Description

- **app**: An Application (object)
- **value**: (Boolean) Choose from:
  - True - enable SICL
  - False - disable SICL

### Return Type
Boolean

### Default
False

### Examples

```vbnet
Dim Pna as AgilentPNA835x.Application
Dim siclState as Boolean
Set Pna = CreateObject("AgilentPNA835x.Application")
Pna.SICL = true 'write

siclState = Pna.SICL 'Read
```

### C++ Syntax

```cpp
HRESULT get_SICL(VARIANT_BOOL *pVal)
HRESULT put_SICL(VARIANT_BOOL newVal)
```

### Interface
IAgilentPNA835x.Application5
**SICLAddress Property**

**Description**
Sets and returns the PNA SICL address. This is the address used for SICL over LAN.

**VB Syntax**

```vbnet
app.SICLAddress = value
```

**Variable (Type) - Description**

- **app**
  - An Application (object)

- **value**
  - (Integer) SICL Address of the PNA. Choose a value between 0 and 30.

**Return Type**
Short Integer

**Default**
16

**Examples**

```vbnet
address=app.SICLAddress 'Read

app.SICLAddress=16 'Write
```

**C++ Syntax**

```cpp
HRESULT get_SICLAddress(short busIndex, short* address);
HRESULT put_SICLAddress(short busIndex,short address);
```

**Interface**
IApplication8
Simultaneous2PortAcquisition Property

**Description**
Specifies whether a 2-port calibration will be done with a single set of standards (one port at a time) or with two sets of standards (simultaneously).

The AcquireCalStandard2 command uses the same standard index for each calibration class. To specify the calibration standard gender for each port, you must first ensure that the order of calibration class accurately reflects the configuration of your DUT. For example, for a DUT with a male connector on port 1 and a female connector on port 2, order the devices within the S11 classes (A, B, and C) such that the MALE standards are first in the list. Then order the S22 classes specifying the FEMALE standards as the first in the list.

**VB Syntax**
```
cal.Simultaneous2PortAcquisition = state
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>cal</td>
<td>A Calibrator (object)</td>
</tr>
<tr>
<td>state</td>
<td>(boolean) - Choose from:</td>
</tr>
<tr>
<td></td>
<td>True - measures 2 ports simultaneously</td>
</tr>
<tr>
<td></td>
<td>False - measures 1 port at a time</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
True

**Examples**
```
cal.Simultaneous2PortAcquisition = True
```

**C++ Syntax**
```
HRESULT put_Simultaneous2PortAcquisition( VARIANT_BOOL bTwoSetsOfStandards)
HRESULT Simultaneous2PortAcquisition( VARIANT_BOOL *bTwoSetsOfStandards)
```

**Interface**
ICalibrator

Last modified:

9/20/06 Changed default to True

9/12/06 Modified for cross-browser
SmartSweepMaximumIterations Property

### Description
Set and read the maximum permitted number of iterations which SMART Sweep may utilize to find the desired compression level, to within the specified tolerance.

### VB Syntax
\[
gca\text{.SmartSweepMaximumIterations} = value
\]

### Variable (Type) - Description
- **gca** A GainCompression (object)
- **value** (integer) - Maximum number of iterations. Choose a value between 1 and 50.

### Return Type
Integer

### Default
20

### Examples
\[
\text{gca.SmartSweepMaximumIterations} = 10 \quad \text{Write}
\]
\[
\text{iters} = \text{gca.SmartSweepMaximumIterations} \quad \text{Read}
\]

### C++ Syntax
- HRESULT get_SmartSweepMaximumIterations(int* pVal)
- HRESULT put_SmartSweepMaximumIterations(int newVal)

### Interface
IGainCompression

---

**Last Modified:**

11-Sep-2007   MX New topic
### SmartSweepSettlingTime Property

**Description**
Set and read the amount of time SMART Sweep will dwell at the first point where the input power changes by the Backoff or X level.

[Learn more.](#)

**VB Syntax**

```vbnet
gca.SmartSweepSettlingTime = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>gca</code></td>
<td>A <a href="#">GainCompression</a> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) - Settling time in seconds. Choose any positive value.</td>
</tr>
</tbody>
</table>

**Return Type**

Double

**Default**

0

**Examples**

```vbnet
gca.SmartSweepSettlingTime = .01 'Write
sTime = gca.SmartSweepSettlingTime 'Read
```

**C++ Syntax**

```
HRESULT get_SmartSweepSettlingTime(double* pVal)
HRESULT put_SmartSweepSettlingTime(double newVal)
```

**Interface**

IGainCompression

---

**Last Modified:**

21-Nov-2007    MX New topic
**SmartSweepShowIterations Property**

**Description**  Set and read whether to show intermediate results for each iteration in SMART sweep.

**VB Syntax**  
\[ gca.SmartSweepShowIterations = value \]

**Variable (Type) - Description**
- \( gca \)  A \GainCompression\ (object)
- \( value \)  (Boolean) Choose from:
  - **True**  Compression traces are updated after each iteration.
  - **False**  Compression traces are updated after ALL iterations are complete.

**Return Type**  Boolean

**Default**  False

**Examples**
- \( gca.SmartSweepShowIterations = \text{True} \) 'Write
- \( SShow = gca.SmartSweepShowIterations \) 'Read

**C++ Syntax**
- \( \text{HRESULT get\_SmartSweepShowIterations(VARIANT\_BOOL } *\text{pVal)} \)
- \( \text{HRESULT put\_SmartSweepShowIterations(VARIANT\_BOOL } \text{newVal)} \)

**Interface**  IGainCompression

---

Last Modified:

- 21-Nov-2007  MX New topic
### SmartSweepTolerance Property

**Description**  
Set and read the acceptable range SMART Sweep will allow for the measured compression level.

**VB Syntax**  
gca.SmartSweepTolerance = value

**Variable**  

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gca</td>
<td>A GainCompression (object)</td>
</tr>
<tr>
<td>value</td>
<td>(double) - Tolerance level in dB. Choose a value between .01 and 10</td>
</tr>
</tbody>
</table>

**Return Type**  
Double

**Default**  
.05

**Examples**  
gca.SmartSweepTolerance = .01 'Write

tol = gca.SmartSweepTolerance 'Read

**C++ Syntax**  
HRESULT get_SmartSweepTolerance(double* pVal)

HRESULT put_SmartSweepTolerance(double newVal)

**Interface**  
IGainCompression
**SmoothingAperture Property**

**Description**
Specifies or returns the amount of smoothing as a ratio of the number of data points in the measurement trace.

There is no COM command for specifying smoothing by number of aperture points.

**VB Syntax**
`meas.SmoothingAperture = value`

**Variable**

- **meas** (Type) *A Measurement* (object)
- **value** (double) - Smoothing Aperture. A ratio of (aperture points / trace points). Choose any number between .01 and .25.

**Return Type**
Double

**Default**
.25

**Examples**

```vbnet
meas.SmoothingAperture = .10 'Write
saperture = meas.SmoothingAperture 'Read
```

**C++ Syntax**

```csharp
HRESULT get_SmoothingAperture(double *pVal)
HRESULT put_SmoothingAperture(double newVal)
```

**Interface**
IMeasurement

---

Last modified:

Oct. 25, 2006  Fixed formula for smoothing
Smoothing Property

Description: Turns ON and OFF data smoothing.

VB Syntax: `meas.Smoothing = state`

Variable (Type) - Description:
- `meas`: A Measurement (object)
- `state`: (boolean)
  - `True`: Turns smoothing ON
  - `False`: Turns smoothing OFF

Return Type: Boolean

Default: False

Examples:
- `meas.Smoothing = False` 'Write
- `smooth = meas.Smoothing` 'Read

C++ Syntax:
- `HRESULT get_Smoothing(VARIANT_BOOL *pVal)`
- `HRESULT put_Smoothing(VARIANT_BOOL newVal)`

Interface: IMeasurement
SnPFormat Property

**Description**
Specifies the format of .SnP files.

Use either app. `Save` (saves data to file) or meas. `Get_SnpDataWithSpecifiedPorts` (reads data into variant array).

**VB Syntax**
```vbnet
pref.SnPFormat = value
```

**Variable (Type) - Description**

`pref` A `Preferences` (object)

`value` (string) - Format of the .S1P, .S2P, .S3P, .S4P data. Choose from:

- "MA" - Linear Magnitude / degrees
- "DB" - Log Mag / degrees
- "RI" - Real / Imaginary
- "Auto" - Format in which the trace is already displayed. If other than Log Mag, Linear Magnitude, or Real/Imag, then the format will be in Real/Imag.

**Return Type**
String

**Default**
"Auto"

**Examples**
```vbnet
pref.SnPFormat = "MA" 'Write
format = pref.SnPFormat 'Read
```

**C++ Syntax**

- HRESULT get_SnPFormat(BSTR *Format)
- HRESULT put_SnPFormat(BSTR Format)

**Interface**
IPreferences

---

**Last Modified:**

13-Jun-2011 Updated list to get method
**SoundOnFail Property**

**Description**
Turns ON or OFF the audio indicator for limit failures.

**VB Syntax**

\[
\text{limitst.\textbf{SoundOnFail} = state}
\]

**Variable**

- **Type**: Description
- **state** (boolean)
  - **False** - Turns the sound OFF
  - **True** - Turns the sound ON

**Return Type**
Long Integer

**Default**
True

**Examples**

- `Limittest.SoundOnFail = False`  
  **Write**

- `sound = Limittest.SoundOnFail`  
  **Read**

**C++ Syntax**

- `HRESULT get_SoundOnFail(VARIANT_BOOL *pVal)`
- `HRESULT put_SoundOnFail(VARIANT_BOOL newVal)`

**Interface**
ILimitTest
SourceCount Property

Description
Returns the number of sources in the remote PNA.

VB Syntax

\nvalue = cap.SourceCount
\n
Variable (Type) - Description

value (Long) - Variable to store the returned number of sources.

cap A Capabilities (object)

Return Type
Long

Default
Not Applicable

Examples

\nvalue = cap.SourceCount 'Read
\n
C++ Syntax

HRESULT get_SourceCount(long * sourceCount);

Interface
ICapabilities
**SourcePort Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the source port of measurement. To understand how this property is useful, see <a href="#">IMeasurement2 Interface</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>value = meas.SourcePort</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(Long)</em> - Variable to store the returned value</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Long Integer</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>sp = meas.SourcePort</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SourcePort( [out, retval] Long* srcPort);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMeasurement2</td>
</tr>
</tbody>
</table>
SourcePortCount Property

**Description**
Returns the number of ports that can output a signal.
To learn more, see [Remotely Specifying a Source Port](#).

**VB Syntax**
```vbnet
typevalue = object.SourcePortCount
```

**Variable**
- **Type** - Description
  - `value` (Long) - Variable to store the returned integer value of the number of source ports.
  - `object` A [Channel](#) object - always more complete than capabilities object.
  - A [Capabilities](#) object - use when a channel is not available, or to find the common ports across all channels.

**Return Type**
Long

**Default**
Not Applicable

**Examples**
```vbnet
typevalue = chan.SourcePortCount 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SourcePortCount(long * count);
```

**Interface**
- IChannel13
- ICapabilities4

---

**Last Modified:**
- 23-May-2008 Added channel object
- 14-Jan-2007 MX New topic
SourcePortMode Property

**Description**
Sets the state of the PNA source for the specified port.

**VB Syntax**
`chan.SourcePortMode(sourcePort) = value`

**Variable**
- **(Type)** - **Description**
  - `chan` (object) - A `Channel` object
  - `sourcePort` (long integer) - The source port for which to make this setting.
  
  **Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- `value` (enum) - State of the source. Choose from:
  - 0 - `naSourcePortAuto` Port power is turned on when required for a measurement.
  - 1 - `naSourcePortOn` Port power is always ON, regardless of the measurement.
  - 2 - `naSourcePortOff` Port power is always OFF, regardless of the measurement.

**Return Type**
Enum

**Default**
0 - `naSourcePortAuto`

**Examples**
- `chan.SourcePortMode(1) = naSourcePortOn`  
  - "Write"
- `state = chan.SourcePortMode(4)`  
  - "Read"

**C++ Syntax**

```
HRESULT get_SourcePortMode(long sourcePort, enum NASourcePortMode*);
HRESULT put_SourcePortMode(long sourcePort, enum NASourcePortMode);
```

**Interface**
IChannel9

---

**Last modified:**
- 10-Mar-2011   Removed ON/OFF restrictions
- 24-Apr-2008   Added note for strings
- 30-Apr-2007   Edited for src strings
- 10/18/06      MQQ New topic
SourcePortNames Property

Description
Returns the string names of ports that can output a signal.
The following is a list of string names for the PNA-X. Your PNA will NOT have all of these ports. Use GetPortNumber Method to return the correct port number for the specified port name.

- "Port 1"
- "Port 2"
- "Port 3"
- "Port 4"
- "Src2 Out1"
- "Src2 Out2"
- "Port 1 Src2"

For iTMSA (Opt 460)

- "Bal Port 1"
- "Bal Port 2"
- "SE Port1"
- "SE Port 2"

This command also lists the External Sources that are currently configured and selected.
To learn more, see Remotely Specifying a Source Port.

VB Syntax
`value = object.SourcePortNames`

Variable (Type) - Description

`value` (Variant array) - Variable to store the returned integer value of the number of source ports.

`object` A Channel (object) - always more complete than capabilities object.
A Capabilities (object) - use when a channel is not available, or to find the common ports across all channels.

Return Type
Variant array of string names.

Default
Not Applicable

Examples
`value = chan.SourcePortNames` 'Read
**C++ Syntax**

HRESULT get_SourcePortNames(VARIANT *names);

**Interface**

IChannel13
ICapabilities4

---

Last Modified:

3-Mar-2009  Added SE Port 2
23-May-2008  Added iTMSA sources and channel object
23-Jul-2007  Clarification
14-Jan-2007  MX New topic
### Write/Read

<table>
<thead>
<tr>
<th>Source Property</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Description</strong></td>
</tr>
<tr>
<td><strong>VB Syntax</strong></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
</tr>
<tr>
<td><code>trigSetup</code></td>
</tr>
<tr>
<td><strong>value</strong></td>
</tr>
<tr>
<td>0 - <code>naTriggerSourceInternal</code> - free run</td>
</tr>
<tr>
<td>2 - <code>naTriggerSourceExternal</code> - a trigger signal is generated when a trigger signal is sensed on the external trigger pin of the Aux IO connector. Use <code>ExternalTriggerConnectionBehavior</code> to configure the characteristics of the external trigger signal.</td>
</tr>
<tr>
<td>This setting has implications on Calibration. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
</tr>
<tr>
<td><strong>Default</strong></td>
</tr>
<tr>
<td><strong>Examples</strong></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
</tr>
</tbody>
</table>
SourcePowerCalPowerOffset Property

Description
Sets or returns a power level offset from the PNA test port power. This can be a gain or loss value (in dB) to account for components you connect between the source and the reference plane of your measurement. For example, specify 10 dB to account for a 10 dB amplifier at the input of your DUT.

Cal power is the sum of the test port power setting and this offset value. Following the calibration, the PNA power readouts are adjusted to the cal power.

This property performs the same function as the power offset argument on SetCallInfoEx Method, except that this property can read the offset value.

VB Syntax

`chan.SourcePowerCalPowerOffset(sourcePort) = value`

Variable (Type) - Description

`chan` (object) - A Channel object

`sourcePort` (long integer) - The source port for which to set this power offset value.

`value` (double) - Gain or loss value in dB. Choose a value between -200 and 200.

Return Type
Double

Default
0 dB

Examples

`chan.SourcePowerCalPowerOffset(1) = 10 'Write`

`offset = chan.SourcePowerCalPowerOffset(2) 'Read`

C++ Syntax

```cpp
HRESULT get_SourcePowerCalPowerOffset(long sourcePort, double *pVal);
HRESULT put_SourcePowerCalPowerOffset(long sourcePort, double newVal);
```

Interface
IChannel4

Last Modified:

1-May-2007 Modified link to EX method.
SourcePowerCorrection Property

Description
Sets source power correction ON or OFF for a specific source port on this channel, or returns the current ON or OFF state of correction for that source port.

VB Syntax
`chan.SourcePowerCorrection (srcPort) = value`

Variable (Type) - Description
- `chan` (object) – A Channel object
- `srcPort` (long integer) – Source port for which to set or return the ON or OFF state of source power correction.

Note: If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

- `value` (boolean)
  - False – Turns source power correction OFF for the source port.
  - True – Turns source power correction ON for the source port.

Return Type
Boolean

Default
False - Source power correction will turn correction ON

Examples
```
chan.SourcePowerCorrection(1) = False 'Write
calOnPort2 = chan.SourcePowerCorrection(2) 'Read
```

C++ Syntax
```
HRESULT put_SourcePowerCorrection(VARIANT_BOOL bState);
HRESULT get_SourcePowerCorrection(VARIANT_BOOL *bState);
```

Interface
IChannel

Last Modified:
- 24-Apr-2008  Added note for string names
- 30-Apr-2007  Modified for src strings
SourcePowerOption Property

**Description**
Enables the source power to be set on individual sweep segments. This property must be set True before `seg.TestPortPower = value` is sent. Otherwise, the test port power command will be ignored.

**VB Syntax**
```
segs.SourcePowerOption = state
```

**Variable**
- **Type** - Description

- `segs` A Segments collection *(object)*
- `state` *(boolean)*
  - **True** - Enables variable TestPortPower to be set segment sweep
  - **False** - Disables variable TestPortPower to be set segment sweep

**Return Type**
Boolean

**Default**
False

**Examples**
```
segs.SourcePowerOption = True  'Write
powerOption = SourcePowerOption  'Read
```

**C++ Syntax**
```
HRESULT get_SourcePowerOption(VARIANT_BOOL *pVal)
HRESULT put_SourcePowerOption(VARIANT_BOOL newVal)
```

**Interface**
ISegments
# SourcePowerState Property

**Description**

Turns Source Power ON and OFF.  
See note about source power state with instrument state save and recall.

**VB Syntax**

```
app.SourcePowerState = state
```

**Variable** *(Type) - Description*

- `app` An Application *(object)*
- `state` *(boolean)*
  - `False` - Turns Source Power OFF
  - `True` - Turns Source Power ON

**Return Type**

Boolean

**Default**

True

**Examples**

```
app.SourcePowerState = True 'Write
pwr = app.SourcePowerState 'Read
```

**C++ Syntax**

```
HRESULT get_SourcePowerState(VARIANT_BOOL *pVal)
HRESULT put_SourcePowerState(VARIANT_BOOL newVal)
```

**Interface**

IApplication
SourcePullForSParameters Property

Description
Enables and disables the use of source pull technique to compute S22 on Noise Figure on Converters. Learn more.

VB Syntax
nfx.SourcePullForSParameters = value

Variable (Type) - Description
nfx A NoiseFigure (object)
value (Boolean) Source pull technique state. Choose from:
    False - Disable use of source pull technique.
    True - Enable use of source pull technique.

Return Type
Boolean

Default
False

Examples
nfx.SourcePullForSParameters = true 'Write
sourcePull = nfx.SourcePullForSParameters 'Read

C++ Syntax
HRESULT get_SourcePullForSParameters (VARIANT_BOOL * Val)
HRESULT put_SourcePullForSParameters (VARIANT_BOOL newVal)

Interface INoiseFigure4

Last Modified:
7-Oct-2009   MX New topic
SourceStepAttenuatorStepSize Property

Description
Returns a value indicating the step size of the source attenuator.

VB Syntax
value = cap.SourceStepAttenuatorStepSize(n)

Variable (Type) - Description
value (Double) - Variable to store the returned value of the attenuator step size.
cap A Capabilities (object)
n (Long) - port number to query for the value of the attenuator step size.

Return Type
Double

Default
Not Applicable

Examples
value = cap.SourceStepAttenuatorStepSize(1)

C++ Syntax
HRESULT get_SourceStepAttenuatorStepSize(long portNumber, double * stepSize);

Interface
ICapabilities5
## Span Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the Span time of either Gating or Time Domain transform windows</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>object.Span = value</code></td>
</tr>
<tr>
<td>Variable (Type) - Description</td>
<td></td>
</tr>
<tr>
<td><code>object</code> (object) As Gating</td>
<td></td>
</tr>
<tr>
<td>or</td>
<td></td>
</tr>
<tr>
<td><code>object</code> (object) As Transform</td>
<td></td>
</tr>
<tr>
<td><code>value</code> (double) - Span time in seconds. Choose any number between: ( 2^{\left(\frac{\text{number of points} - 1}{\text{frequency span}}\right)} ) and 0</td>
<td></td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>20ns</td>
</tr>
<tr>
<td>Examples</td>
<td></td>
</tr>
<tr>
<td><code>Trans.Span = 4.5e-9</code> ( \text{sets the time span of a transform window} ) - Write</td>
<td></td>
</tr>
<tr>
<td><code>Gate.Span = 4.5e-9</code> ( \text{sets the Span time of a gating window} ) - Write</td>
<td></td>
</tr>
<tr>
<td><code>span = Trans.Span</code> ( \text{Read} )</td>
<td></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_Span(double *pVal)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Span(double newVal)</td>
</tr>
<tr>
<td>Interface</td>
<td>ITransform</td>
</tr>
<tr>
<td></td>
<td>IGating</td>
</tr>
</tbody>
</table>
Span Property

**Description**
Returns the stimulus span of the measurement (stop-start data points). To understand how this property is useful, see [IMeasurement2 Interface](#).

**VB Syntax**
```
value = meas.Span
```

**Variable**
- **Type**: Description
- **value** (Double) - Variable to store the returned value.
- **meas** A Measurement (object)

**Return Type**
Double

**Default**
Not Applicable

**Examples**
Print `meas.Span` 'prints the span of the measurement

**C++ Syntax**
```
HRESULT get_Span(double * Val);
```

**Interface**
IMeasurement2
### SpectrumCenterFrequency Property

**Description**
Sets and returns the receiver Center frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when **Sweep Type** = Linear. Otherwise, this setting is ignored.

**VB Syntax**
```vbnet
ims.SpectrumCenterFrequency = value
```

**Variable**
**Type** - Description

- **ims**:
  - An **IMSpectrum** Object

- **value**:
  - (Double) Center frequency in Hz. Choose a frequency within the range of the PNA.

**Return Type**
Double

**Default**
1.0 GHz

**Examples**
```vbnet
ims.SpectrumCenterFrequency = 10e9 'Write

value = ims.SpectrumCenterFrequency 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SpectrumCenterFrequency(double *pVal)
HRESULT put_SpectrumCenterFrequency(double newVal)
```

**Interface**
IMSpectrum

---

**Last Modified:**
19-Aug-2008   MX New topic
SpectrumSpanFrequency Property

Description
Sets and returns the Span of receiver frequencies for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

VB Syntax
\[ ims.SpectrumSpanFrequency = value \]

Variable
\( ims \) An IMSpectrum Object
\( value \) (Double) Frequency span in Hz. Choose a frequency within the range of the PNA.

Return Type
Double

Default
100 MHz

Examples
\[ ims.SpectrumSpanFrequency = 10e9 \] ‘Write
\[ value = ims.SpectrumSpanFrequency \] ‘Read

C++ Syntax
HRESULT get_SpectrumSpanFrequency(double *pVal)
HRESULT put_SpectrumSpanFrequency(double newVal)

Interface
IIMSpectrum

Last Modified:
19-Aug-2008   MX New topic
SpectrumStartFrequency Property

Description
Sets and returns the receiver Start frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

VB Syntax
```vb
ims.SpectrumStartFrequency = value
```

Variable (Type) - Description
- **ims**: An IMSpectrum Object
- **value**: (Double) Start frequency in Hz. Choose a frequency within the range of the PNA.

Return Type
Double

Default
950 MHz

Examples
```vb
ims.SpectrumStartFrequency = 10e9 'Write
value = ims.SpectrumStartFrequency 'Read
```

C++ Syntax
- HRESULT get_SpectrumStartFrequency(double *pVal)
- HRESULT put_SpectrumStartFrequency(double newVal)

Interface
IIMSpectrum

Last Modified:
19-Aug-2008   MX New topic
**SpectrumStopFrequency Property**

**Description**
Sets and returns the receiver Stop frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

**VB Syntax**
```vbnet
ims.SpectrumStopFrequency = value
```

**Variable (Type) - Description**
- `ims`: An `IMSpectrum` Object
- `value`: (Double) Stop frequency in Hz. Choose a frequency within the range of the PNA.

**Return Type**
Double

**Default**
950 MHz

**Examples**
```vbnet
ims.SpectrumStopFrequency = 10e9 'Write
value = ims.SpectrumStopFrequency 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SpectrumStopFrequency(double *pVal)
HRESULT put_SpectrumStopFrequency(double newVal)
```

**Interface**
`IIMSpectrum`

---

Last Modified:
19-Aug-2008     MX New topic
### SSB_BalPortNegative Property

**Description**  
With a Single-ended - Single-ended - Balanced topology, returns the PNA port number that is connected to the Negative side of the DUT's Balanced Port.


**VB Syntax**  
```vbnet
var = balTopology.SSB_BalPortNegative
```

**Variable**  
*(Type)* - Description

```vbnet
var (Long Integer) Variable to store the returned value.
```

**balTopology**  
A [BalancedTopology](#) object

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**  
```vbnet
variable = balTopology.SSB_BalPortNegative 'Read
```

**C++ Syntax**  
```cpp
HRESULT get_SSB_BalPortNegative(long *bVal)
```

**Interface**  
IBalancedTopology
SSB_BalPortPositive Property

**Description**
With a Single-ended - Single-ended - Balanced topology, returns the PNA port number that is connected to the Positive side of the DUT's Balanced Port.


**VB Syntax**
```vbnet
var = balTopology.SSB_BalPortPositive
```

**Variable**

<table>
<thead>
<tr>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>var</td>
</tr>
</tbody>
</table>

(Long Integer) Variable to store the returned value.

**balTopology**
A [BalancedTopology](#) (object)

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```
variable = balTopology.SSB_BalPortPositive   'Read
```

**C++ Syntax**
```
HRESULT get_SSB_BalPortPositive(long *bVal)
```

**Interface**
IBalancedTopology
SSB_SEPort1 Property

**Description**
With a Single-ended - Single-ended - Balanced topology, returns the PNA port number that is connected to the DUT's Logical Port 1.

**VB Syntax**
```
var = balTopology.SSB_SEPort1
```

**Variable**
- **(Type)** - Description
  - `var` (Long Integer) Variable to store the returned value.

**balTopology**
A BalancedTopology (object)

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```
variable = balTopology.SSB_SEPort1  'Read
```

**C++ Syntax**
```
HRESULT get_SSB_SEPort1(long *bVal)
```

**Interface**
IBalancedTopology
**SSB_SEPort2 Property**

**Description**  
With a Single-ended - Single-ended - Balanced topology, returns the PNA port number that is connected to the DUT's Logical Port 2.  

**VB Syntax**  
```vbnet
var = balTopology.SSB_SEPort2
```

**Variable** *(Type) - Description*  
var (Long Integer) Variable to store the returned value.

**balTopology**  
A [BalancedTopology](#) *(object)*

**Return Type**  
Long Integer

**Default**  
Not Applicable

**Examples**  
```vbnet
variable = balTopology.SSB_SEPort2 'Read
```

**C++ Syntax**  
HRESULT get_SSB_SEPort2(long *bVal)

**Interface**  
IBalancedTopology
SSBMeasurement Property

**Description**  

**VB Syntax**  
`balMeas.SSBMeasurement = value`

**Variable (Type) - Description**

- **balMeas**  
  A `BalancedMeasurement` (object)

- **value**  
  (String) - Single-ended - Single-ended - Balanced Measurement parameter. Not case sensitive. Choose from:

<table>
<thead>
<tr>
<th>Sss11</th>
<th>Sss12</th>
<th>Ssd13</th>
<th>Ssc13</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sss21</td>
<td>Sss22</td>
<td>Ssd23</td>
<td>Ssc23</td>
</tr>
<tr>
<td>Sds31</td>
<td>Sds32</td>
<td>Sdd33</td>
<td>Sdc33</td>
</tr>
<tr>
<td>Scs31</td>
<td>Scs32</td>
<td>Scd33</td>
<td>Scc33</td>
</tr>
<tr>
<td>Imb1</td>
<td>Imb2</td>
<td>CMRR1</td>
<td>CMRR2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds31/Scs31)</td>
<td>(Sds32/Scs32)</td>
</tr>
</tbody>
</table>

**Return Type**  
String

**Default**  
Sss11

**Examples**  
```
balMeas.SSBMeasurement = "Sss11" 'Write
variable = balMeas.SSBMeasurement  'Read
```

**C++ Syntax**

- `HRESULT get_SSBMeasurement(BSTR *pVal)`
- `HRESULT put_SSBMeasurement(BSTR p newVal)`

**Interface**  
`IBalancedMeasurement`
### Stage1Coefficients Property

**Description**
Sets and returns the digital filter coefficients of stage1.

**VB Syntax**
```vba
spm4.Stage1Coefficients = value
```

**Variable**  
**Type** — Description

- `spm4`: A `SignalProcessingModuleFour (object)`
- `value`: (Variant Array) Coefficients. An array of real values.

**Return Type**
Variant

**Default**
Stage dependent.

**Examples**
```vba
spm4.Stage1Coefficients = 0,0.1,0.7,0.7,0.1 'Write
mode = spm4.Stage1Coefficients 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Stage1Coefficients(VARIANT* pCoefs);
HRESULT put_Stage1Coefficients(VARIANT pCoefs);
```

**Interface**
ISignalProcessingModuleFour

---

Last Modified:

18-Jun-2007  MX New topic
Stage1Frequency Property

Description
Sets and returns the Numerically Controlled Oscillator (NCO) frequency of the Stage 1 filter. This command is only used when FilterMode Property is set to Manual.

VB Syntax

```vbnet
spm4.Stage1Frequency = value
```

Variable (Type) - Description

- `spm4` A SignalProcessingModuleFour (object)
- `value` (Double) Stage 1 Frequency. Min value= 0 Hz
  - Stage 1 Frequency. Min value= 0 Hz
  - With DSP 4 versions, Max value= 15 MHz.
  - With DSP 5 versions, Max value = 38 MHz.
  - Learn more about DSP versions.
  - Or programmatically use MinimumIFFrequency Property and MaximumIFFrequency Property to determine the range of settable values.

Return Type
Double

Default
Nominal IF Frequency. Learn more

Examples

```vbnet
spm4.Stage1Frequency = 9E6 'Write
mode = spm4.Stage1Frequency 'Read
```

C++ Syntax

```c++
HRESULT get_Stage1Frequency(double *val);
HRESULT put_Stage1Frequency(double val);
```

Interface
ISignalProcessingModuleFour

Last Modified:

- 26-Aug-2010 Updated for DSP 5
- 18-Jun-2007 MX New topic
Stage1MaximumCoefficient Property

**Description**
Returns the maximum value of any single coefficient.

**VB Syntax**
```
value = spm4.Stage1MaximumCoefficient
```

**Variable**
- **(Type)**: Description
  - **value**: (Long) Variable to store the returned Max coefficient.
  - **spm4**: A SignalProcessingModuleFour (object)

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage1MaximumCoefficient
```

**C++ Syntax**
```
HRESULT get_Stage1MaximumCoefficient(long* val);
```

**Interface**
ISignalProcessingModuleFour

Last Modified:
1-Jan-2007  MX New topic
### Stage1MaximumCoefficientCount Property

**Description**
Returns the maximum number of coefficients for Stage1.

**VB Syntax**
```
value = spm4.Stage1MaximumCoefficientCount
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Long) Variable to store the returned Max coefficient count.</td>
</tr>
<tr>
<td>spm4</td>
<td>A SignalProcessingModuleFour (object)</td>
</tr>
</tbody>
</table>

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage1MaximumCoefficientCount
' Read
```

**C++ Syntax**
```
HRESULT get_Stage1MaximumCoefficientCount(long* val);
```

**Interface**
ISignalProcessingModuleFour

---

**Last Modified:**
1-Jan-2007  MX New topic
**Stage1MaximumCoefficientSum Property**

**Description**
Returns the maximum sum of all Stage1 coefficients.

**VB Syntax**
```
value = spm4.Stage1MaximumCoefficientSum
```

**Variable**
- **Type**: __int64* val
- **Description**: Variable to store the returned Max sum of all coefficients.

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage1MaximumCoefficientSum 'Read
```

**C++ Syntax**
```
HRESULT get_Stage1MaximumCoefficientSum(__int64* val);
```

**Interface**
ISignalProcessingModuleFour

---

**Last Modified:**
1-Jan-2007  MX New topic
Stage1MinimumCoefficientCount Property

**Description**
Returns the minimum number of coefficients for Stage1

**VB Syntax**
```
value = spm4.Stage1MinimumCoefficientCount
```

**Variable (Type) - Description**
- **value** (Long) Variable to store the returned Min coefficient count.
- **spm4** A SignalProcessingModuleFour (object)

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage1MinimumCoefficientCount
'C++ Syntax
HRESULT get_Stage1MinimumCoefficientCount(long* val);

**Interface**
ISignalProcessingModuleFour
```

Last Modified:
1-Jan-2007 MX New topic
### Stage2Coefficients Property

**Description**  
Sets and returns Stage2Coefficients.  
**Note:** Stage2 settings are ignored when using DSP Version 5. [Learn more.](#)

**VB Syntax**  
```vbnet
spm4.Stage2Coefficients = value
```

**Variable**  
(Type) - Description

- `spm4`  
  A [SignalProcessingModuleFour](#) *(object)*

- `value` *(Variant)*  
  An array of real numbers. Filter coefficients

**Return Type**  
Variant

**Default**  
Not Applicable

**Examples**  
```vbnet
spm4.Stage2Coefficients = 'Write
mode = spm4.Stage2Coefficients 'Read
```

**C++ Syntax**  
```cpp
HRESULT get_Stage2Coefficients(VARIANT* pCoefs);
HRESULT put_Stage2Coefficients(VARIANT pCoefs);
```

**Interface**  
[ISignalProcessingModuleFour](#)

---

Last Modified:

1-Jan-2007   MX New topic
Stage2MaximumCoefficient Property

Description Returns the maximum value of any single coefficient.

VB Syntax  

```vbnet
value = spm4.Stage2MaximumCoefficient
```

Variable (Type) - Description

- `value` (Long) Variable to store the returned Max coefficient.
- `spm4` A SignalProcessingModuleFour (object)

Default Not Applicable

Examples
```
mode = spm4.Stage2MaximumCoefficient
```

C++ Syntax  

```
HRESULT get_Stage2MaximumCoefficient(long* val);
```

Interface ISignalProcessingModuleFour

Last Modified:

1-Jan-2007 MX New topic
## Stage2MaximumCoefficientCount Property

**Description**  
Returns the maximum number of coefficients for Stage2.  
**Note:** Stage2 settings are ignored when using DSP Version 5. [Learn more.](#)

<table>
<thead>
<tr>
<th><strong>VB Syntax</strong></th>
<th>value = spm4.Stage2MaximumCoefficientCount</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td>value</td>
<td><strong>(Long)</strong> Variable to store the returned Max coefficient count.</td>
</tr>
<tr>
<td>spm4</td>
<td>A SignalProcessingModuleFour (object)</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**Examples**  
mode = spm4.Stage2MaximumCoefficientCount 'Read

**C++ Syntax**  
HRESULT get_Stage2MaximumCoefficientCount(long* val);

**Interface**  
ISignalProcessingModuleFour

---

Last Modified:  
1-Jan-2007  
MX New topic
### Stage2MaximumCoefficientSum Property

**Description**
Returns the maximum sum of all Stage2 coefficients.

**Note:** Stage2 settings are ignored when using DSP Version 5. Learn more.

**VB Syntax**
```
value = spm4.Stage2MaximumCoefficientSum
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>val</code></td>
<td>Variable to store the returned Max sum of all coefficients.</td>
</tr>
</tbody>
</table>

| `spm4`   | A SignalProcessingModuleFour (object) |

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage2MaximumCoefficientSum 'Read
```

**C++ Syntax**
```
HRESULT getStage2MaximumCoefficientSum(__int64* val);
```

**Interface**
ISignalProcessingModuleFour

---

Last Modified:

1-Jan-2007  MX New topic
Stage2MinimumCoefficientCount Property

Description
Returns the minimum number of coefficients for Stage2.

Note: Stage2 settings are ignored when using DSP Version 5. Learn more.

VB Syntax
$value = spm4.Stage2MinimumCoefficientCount$

Variable (Type) - Description

$value$ (Long) Variable to store the returned Min coefficient count.

$spm4$ A SignalProcessingModuleFour (object)

Default
Not Applicable

Examples
$mode = spm4.Stage2MinimumCoefficientCount 'Read$

C++ Syntax
HRESULT get_Stage2MinimumCoefficientCount(long* val);

Interface
ISignalProcessingModuleFour

Last Modified:
1-Jan-2007   MX New topic
Write-Read

Stage3FilterType Property

Description  Sets and returns the Stage 3 filter type. This command is only used when FilterMode is set to Manual.

VB Syntax  

```
spm4.Stage3FilterType = value
```  

Variable  

**Type** - Description

- *spm4*  A SignalProcessingModuleFour (object)
- *value*  (String) Filter type. Chose from:
  - "RECT" Rectangular Window Filter
  - "TUKEY" Tukey Filter
  - "PWIN" Pulse window filter

Default  TUKEY

Examples  

```
spm4.Stage3FilterType = "PWIN"
```

```
mode = spm4.Stage3FilterType  'Read
```

C++ Syntax  

```
HRESULT get_Sstage3FilterType(BSTR* pFType);
HRESULT put_Sstage3FilterType(BSTR GType);
```

Interface  

ISignalProcessingModuleFour

Last Modified:

1-Jan-2007  MX New topic
Stage3FilterTypes Property

**Description**
Returns a list of strings for the currently supported filter types that can be used for the stage 3 filter. This command is only used when FilterMode is set to False (Manual). See Stage3FilterType for a list of currently supported filter types.

**VB Syntax**
values = spm4.Stage3FilterTypes

**Variable**
- **value** (Variant) Variable to store the returned filter types.
- **spm4** A SignalProcessingModuleFour (object)

**Return Type**
Variant Array

**Default**
Not Applicable

**Examples**
mode = spm4.Stage3FilterTypes 'Read

**C++ Syntax**
HRESULT get_Stage3FilterTypes(VARIANT* pTypes);

**Interface**
ISignalProcessingModuleFour
Stage3Parameter Property

Description
Sets and returns the Stage 3 filter parameters.
Must first select the filter type using Stage3FilterType before setting these parameters.
Use Stage3Parameters to return a list of the available parameters for the currently selected filter type.

VB Syntax

```
spm4.Stage3Parameter(param) = value
```

Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable (Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>spm4 SignalProcessingModuleFour (object)</td>
<td></td>
</tr>
<tr>
<td>param (String)</td>
<td>Filter parameter. Choose from:</td>
</tr>
<tr>
<td>&quot;C&quot; - Tap count (Tukey, RECT, PWIN)</td>
<td></td>
</tr>
<tr>
<td>&quot;P&quot; - Period (PWIN ONLY)</td>
<td></td>
</tr>
<tr>
<td>&quot;D&quot; - Delay (PWIN ONLY)</td>
<td></td>
</tr>
<tr>
<td>&quot;W&quot; - Width (PWIN ONLY)</td>
<td></td>
</tr>
<tr>
<td>&quot;R&quot; - Ramp Count (PWIN ONLY)</td>
<td></td>
</tr>
</tbody>
</table>

value (String) Parameter Value for the specified stage 3 parameter. Use Stage3ParameterMaximum and Stage3ParameterMinimum to return a range of values for the specified parameter.

Default

RECT: C = 1
PWIN: C=1E6, P=10ms, D=50us, W=50us, R=7
TUKEY: C=1

Examples

```
spm4.Stage3Parameter("C") = 2
```

```
mode = spm4.Stage3Parameter("pwin") 'Read
```

C++ Syntax

```cpp
HRESULT get_Stage3Parameter(BSTR pName, double* pVal);
HRESULT put_Stage3Parameter(BSTR pName, double pVal);
```

Interface

ISignalProcessingModuleFour

Last Modified:

1-Jan-2007  MX New topic
Stage3ParameterMaximum Property

Description  Returns maximum parameter value for the current filter type.

VB Syntax  \( \text{values} = \text{spm4}.\text{Stage3ParameterMaximum}(\text{parameter}) \)

Variable  (Type) - Description

value  (Variant) Variable to store the maximum parameter value.

spm4  A SignalProcessingModuleFour (object)

parameter  (String) Parameter name. See Stage3Parameter Property for a list of parameters.

Return Type  Double

Default  Not Applicable

Examples  \( \text{mode} = \text{spm4}.\text{Stage3ParameterMaximum}("c") \) 'Read

C++ Syntax  HRESULT get_Stage3ParameterMaximum(BSTR pName, double* pVal);

Interface  ISignalProcessingModuleFour

Last Modified:  1-Jan-2007  MX New topic
# Stage3ParameterMinimum Property

**Description**
Returns minimum parameter value for the current filter type.

**VB Syntax**
```vbnet
values = spm4.Stage3ParameterMinimum (parameter)
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(Variant) Variable to store the minimum parameter value.</td>
</tr>
<tr>
<td>spm4</td>
<td>A SignalProcessingModuleFour (object)</td>
</tr>
</tbody>
</table>

**parameter**
(String) Parameter name. See [Stage3Parameter Property](#) for a list of parameters.

**Return Type**
Double

**Default**
Not applicable

**Examples**
```vbnet
mode = spm4.Stage3ParameterMinimum ("c")  'Read
```

**C++ Syntax**
```cpp
HRESULT get_Stage3ParameterMinimum(BSTR pName, double* pVal);
```

**Interface**
ISignalProcessingModuleFour

---

Last Modified:

1-Jan-2007  MX New topic
**Read-only**

### Stage3Parameters Property

**Description**
Returns the names of parameters for the current filter type. Use `Stage3FilterType Property` to set the filter type.

**VB Syntax**
```
values = spm4.Stage3Parameters
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>value</code></td>
<td>(Variant) Variable to store the returned parameter names.</td>
</tr>
<tr>
<td><code>spm4</code></td>
<td>A <code>SignalProcessingModuleFour</code> (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
```
mode = spm4.Stage3Parameters 'Read
```

**C++ Syntax**
```
HRESULT get_Stage3Parameters(VARIANT* pNames);
```

**Interface**
`ISignalProcessingModuleFour`

---

Last Modified:

18-Jan-2007   MX New topic
StandardDeviation Property

**Description**

Returns the standard deviation of the measurement. To retrieve all 3 statistics value at the same time, use `meas.GetTraceStatistics`.

**VB Syntax**

```vbnet
stdev = meas.StandardDeviation
```

**Variable**

- **Type**
  - `stdev` (single) - Variable to store standard deviation value
- **meas** A Measurement (object)

**Return Type**

Single

**Default**

Not applicable

**Examples**

```vbnet
stdev = meas.StandardDeviation 'Read
```

**C++ Syntax**

```cpp
HRESULT get_StandardDeviation(float* stdDeviation)
```

**Interface**

IMeasurement
StandardForClass Property - Superseded

**Description**  
Superseded  This command sets a single standard to a calibration class. Does NOT set or dictate the order for measuring the standards. Use GetStandardForClass and SetStandardForClass. These commands allow up to seven standards to be assigned to a cal class.

**VB Syntax**  
\`calKit.StandardForClass(class, portNum) = value\`

**Variable (Type) - Description**

- `calKit`  
  A CalKit (object). Use calKit.GetCalStandard to get a handle to the standard.

- `class`  
  (enum NACalClass) Standard. Choose from:

  1 - naClassA  
  2 - naClassB  
  3 - naClassC  
  4 - naClassD  
  5 - naClassE  
  6 - naReferenceRatioLine  
  7 - naReferenceRatioThru

**SOLT Standards**

  1 - naSOLT_Open  
  2 - naSOLT_Short  
  3 - naSOLT_Load  
  4 - naSOLT_Thru  
  5 - naSOLT_Isolation

**TRL Standards**

  1 - naTRL_Reflection  
  2 - naTRL_Line_Reflection  
  3 - naTRL_Line_Tracking  
  4 - naTRL_Thru  
  5 - naTRL_Isolation
**portNum** *(long)* - The port number the standard will be connected to. For example, you may have a 3.5mm connector designated for port 1, and Type N designated for port 2.

**value** *(long)* - Calibration class number. Choose a number between 1 and 8. The `<value>` numbers are associated with the following calibration classes:

<table>
<thead>
<tr>
<th><code>&lt;value&gt;</code></th>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>S11A</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>2</td>
<td>S11B</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>3</td>
<td>S11C</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>4</td>
<td>S21T</td>
<td>Thru standard</td>
</tr>
<tr>
<td>5</td>
<td>S22A</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>6</td>
<td>S22B</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>7</td>
<td>S22C</td>
<td>Reflection standard</td>
</tr>
<tr>
<td>8</td>
<td>S21T</td>
<td>Thru standard</td>
</tr>
</tbody>
</table>

**Return Type** Long Integer

**Default** Not Applicable

**Examples**

```c
calKit.StandardForClass(naSOLT_Short, 1) = 1
Kclass = calKit.StandardForClass(naSOLT_Short, 1)
```

**C++ Syntax**

```c
HRESULT put_StandardForClass (NACalClass item, long pNum);
HRESULT get_StandardForClass (NACalClass* item, long *pNum);
```

**Interface** ICalKit
# StartFrequency (Cal Set) Property

**Description**
Returns the start frequency that is stored in the Cal Set.

**VB Syntax**
```vbnet
value = CalSet.StartFrequency(range)
```

**Variable**
- **(Type)** - **Description**
  - `value` *(double)* - returned Start frequency in Hertz.
  - `CalSet` *(object)*
  - `range` *(Long)* - Choose: 0 (Source and receiver frequency)

**Return Type**
Double

**Default**
Not Applicable

**Examples**
```vbnet
start = calset.StartFrequency(0) 'Reads the start frequency stored in the cal set.
```

**C++ Syntax**
```c++
HRESULT get_StartFrequency(long range, *pVal)
```

**Interface**
`ICalSet3`

---

Last modified:
- 25-Jan-2011  Fixed range argument
- Nov. 1, 2006  New command - split from ch.StartFreq
### StartFrequency Property

**Description**
Sets or returns the start frequency of the channel. (Channel Object)
Sets or returns the start frequency of the segment. (Segment Object)
Sets or returns the start frequency of the FOM Range. (FOMRange Object)
Sets or returns the start frequency of the Power Sensor coverage (GuidedCalibrationPowerSensor Object)

See also [Measurement2](#) interface

**VB Syntax**
```
object.StartFrequency = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>Channel (object)</td>
</tr>
<tr>
<td></td>
<td>Segment (object)</td>
</tr>
<tr>
<td></td>
<td>FOMRange (object)</td>
</tr>
<tr>
<td></td>
<td>GuidedCalibrationPowerSensor (object)</td>
</tr>
</tbody>
</table>

| value     | (double) - Start frequency in Hertz. Choose any number between the minimum and maximum frequencies of the analyzer. |

**Default**
Channel - Minimum frequency of the analyzer
Segment - 0
FOMRange - Minimum frequency of the analyzer
PowerSensor - Minimum frequency of the analyzer

**Examples**

```
chan.StartFrequency = 4.5e9 'sets the start frequency of a linear sweep for the channel object -Write
```

```
startfreq = Chan.StartFrequency 'Read
```

**C++ Syntax**
```cpp
HRESULT get_StartFrequency(double *pVal)
HRESULT put_StartFrequency(double newVal)
```

**Interface**
ICHannel
ISegment
IFOMRange
IGuidedCalibrationPowerSensor
Last modified:

8-Feb-2011  Added Power Sensor
8-Mar-2007  Added FOMRange
1-Nov-2006  Removed Cal Set object. There is now a new cs.StartFreq
StartPower Property

Description
Sets the start power of the analyzer when sweep type is set to Power Sweep. Frequency of the measurement is set with chan.CWFrequency.

VB Syntax
object.StartPower = value

Variable (Type) - Description
object One of the following:

- Channel (object)
- CalSet (object) - Read-only property

value (double) - Start Power in dBm.

Note: The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, use cap.MaximumSourceALCPower and cap.MinimumSourceALCPower

Auto attenuation is not allowed in Power Sweep.

Return Type
Double

Default
0

Examples
Chan.StartPower = -10  'Write
strtpwr = Chan.StartPower 'Read

C++ Syntax
HRESULT get_StartPower(double *pVal)
HRESULT put_StartPower(double newVal)

Interface
IChannel
|CalSet3

Last Modified:
7-Jan-2008  Removed FOMRange
8-Mar-2007  Added FOMRange
Start Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the start time of either Gating or Time Domain transform windows</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>object.Start = value</code></td>
</tr>
</tbody>
</table>
| **Variable** | `(object) As Gating  
 or  
 (object) As Transform` |
| **value** | `(double) - Start time in seconds. Choose any number between:  
± (number of points-1) / frequency span` |
| **Return Type** | Double |
| **Default** | -10ns |
| **Examples** | `Trans.Start = 4.5e-9 'sets the start time of a transform window`  
`Gate.Start = 4.5e-9 'sets the start time of a gating window`  
`strt = Trans.Start 'Read` |
| **C++ Syntax** | `HRESULT get_Start(double *pVal)`  
`HRESULT put_Start(double newVal)` |
| **Interface** | ITransform  
IGating |
Read-only

Start Property

**Description**
Returns the stimulus value of the first data point for the measurement. To understand how this property is useful, see [IMeasurement2 Interface](#).

**VB Syntax**
```vbnet
value = meas.Start
```

**Variable (Type) - Description**
- **value** *(Double)* - Variable to store the returned value
- **meas** A Measurement *(object)*

**Return Type**
Double

**Default**
Not Applicable

**Examples**
Print meas.Start 'prints the stimulus value of the first data point

**C++ Syntax**
```cpp
HRESULT get_Start (double * Val);
```

**Interface**
IMeasurement2
Write/Read

About Phase Control

StartPhase Property

**Description**
Write and read the start value of phase sweep. Must also send [Sweep Type Property](#) to put the analyzer into phase sweep mode.

**VB Syntax**

```vbnet
phase.StartPhase(srcPort) = value
```

**Variable** *(Type) - Description*

- **phase**
  A [PhaseControl](#) Object

- **srcPort**
  (Long Integer) Source port for which to make phase control settings.

  **Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#)

- **value**
  (Double) Start phase value in degrees. Choose a value between -360 and 360.

**Return Type**
Double

**Default**
0 degrees

**Examples**

```vbnet
phase.StartPhase 1 = 60 ' Write

value = phase.StartPhase 2' Read
```

**C++ Syntax**

```cpp
HRESULT get_StartPhase(long port, double* pVal);
HRESULT put_StartPhase(long port, double newVal);
```

**Interface**
IPhaseControl

---

Last Modified:

8-Dec-2010   MX New topic
**StartPowerEx Property**

**Description**
Sets and reads the power sweep start power value for a specific port. This allows uncoupled forward and reverse power sweep ranges. Must also set **SweepType** = naPowerSweep, **Coupled** = False (Off), and **StopPowerEx**.

**VB Syntax**

```vbnet
chan.StartPowerEx (srcPort) = value
```

**Variable**

- **(Type)** - **Description**
  - **chan** - A **Channel** (object)
  - **srcPort** - (long integer) – Source port for which to set the Start power value.
    - **Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see **Remotely Specifying a Source Port**.

- **value** - (double) - Start Power in dBm.
  - **Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, use `cap.MaximumSourceALCPower` and `cap.MinimumSourceALCPower`.
  - Auto attenuation is not allowed in Power Sweep.

**Return Type**

- Double

**Default**

- -10 dBm

**Examples**

```vbnet
Chan.StartPowerEx 1 = -10 'Write
strtpwr = Chan.StartPowerEx 2 'Read
```

**C++ Syntax**

- `HRESULT get_StartPowerEx(long port, double *pVal)`
- `HRESULT put_StartPowerEx(long port, double newVal)`

**Interface**

- `IChannel13`

---

**Last Modified:**

- 23-May-2008  MX New topic
**StartRatioedPower Property**

**Description**
Write and read the start power ratioed value. Must also set **SweepType** to **Power**.

**VB Syntax**
```
phase.StartRatioedPower(srcPort) = value
```

**Variable**
(Type) - Description
- **phase**: A **PhaseControl** Object
- **srcPort**: (Long Integer) Source port for which to make phase control settings.

**Note**: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use **chan.getPortNumber** to translate the string into a port number. To learn more see **Remotely Specifying a Source Port**

- **value**: (Double) Start power ratio value in dBc. Must be within the allowable range of the PNA.

**Return Type**
Double

**Default**
0 dBc

**Examples**
```
phase.StartRatioedPower 1 = -1 ' Write
value = phase.StartRatioedPower 2' Read
```

**C++ Syntax**
```
HRESULT get_StartRatioedPower(long port, double* pVal);
HRESULT put_StartRatioedPower(long port, double newVal);
```

**Interface**
IPhaseControl

---

Last Modified:
- 8-Dec-2010  MX New topic
## State (GPL) Property

**Description**
Enables and disables Global Power Limiting for the specified port.

**VB Syntax**
```
gpl.State(port) = bool
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gpl</td>
<td>A</td>
<td>GlobalPowerLimit (object)</td>
</tr>
<tr>
<td>port</td>
<td>Long</td>
<td>Port number for which power limit state is to be set.</td>
</tr>
<tr>
<td>bool</td>
<td>Boolean</td>
<td>Choose from:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>True - Power Limiting is enabled.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>False - Power Limiting is disabled.</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**
```
gpl.State(1) = True  'Write
Limit = gpl.State(2) 'Read
```

**C++ Syntax**
```c++
HRESULT get_State(long port, VARIANT_BOOL *pVal)
HRESULT put_Limit(long port, VARIANT_BOOL  newVal)
```

**Interface**
IGlobalPowerLimit

---

**Last Modified:**
10-Aug-2009  MX New topic
Write/Read

State Property

**Description**
Turns an Object ON and OFF.

**VB Syntax**
```
object.State = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>Applies to any of the following objects:</td>
</tr>
<tr>
<td></td>
<td>FOM</td>
</tr>
<tr>
<td></td>
<td>Gating</td>
</tr>
<tr>
<td></td>
<td>InterfaceControl</td>
</tr>
<tr>
<td></td>
<td>LimitTest</td>
</tr>
<tr>
<td></td>
<td>Port Extension - Superseded (See Fixturing Object)</td>
</tr>
<tr>
<td></td>
<td>Segment</td>
</tr>
<tr>
<td></td>
<td>Transform</td>
</tr>
<tr>
<td></td>
<td>Equation</td>
</tr>
<tr>
<td></td>
<td>FIFO</td>
</tr>
</tbody>
</table>

**Notes:**
- **LimitTest.State** - If using Global Pass/Fail status, trigger the PNA AFTER turning Limit testing ON.
- **Segment.State** - At least ONE segment must be ON or **Sweep Type** is automatically set to **Linear**.

**value**

<table>
<thead>
<tr>
<th>boolean</th>
</tr>
</thead>
<tbody>
<tr>
<td>False</td>
</tr>
<tr>
<td>True</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
Depends on the object:
- 0 - FOM
- 0 - Gating
- 0 - InterfaceControl
- 0 - LimitTest
- 0 - Port Extension
- 1 - Segment
- 0 - Transform
0 - Equation
0 - FIFO

Examples

Seg.State = 1 'Turns the segment object ON -Write

tran = Trans.State 'returns the state of Transform -Read

C++ Syntax

HRESULT get_State(VARIANT_BOOL *pVal)
HRESULT put_State(VARIANT_BOOL newVal)

Interface

ISegment
IInterfaceControl
ITransform
IGating
ILimitTest
IPortExtension
IFOM
IEquation
IEmbeddedLO
IFIFO

Last Modified:

10-Oct-2008  Added FIFO
13-Apr-2007  Added EmbeddedLO
# State (Rx Leveling) Property

**Description**
Sets and reads the state of Receiver Leveling for a specific source port.

**VB Syntax**

```vbnet
RxLevel.State(srcPort) = value
```

**Variable**

- **RxLevel**
  A `ReceiverLeveling` Object

- **value**
  (Boolean) Choose from:
  - `True` - Receiver leveling ON
  - `False` - Receiver leveling OFF

- **srcPort**
  (Long Integer) Source port for which to set the state of Receiver Leveling.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

**Return Type**

Variant Boolean

**Default**

False

**Examples**

```
rxLevel.State (1) = True ' Write
```

```
value = rxLevel.State 2' Read
```

**C++ Syntax**

```
HRESULT get_State(long port, VARIANT_BOOL* pLevelingState);
HRESULT put_State(long port, VARIANT_BOOL LevelingState);
```

**Interface**

`IRceiverLevelingConfiguration`

---

**Last Modified:**
12-Feb-2009  MX New topic
**State Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Turns the specified pulse generator ON and OFF.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>pulse.State (n) = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td>pulse</td>
<td>A <a href="object">PulseGenerator</a></td>
</tr>
</tbody>
</table>
| n           | *(Integer)* Pulse generator number. Choose from 0 to 4.  
0 is the generator that pulses the ADC. |
| value       | **True** - turns pulse output ON.             |
|             | **False** - turns pulse output OFF.           |
| Return Type | Boolean                                       |
| Default     | False                                         |
| Examples    | `pulse.State(1) = True` `Write`               |
|             | `value = pulse.State(4)` `Read`               |
| C++ Syntax  | HRESULT get_State(VARIANT_BOOL *pVal);         |
|             | HRESULT put_State(VARIANT_BOOL newVal);       |
| Interface   | IPulseGenerator                               |

**Last Modified:**

2-Jan-2007 MX New topic
**StatisticsRange Property**

**Description**
Sets the User Range number for calculating measurement statistics. Set the start and stop values for a User Range with `UserRangeMin` and `UserRangeMax`.

There are 16 User Ranges per channel. User ranges are applied independently to any measurement.

**VB Syntax**
```
meas.StatisticsRange = value
```

**Variable**
- **meas**: A Measurement (object)
- **value**: (long integer) - Range Number. Choose any number between 0 and 16
  - 0 is Full Span
  - 1 - 16 are user-defined ranges

**Return Type**
Long Integer

**Default**
0

**Examples**
```
meas.StatisticsRange = 2 'Write
statrange = meas.StatisticsRange 'Read
```

**C++ Syntax**
```
HRESULT get_StatisticsRange(long* rangeNumber)
HRESULT put_StatisticsRange(long rangeNumber)
```

**Interface**
IMeasurement
**StatusAsString Property**

**Description**
Returns a string that describes the result of the last tuning sweeps.

**VB Syntax**
```
value = embedLODiag.StatusAsString
```

**Variable**

**Type** - Description

- `value` (String) Variable to store the returned data.

**embedLODiag**
An `EmbeddedLODiagnostic` (object)

**Return Type**
(String)

**Default**
Not Applicable

**Examples**
```
data= embedLO.StatusAsString 'read
```

**C++ Syntax**
```
HRESULT get_StatusAsstring(BSTR* status);
```

**Interface**
IEembedLODiagnostic

---

Last Modified:

12-Apr-2007  MX New topic
**StepRiseTime Property**

**Description**
Sets or returns the Rise time of the stimulus in Low Pass Step Mode.

**VB Syntax**
\[trans.StepRiseTime = value\]

**Variable (Type) - Description**
- \(trans\) A Transform (object)
- \(value\) (double) - Rise time in seconds. Choose any number between \(5.0e-13\) and \(1.63e-14\).

**Return Type**
Double

**Default**
0

**Examples**
\[trans.StepRiseTime = 1.0e-14 \text{'sets the step rise time to 100 psec.'} \text{Write}\]
\[rt = trans.StepRiseTime \text{'Read}\]

**C++ Syntax**
- HRESULT get_StepRiseTime(double *pVal)
- HRESULT put_StepRiseTime(double newVal)

**Interface**
ITransform
StepData Property

Description
Returns an array of data from the specified tuning sweep.

VB Syntax
`value = embedLODiag.StepData (n)`

Variable (Type) - Description

- `value` (Variant Array) Variable to store the returned data.
- `embedLODiag` An EmbeddedLODiagnostic (object)
- `n` (Long) Tuning sweep number. Use NumberOfSweeps to find the number of sweeps taken.

Default
Not Applicable

Examples
`data = embedLO.StepData 3 'read`

C++ Syntax
`HRESULT StepData(long sweep,VARIANT* pArray);`

Interface
IEmbededLODiagnostic

Last Modified:
12-Apr-2007    MX New topic
# StepTitle Property

**Description**
Returns the title of the specified tuning sweep.

**VB Syntax**

```vbnet
value = embedLODiag.StepTitle (n)
```

**Variable**

- **value** *(String)* Variable to store the returned data.
- **embedLODiag** An EmbeddedLODiagnostic *(object)*
- **n** *(Long)* Tuning sweep number. Use NumberOfSweeps to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**

```vbnet
data = embedLO StepTitle 3 'read
```

**C++ Syntax**

```csharp
HRESULT StepTitle (long sweep, BSTR * title);
```

**Interface**
IEmbededLODiagnostic

---

Last Modified:
13-Apr-2007  MX New topic
StimulusValues Property

Description  Returns the specified X-axis FOM frequency range. The array contains one frequency value for each data point.

VB Syntax  value = calSet.StimulusValues (range)

Variable  (Type) - Description

calSet  CalSet (object) - Read-only property

range  (Long) FOM frequency range to read.
0  - returns source frequencies.
1  - returns response frequencies.
2  - returns primary frequencies.

Return Type  1-dimensional variant array

Default  Not Applicable

Examples  array = CalSet.StimulusValues 'Read

C++ Syntax  HRESULT get_StimulusValues (long range, VARIANT* vals)

Interface  ICalSet3

Last Modified:

16-Sep-2009  Added 2 - primary

19-Oct-2007  Fixed title and syntax
StopFrequency Property

Description
Sets or returns the stop frequency of the channel. (Channel Object)
Sets or returns the stop frequency of the segment. (Segment Object)
Sets or returns the stop frequency of the FOM Range. (FOMRange Object)
Sets or returns the stop frequency of the Power Sensor coverage (GuidedCalibrationPowerSensor Object)

See also Measurement2 interface.

VB Syntax
object.StopFrequency = value

Variable (Type) - Description

object Any of the following:
Channel (object)
Segment (object)
FOMRange (object)
GuidedCalibrationPowerSensor (object)

value (double) - Stop frequency in Hertz. Choose any number between the minimum and maximum frequencies of the analyzer.

Return Type
Double

Default
Channel - Maximum frequency of the analyzer.
Segment - 0
FOMRange - Maximum frequency of the analyzer.
GuidedCalibrationPowerSensor - Maximum frequency of the analyzer.

Examples
chan.StopFrequency = 4.5e9 'sets the stop frequency for the channel
object -Write

stopfreq = Chan.StopFrequency 'Read

C++ Syntax
HRESULT get_StopFrequency(double *pVal)
HRESULT put_StopFrequency(double newVal)

Interface
IChannel
ISegment
IFOMRange
IGuidedCalibrationPowerSensor
Last modified:

8-Feb-2011  Added GuidedCalibrationPowerSensor
8-Mar-2007  Added FOMRange
1-Nov-2006  Removed Cal Set object - created cs.stopfreq
StopFrequency (Cal Set) Property

Description
Returns the stop frequency that is stored in the Cal Set.

VB Syntax
value = CalSet.StopFrequency (range)

Variable (Type) - Description
value (double) - returned Stop frequency in Hertz.

CalSet (object)

range (Long) Choose: 0 (Source and receiver frequency)

Return Type
Double

Default
Not Applicable

Examples
stop = calset.StopFrequency(0) 'Reads the stop frequency stored in the cal set.

C++ Syntax
HRESULT get_StopFrequency(long range, double *pVal)

Interface |CalSet3

Last modified:
25-Jan-2011 Fixed range argument
Nov. 1, 2006 New command - split from ch.StopFreq
StopPower Property

Description
Sets the Stop Power of the analyzer when sweep type is set to Power Sweep. Frequency of the measurement is set with chan.CWFrequency.

VB Syntax
object.StopPower = value

Variable (Type) - Description

object One of the following:
  • Channel (object)
  • CalSet (object) - Read-only property

value (double) - Stop Power in dB.

Note: The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, use cap.MaximumSourceALCPower and cap.MinimumSourceALCPower.

Auto attenuation is not allowed in Power Sweep.

Return Type
Double

Default
0

Examples
Chan.StopPower = -10 'Write
stppwr = Chan.StopPower 'Read

C++ Syntax
HRESULT get_StopPower(double *pVal)
HRESULT put_StopPower(double newVal)

Interface
IChannel
|CalSet3

Last Modified:

7-Jan-2008 Removed FOMRange
8-Mar-2007 Added FOMRange
### Write/Read

#### Stop Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets or returns the Stop time of either Gating or Time Domain transform windows</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>object.Stop = value</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td>object</td>
<td><em>(object)</em> As Gating or <em>(object)</em> As Transform</td>
</tr>
<tr>
<td>value</td>
<td><em>(double)</em> - Start time in seconds. Choose any number between: ± (number of points - 1) / frequency span</td>
</tr>
<tr>
<td>Return Type</td>
<td>Double</td>
</tr>
<tr>
<td>Default</td>
<td>10 ns</td>
</tr>
</tbody>
</table>
| Examples    | `Trans.Stop = 4.5e-9 'sets the stop time of a transform window - Write`  
`Gate.Stop = 4.5e-9 'sets the stop time of a gating window - Write`  
`stp = Trans.Stop 'Read` |
| C++ Syntax  | `HRESULT get_Stop(double *pVal)`  
`HRESULT put_Stop(double newVal)` |
| Interface   | ITransform  
IGating |
**Read-only**

## Stop Property

**Description**  Returns the stimulus value of the last data point for the measurement. To understand how this property is useful, see [IMeasurement2 Interface](#).

**VB Syntax**  

```
value = meas.Stop
```

**Variable**  

- **(Type) - Description**
  - **value**  (Double) Variable to store the returned value
  - **meas**  A Measurement (object)

**Return Type**  Double

**Default**  Not Applicable

**Examples**  

```vbnet
Print meas.Stop 'prints the stimulus value of the last data point
```

**C++ Syntax**  

```cpp
HRESULT get_Stop(double * Val);
```

**Interface**  IMeasurement2
**StopPhase Property**

**Description**
Write and read the stop value of phase sweep. Must also send Sweep Type Property to put the analyzer into phase sweep mode.

**VB Syntax**
```
phase.StopPhase(srcPort) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>phase</code></td>
<td>A <code>PhaseControl</code> Object</td>
</tr>
<tr>
<td><code>srcPort</code></td>
<td>(Long Integer) Source port for which to make phase control settings. Note: If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use <code>chan.getPortNumber</code> to translate the string into a port number. To learn more see <a href="#">Remotely Specifying a Source Port</a></td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Double) Stop phase value in degrees. Choose a value between -360 and 360.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
0 degrees

**Examples**
```
phase.StopPhase 1 = 60 ' Write
value = phase.StopPhase 2 ' Read
```

**C++ Syntax**
```
HRESULT get_StopPhase(long port, double* pVal);
HRESULT put_StopPhase(long port, double newVal);
```

**Interface**
IPhaseControl

---

Last Modified:
8-Dec-2010    MX New topic
StopPowerEx Property

Sets and reads the power sweep start power value for a specific port. This allows uncoupled forward and reverse power sweep ranges. Must also set `SweepType` = naPowerSweep, `Coupled` = False (Off), and `StartPowerEx`.

**VB Syntax**

```
chan.StopPowerEx (srcPort) = value
```

**Variable**

(*Type*) - Description

- **chan** - A Channel (object)
- **srcPort** - (long integer) – Source port for which to set the Stop power value.
  
  **Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

- **value** - (double) - Stop Power in dBm.
  
  **Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, use `cap.MaximumSourceALCPower` and `cap.MinimumSourceALCPower`

Auto attenuation is not allowed in Power Sweep.

**Return Type**

Double

**Default**

0 dBm

**Examples**

- `Chan.StopPowerEx 1 = -10` 'Write
- `stopPwr = Chan.StopPowerEx 2` 'Read

**C++ Syntax**

```
HRESULT get_StopPowerEx(long port, double *pVal)
HRESULT put_StopPowerEx(long port, double newVal)
```

**Interface**

IChannel13
StopRatioedPower Property

**Description**  Write and read the stop power ratioed value. Must also set SweepType to Power.

**VB Syntax**  

\[
\text{phase.StopRatioedPower}(\text{srcPort}) = \text{value}
\]

**Variable**  

<table>
<thead>
<tr>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>phase</td>
</tr>
</tbody>
</table>

A PhaseControl Object

<table>
<thead>
<tr>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>srcPort</td>
</tr>
</tbody>
</table>

(Long Integer) Source port for which to make phase control settings.

**Note:** If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port

<table>
<thead>
<tr>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
</tr>
</tbody>
</table>

(Double) Stop power ratio value in dBc. Must be within the allowable range of the PNA.

**Return Type**  Double

**Default**  0 dBc

**Examples**  

phase.StopRatioedPower 1 = -1 'Write

value = phase.StopRatioedPower 2' Read

**C++ Syntax**  

HRESULT get_StopRatioedPower(long port, double* pVal);

HRESULT put_StopRatioedPower(long port, double newVal);

**Interface**  IPhaseControl

Last Modified:

8-Dec-2010    MX New topic
Write/Read

strPort2Pdeembed_S2PFile Property

Description
Sets and returns the 2 port De-embedding .S2P file name for the specified port number. Model is applied when both the file name is specified and User is specified using Port2PdeembedCktModel Property.

Learn more about S2P files.

Note: This command affects ALL measurements on the channel.

VB Syntax
fixture.strPort2Pdeembed_S2PFile(port) = value

Variable (Type) - Description

fixture
A Fixturing (object)

port
(Integer) Port number to receive circuit model.

value
(String) Full path, file name, and extension (.s2P) of the de-embedding circuit.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

Return Type
String

Default
Not Applicable

Examples
fixture.strPort2Pdeembed_S2PFile(2) = "C:/Program Files/Agilent/Network Analyzer/Documents/myFile.s2p" 'Write

value = fixture.strPort2Pdeembed_S2PFile(1) 'Read

C++ Syntax
HRESULT get_strPort2Pdeembed_S2PFile(short port BSTR *bstrFile)
HRESULT put_strPort2Pdeembed_S2PFile(short port BSTR bstrFile)

Interface
IFixturing
strPortMatch_S2PFile Property

**Description**
Sets and returns the Port Matching 'S2P' file name for the specified port number. Model is applied when both the file name is specified and User is specified using PortMatchingCktModel Property. Learn more about S2P files.

**Note:** This command affects ALL measurements on the channel.

**VB Syntax**
```
fixture.strPort2PMatch_S2PFile(port) = value
```

**Variable**
- **(Type)** - Description

  - `fixture`  A Fixturing (object)
  - `port`  (Integer) Port number to receive circuit model.
  - `value`  (String) Full path, file name, and extension (.s2P) of the matching circuit. Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents".

**Return Type**
String

**Default**
Not Applicable

**Examples**
```
fixture.strPort2PMatch_S2PFile(2) = "C:/Program Files/Agilent/Network Analyzer/Documents/myFile.s2p" 'Write

value = fixture.strPort2PMatch_S2PFile(1) 'Read
```

**C++ Syntax**
- HRESULT get_strPort2PMatch_S2PFile(short port BSTR *bstrFile)
- HRESULT put_strPort2PMatch_S2PFile(short port BSTR bstrFile)

**Interface**
IFixturing
# SubPointTrigger Property

## Description
Enables / Disables subpoint triggering. When enabled and performing [Point Averaging](#), each rising edge of P0 triggers a subpoint (one of N acquisitions in an N point average). Must also enable the P0 generator using `pulse.State(n)`.

## VB Syntax

```vbnet
pulse.SubPointTrigger(n) = value
```

## Variable

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pulse</td>
<td>PulseGenerator (object)</td>
<td>A PulseGenerator (object)</td>
</tr>
<tr>
<td>n</td>
<td>Integer</td>
<td>Pulse generator number. <strong>Must be 0</strong> as this is the generator that triggers the ADC.</td>
</tr>
<tr>
<td>value</td>
<td>Boolean</td>
<td>Enable or disable SubPointTrigger</td>
</tr>
</tbody>
</table>

- **True** - turns subpoint triggering ON.
- **False** - turns subpoint triggering OFF.

## Return Type
Boolean

## Default
False

## Examples

```vbnet
pulse.SubPointTrigger(0) = True 'Write

bool = pulse.SubPointTrigger(0) 'Read
```

## C++ Syntax

```c++
HRESULT get_SubPointTrigger (integer pulse, VARIANT_BOOL* on_off);
HRESULT put_SubPointTrigger (integer pulse, VARIANT_BOOL on_off);
```

## Interface
IPulseGenerator2

### Last Modified:

- 20-Jul-2009  MX New topic
## Sweep Delay Property

**Description**
Specifies the time to wait just before acquisition begins for each sweep. This delay is in addition to **Dwell Time** and **External Trigger** delay if enabled.

**VB Syntax**

```
chan.SweepDelay = value
```

**Variable (Type) - Description**

- **chan**  
  *Channel (object)*

- **value**  
  *double* - Sweep delay in seconds.

**Return Type**
Double

**Default**
0

**Examples**

```
chan.SweepDelay = 3e-3 'Write

swpdelay = chan.SweepDelay 'Read
```

**C++ Syntax**

```
HRESULT get_SweepDelay(double *pVal)
HRESULT put_SweepDelay(double newVal)
```

**Interface**
IChannel

---

**Last Modified:**

1-Mar-2010  MX New topic
**Read/Write**

**SweepEndMode Property**

**Description**
Sets and reads the event that will cause the Sweep End line to go to a low state. The line will return to a high state after the appropriate calculations are complete.

*Note:* This line is connected to the following pins on the HANDLER IO connector and AUX IO connector in the PNA. Therefore, this command will affect both of these connectors in the same way.

**VB Syntax**

```vbnet
object.SweepEndMode = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>A HandlerIO or AuxIO object</td>
</tr>
</tbody>
</table>

**value**

(enumerator as NASweepEndMode) Choose from:

0 - naSweep - the line goes low when each sweep is complete

1 - naChannelSweep - the line goes low when all the sweeps for each channel is complete.

2 - naGlobalSweep - the line goes low when all sweeps for all triggerable channels are complete.

**Return Type**
Long Integer

**Default**
0 - naSweep

**Examples**

```vbnet
HWAuxIO.PassFailMode = naSweep 'Write
value = HWAuxIO.PassFailMode 'Read
```

**C++ Syntax**

```c++
HRESULT put_SweepEndMode ( tagNASweepEndMode Mode );
HRESULT get_SweepEndMode ( tagNASweepEndMode* Mode );
```

**Interface**

IHWAuxIO

IHWMaterialHandlerIO
**SweepHoldOff Property**

**Description**
Returns a boolean that represents the state of SweepHoldoff line (pin2) of the External Test Set connector.

**VB Syntax**
```vbnet
value = ExtIO.SweepHoldOff
```

**Variable**
- **value** *(boolean)* - Variable to store the returned data
- **ExtIO** *(object)* - An External IO object

**Return Type**
Boolean
- **False** - indicates the line is being held at a TTL Low
- **True** - indicates the line is being held at a TTL High

**Default**
Not Applicable

**Examples**
```vbnet
value = ExtIO.SweepHoldOff
```

**C++ Syntax**
```cpp
HRESULT get_SweepHoldOff( VARIANT_BOOL* bValue);
```

**Interface**
IHWExternaTestSetIO
# SweepOrder Property

**Description**
Sets and returns the order number of IM products to view when `SweepType = NTH` is specified. This actually sets the frequency span to `DeltaF * N` (this command value).

**VB Syntax**
```vbnet
ims.SweepOrder = value
```

**Variable**
<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ims</td>
<td>An IMSpectrum Object</td>
</tr>
<tr>
<td>value</td>
<td>(Integer) Order number of IM product.</td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
9

**Examples**
```vbnet
ims.SweepOrder = 5 'Write
value = ims.SweepOrder 'Read
```

**C++ Syntax**
```c++
HRESULT get_SweepOrder(long *pVal)
HRESULT put_SweepOrder(long newVal)
```

**Interface**
IIMSpectrum

---

**Last Modified:**
19-Aug-2008   MX New topic
### SweepSpeedMode Property

**Description**
Sets and returns the sweep speed mode: FastSweep or Normal.

**VB Syntax**
```vbnet
chan.SweepSpeedMode = value
```

**Variable**
- **Type**: Channel (object)
- **Description**

**Value**
- (enum as NASweepSpeedMode) - Choose from:
  - 0 - naSweepSpeedModeNormal - Standard PNA sweep mode
  - 1 - naSweepSpeedModeFast - Fast sweep mode

**Return Type**
Enum

**Default**
0 - naSweepSpeedModeNormal

**Examples**
```vbnet
chan.SweepSpeedMode = naSweepSpeedModeNormal 'Write
swpSpeed = chan.SweepSpeedMode 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SweepSpeedMode(tagNASweepSpeedModes* pVal)
HRESULT put_SweepSpeedMode(tagNASweepSpeedModes newVal)
```

**Interface**
IChannel14

---

Last Modified:

26-Aug-2008  MX New topic
### SweepGenerationMode Property

**Description**
Sets the method used to generate a sweep: continuous ramp (analog) or discrete steps (stepped).

**VB Syntax**
```
object.SweepGenerationMode = value
```

**Variable**

*object*  
Channel *(object)*  

*or*

CalSet *(object)* - Read-only property

**value** *(enum NASweepGenerationModes)* - Choose either:

- **0 - naSteppedSweep** - source frequency is CONSTANT during measurement of each displayed point. More accurate than Analog. Dwell time can be set in this mode.
- **1 - naAnalogSweep** - source frequency is continuously RAMPING during measurement of each displayed point. Faster than Stepped. Sweep time (not dwell time) can be set in this mode.

**Return Type**
Long Integer

**Default**
Analog

**Examples**
```
Chan.SweepGenerationMode = naAnalogSweep 'Write

swpgen = Chan.SweepGenerationMode 'Read
```

**C++ Syntax**
```
HRESULT get_SweepGenerationMode(tagNASweepGenerationModes* pVal)
HRESULT put_SweepGenerationMode(tagNASweepGenerationModes newVal)
```

**Interface**
IChannel
|CalSet3
## SweepTime Property

### Description
Sets the Sweep time of the analyzer. If sweep time accuracy is critical, use ONLY the values that are attained using the up and down arrows next to the sweep time entry box. See Sweep Time.

### VB Syntax
```
object.SweepTime = value
```

### Variable
- **object (Type)** - Description
  - **object**: Channel (object)
    - or
      - **Segment (object)** first set SweepTimeOption to true.
    - or
      - **CalSet (object)** - Read-only property
- **value (double)** - Sweep time in seconds. The maximum sweep time of the PNA is 86400 seconds (1 day).
  - To set the fastest sweep speed possible, set this value to 0.

### Return Type
**Double**

### Default
0

### Examples
```
chan.SweepTime = 3e-3 'Write
swptme = chan.SweepTime 'Read
```

### C++ Syntax
```
HRESULT get_SweepTime(double *pVal)
HRESULT put_SweepTime(double newVal)
```

### Interface
- IChannel
- CalSet3
- ISegment2
SweepTimeOption Property

**Description**
Enables the Sweep time or Dwell time to be set on individual sweep segments. This property must be set True before the sweep or dwell time commands are sent. Otherwise, those commands will be ignored.

**VB Syntax**
```
segs.SweepTimeOption = state
```

**Variable**
- **Type** - Description
  - **segs** A Segments collection (object)
  - **state** (boolean)
    - **True** - Enables Sweep or Dwell time to be set independently.
    - **False** - Disables Sweep or Dwell time from being set independently.

**Return Type**
Boolean

**Default**
False

**Examples**
```
segs.SweepTimeOption = True 'Write
timeOption = SweepTimeOption 'Read
```

**C++ Syntax**
```
HRESULT get_SweepTimeOption(VARIANT_BOOL *pVal)
HRESULT put_SweepTimeOption(VARIANT_BOOL newVal)
```

**Interface**
ISegments3

Last modified:
9/29/06 MQQ New command
Write/Read About Swept IMD Sweep types

SweepType Property (IMD Opt 087)

Description
Sets and returns the type of sweep for a Swept IMD measurement.
See a list of commands that are relevant for each sweep type.

VB Syntax

```vbnet
imd.SweepType = value
```

Variable (Type) - Description

 IMD A SweptIMD Object

\n
value (Enum as naSweepTypes) - Choose from:

- **0 - naIMDToneCWSweep** The main tone frequencies (F1 and F2) and power levels (P1 and P2) are held constant. Measurements are taken for the specified number of points.

- **1 - naIMDTonePowerSweep** The main tone frequencies are specified as either F1 and F2, or as FC and DeltaF. These frequencies are held constant while the power of each tone is varied from the Start Power to Stop Power.

- **2 - naIMDToneCenterFreqSweep** Maintaining a constant tone spacing (DeltaF) and tone powers (P1 and P2), the center frequency (FC) is swept from Start to Stop, or can also be specified as Center and Span.

- **3 - naIMDDeltaFrequencySweep** The center frequency (FC) is held constant. The tone spacing is increased from Start DeltaF to Stop DeltaF.

- **4 - naIMDToneSegmentSweep** Same as FCenter sweep, except that the center frequencies for the sweep are constructed using the standard segment sweep commands. (NOT valid for IMDx)

- **5 - naLOPowerSweep** All frequencies are fixed while the LO power is swept. (IMDx ONLY)

Return Type

Enum

Default

2 - naIMDToneCenterFreqSweep

Examples

```vbnet
imd.SweepType = naIMDToneCWSweep  'Write

swptyp = imd.SweepType  'Read
```

C++ Syntax

```c++
HRESULT get_SweepType(tagNASweepTypes* pVal)
HRESULT put_SweepType(tagNASweepTypes newVal)
```

Interface

ISweptIMD

Last Modified: 1564
**SweepType Property** (IMSpectrum Opt 087)

**Description**
Sets and returns the type of sweep for an IMSpectrum measurement.

**VB Syntax**

```vbnet
ims.SweepType = value
```

**Variable** *(Type)* - Description

- **ims** An IMSpectrum Object

- **value** (Enum as NAIMSSweepType) - Choose from:
  
  - **0 - naIMSLinearSpan** When Tracking is enabled, allows tuning the Response Settings (receiver) to any values within the frequency range of the PNA. When Tracking is NOT enabled also allows setting the Stimulus (sources) to any values within the frequency range or the PNA.
  
  - **1 - naIMSSecondOrderSpan** The receiver is tuned to view the 2nd order products \((f_2-f_1\) and \(f_1+f_2\)) of the main tones that are currently specified in Stimulus Settings. When Tracking is enabled, the main tones are specified in the Swept IMD channel.
  
  - **2 - naIMSThirdOrderSpan** The receiver is tuned to view the 3rd order products \((2f_1 - f_2\) and \(2f_2-f_1\)) of the main tones that are currently specified in Stimulus Settings. When Tracking is enabled, the main tones are specified in the Swept IMD channel.
  
  - **3 - naIMSNthOrderSpan** The frequency range is set to \(N \times \Delta F\). This algorithm will NOT tune the receivers to see the EVEN order products.

**Return Type**

- **Enum**

**Default**

- **3 - naIMSNthOrderSpan**

**Examples**

```vbnet
ims.SweepType = naIMSNthOrderSpan 'Write
swptyp = ims.SweepType 'Read
```

**C++ Syntax**

- HRESULT get_SweepType(tagNAIMSSweepType* pVal)
- HRESULT put_SweepType(tagNAIMSSweepType newVal)

**Interface**

- IMSpectrum

---

Last Modified:

- 19-Aug-2008 MX New topic
### SweepType Property

**Description**
Sets and returns the type of sweep. First set SweepType, then set sweep parameters such as frequency or power settings.

**VB Syntax**
```vbnet
object.SweepType = value
```

**Variable**
- **(Type)** - Description
  - `object` One of the following:
    - `Channel` (object)
    - `FOMRange` (object) Must be an `UNCOUPLER` range.
    - `CalSet` (object) - Read-only property
  - `value` - Choose from:
    - 0 - naLinearSweep
    - 1 - naLogSweep
    - 2 - naPowerSweep
    - 3 - naCWT ImSweep
    - 4 - naSegmentSweep
    - 5 - naPhaseSweep

**Note**: Sweep type cannot be set to Segment sweep if there are no segments turned ON. A segment is automatically turned ON when a application is created.

**Return Type**
Long Integer

**Default**
naLinearSweep

**Examples**
```vbnet
chan.SweepType = naPowerSweep 'Write
swptyp = chan.SweepType 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SweepType(tagNASweepTypes* pVal)
HRESULT put_SweepType(tagNASweepTypes newVal)
```

**Interface**
- IChannel
- ICaliSet3
- IFOMRange

---

Last Modified: 1567
3-Dec-2010  Added phase sweep
6-May-2010  Added sweep type note.
8-Mar-2007  Added FOMRange
### SystemImpedanceZ0 Property

**Description**
Sets and returns the impedance for the analyzer.

**VB Syntax**
```vbnet
app.SystemImpedanceZ0 = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An <a href="#object">Application</a></td>
</tr>
<tr>
<td><code>value</code></td>
<td>(double) Analyzer Impedance. Choose any number between 0 and 1000 ohms.</td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
50

**Examples**
```vbnet
app.SystemImpedanceZ0 = 75 'Write
z0 = app.SystemImpedanceZ0 'Read
```

**C++ Syntax**
```cpp
HRESULT get_SystemImpedanceZ0(double dSystemZ0)
HRESULT put_SystemImpedanceZ0(double *pdSystemZ0)
```

**Interface**
IApplication
### SystemName Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the computer name of the PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>name = app.SystemName</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>name</code></td>
<td>(String) Variable to store the returned computer name.</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>String</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>name = app.SystemName</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT SystemName(BSTR* computerName)</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IApplication</td>
</tr>
</tbody>
</table>
Write/Read

About Marker Search

TargetValue Property

**Description**  Sets the target value for the marker when doing Target Searches (SearchTargetLeft, SearchTarget, SearchTargetRight).

**VB Syntax**  

```
mark.TargetValue = value
```

**Variable**  (Type) - Description

- **mark**  A Marker (object)

- **value**  (single) - Target value. Choose any number between: -500 and 500

**Return Type**  Single

**Default**  0

**Examples**  

```
mark.TargetValue = 10.5 'Write
```

```
target = mark.TargetValue 'Read
```

**C++ Syntax**  

```
HRESULT get_TargetValue(float *pVal)
HRESULT put_TargetValue(float newVal)
```

**Interface**  IMarker
Write/Read

TestPortPower Property

Description
Sets or returns the RF power level for the channel
or
Sets or returns the RF power level of the segment.

VB Syntax
\[ \text{object}.\text{TestPortPower}(\text{srcPort}) = \text{value} \]

Variable (Type) - Description

- **object**
  - A Channel (object) - to set coupled power, use chan.CouplePorts. If CouplePorts = False, then each port power can be set independently. Otherwise, chanTestPortPower (1) = value sets power level at both ports.
  - or
  - A CalSet (object)
  - or
  - A Segment (object)

- **srcPort**
  - (long integer) - Source Port number.
  - Note: If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

- **value**
  - (double) - RF Power in dBm.
  - Note: The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, use cap.MaximumSourceALCPower and cap.MinimumSourceALCPower.
  - Actual achievable leveled power depends on frequency.

Return Type
Double

Default
0

Examples
\[ \text{chan.TestPortPower}(1) = 5 \ 'sets the port 1 RF power level for the channel} \]
\[ \text{object} \ 'Write} \]

\[ \text{powerlev} = \text{Chan.TestPortPower}(1) 'Read} \]

C++ Syntax
HRESULT get_TestPortPower(long port, double *pVal)
HRESULT put_TestPortPower(long port, double newVal)

Interface
IChannel
ICalSet3
ISegment

Last Modified:
## TestSetType Property

**Description**  Returns the Test Set Type (model) that was used for the Cal Set.

**VB Syntax**  

```vbnet
TSType = calset.TestSetType
```

**Variable**  

- **Type** - Description
- **TSType**  (String) Variable to store the returned test set model.
- **calset**  A **Cal Set** object.

**Return Type**  String

**Default**  Depends on the test set.

**Example**  

```vbnet
TSType = calset.TestSetType
```

**C++ Syntax**  

```cpp
HRESULT get_OutputPorts(BSTR *mapping);
```

**Interface**  ICalset5

---

**Last modified:**  
9/18/06  MQ Added for multiport
### Text Property

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Specifies an equation or expression to be used on the measurement.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>eq.Text = eqText</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>eq</code></td>
<td><strong>Equation</strong> <em>(object)</em></td>
</tr>
<tr>
<td><code>eqText</code></td>
<td><strong>(String)</strong> - Any valid equation or expression.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td><strong>String</strong></td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>eq.Text = &quot;foo=S11/S21&quot;</code></td>
</tr>
<tr>
<td></td>
<td><code>equation = eq.Text</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_Text(BSTR *equation)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Text(BSTR equation)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IEquation</td>
</tr>
</tbody>
</table>
ThruCalMethod Property - Superseded

Description
This command is replaced by PathThruMethod Property. Sets and returns the method for performing the thru portion of the calibration.

VB Syntax
```
obj.ThruCalMethod = value
```

Variable (Type) - Description

- **obj**
  - SMCType (object)
  - or
  - VMCType (object)

- **value**
  - (String) Specifies the Thru method. Case insensitive - include spaces. Choose from:
    - "Default"
    - "Flush Thru" or "FLUSH"
    - "Unknown Thru" or "UNKN"
    - "Adapter Removal" or "ADAP"

Return Type
String

Default
Default

Examples
```
SMC.ThruCalMethod = "UNKN"
```

C++ Syntax
```
HRESULT put_ThruCalMethod(enum NAThruCalMethod thruMethod);
HRESULT get_ThruCalMethod(enum NAThruCalMethod *thruMethod);
```

Interface
SMCType
VMCType
Read/Write

**ThruCalMethod Property  Superseded**

- **Description**: This command is replaced by PathThruMethod Property.
  Sets and returns the method for performing the Cal Method and the THRU portion of the calibration.

- **VB Syntax**: `guidedCal`.ThruCalMethod = value

- **Variable** (Type) - Description
  - `guidedCal` - GuidedCalibration (object)
  - `value` - (Enum as NATruCalMethod) Choose from:
    - **0 - naDefaultCalMethod** - allow the PNA to choose the best possible method (from the following) depending on whether the device or ECal module is insertable or non-insertable and given the model number of the PNA. (default selection if omitted.)
    - **1 - naAdapterRemoval** - Perform Adapter removal calibration.
    - **2 - naFlushThru** - Perform Flush Thru calibration.
    - **3 - naDefinedThru** - Perform Defined Thru calibration. If performing an ECal, this is the Thru standard in the ECal module.
    - **4 - naUnknownThru** - Perform Unknown Thru calibration.
    - **5 - naSOLT** - Perform SOLT calibration
    - **6 - naTRL** - Perform TRL calibration
    - **7 - naQSOLT** - Perform QSOLT calibration.

- **Return Type**: Enum
- **Default**: 0 - naDefaultCalMethod

- **Examples**: `guided.ThruCalMethod = naDefinedThru`

- **C++ Syntax**:
  - `HRESULT get_ThruCalMethod(enum NATruCalMethod *thruMethod);`
  - `HRESULT put_ThruCalMethod(enum NATruCalMethod thruMethod);`

- **Interface**: IGuidedCalibration

---

Last Modified: 30-Apr-2007  MX Superseded
Read/Write
ThruPortList Property

Description

**Note:** Available only on PNA releases 5.0 and greater.

**Note:** Do NOT send this command to rely on SmartCal to determine the most accurate Thru port pairs for the cal. You can send the query form of this command to learn the port pairs determined by SmartCal.

Sets and returns the thru connection port pairs for the calibration.

- For 3-port cals, specify at least two pairs.
- For 4-port cals, specify at least three pairs.
- For highest accuracy, specify more than the minimum pairs.
- For a 2-port cal, there is only one port pair. It is the only pair in the list and it is required.

Learn more about [Thru method and port pairings](#).
See an example of a [4-port guided calibration using COM](#).

<table>
<thead>
<tr>
<th>VB Syntax</th>
<th>Variable (Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>guidedCal.ThruPortList = t1a, t1b, t2a, t2b, t3a, t3b</code></td>
<td><code>guidedCal</code> GuidedCalibration (object)</td>
</tr>
</tbody>
</table>

- `t1a, t1b...` (Variant) Port numbers in pairs - a one-dimensional array of Long integers.
  - `t1a, t1b` (Thru1 - port A and port B)
  - `t2a, t2b` (Thru2 - port A and port B)
  - `t3a, t3b` (Thru3 - port A and port B)

**Return Type** Variant - a one-dimensional array of Long integers.

**Default** The most accurate port pairs for the cal.

**Example**

```vbnet
thruList = Array(1,2,1,3,1,4)
guided.ThruPortList = thruList
'Sets the following three thru connections for a 4-port calibration:
Thru 1 - ports 1 and 2
Thru 2 - ports 1 and 3
Thru 3 - ports 1 and 4
```
C++ Syntax

HRESULT get_ThruPortList(VARIANT* portList);
HRESULT put_ThruPortList(VARIANT portList);

Interface

IGuidedCalibration
TimeOut Property

Description
Sets and returns the Time out value for communication with the external device. An error is returned if communication with the device is not successful within this period of time.

VB Syntax
extDevices.TimeOut = value

Variable (Type) - Description
extDevices An ExternalDevice (object)
value (Double) Time out in milliseconds.

Return Type
Double

Default
20000 milliseconds (20 seconds)

Examples
extDevices.TimeOut = 1000 Write
value = extDevices.TimeOut 'Read

C++ Syntax
HRESULT get_TimeOut(Double* value);
HRESULT put_TimeOut(Double newVal);

Interface
IExternalDevices

Last Modified:
31-Jul-2009  MX New topic
## Title Property

### Description
Writes or reads a custom title for the window. Newer entries replace (not append) older entries. Turn the title ON and OFF with `TitleState`.

### VB Syntax
```vb
win.Title = string
```

### Variable

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>win</code></td>
<td>NaWindow</td>
<td>A NaWindow (object)</td>
</tr>
<tr>
<td><code>string</code></td>
<td>long</td>
<td>Title limited to 50 characters.</td>
</tr>
</tbody>
</table>

### Return Type
String

### Default
Null

### Examples
```vb
win.Title = "Hello World" 'Write

titl = win.Title 'Read
```

### C++ Syntax
```cpp
HRESULT get_Title(BSTR *title)
HRESULT put_Title(BSTR title)
```

### Interface
INAWindow
### TitleState Property

**Description**
Turns ON and OFF the window title. Write a window title with `Title`.

**VB Syntax**

```vbnet
win.TitleState = state
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>win</code></td>
<td>A NaWindow (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean)</td>
</tr>
<tr>
<td></td>
<td>True - Title ON</td>
</tr>
<tr>
<td></td>
<td>False - Title OFF</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
win.TitleState = True 'Write
```

```vbnet
titlestate = win.TitleState 'Read
```

**C++ Syntax**

```c++
HRESULT get_TitleState(VARIANT_BOOL* bState)
HRESULT put_TitleState(VARIANT_BOOL bState)
```

**Interface**
INAWindow
## Tolerance Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and returns the tolerance value for leveling sweeps.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td>$RxLevel.Tolerance(srcPort) = value$</td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td>$RxLevel$</td>
<td>A $ReceiverLeveling$ Object</td>
</tr>
<tr>
<td>$srcPort$</td>
<td>(Long Integer) Source port for which to set the tolerance value for Receiver Leveling.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>If the source port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model, then use $chan.getPortNumber$ to translate the string into a port number. To learn more see <a href="#">Remotely Specifying a Source Port</a>.</td>
</tr>
<tr>
<td>$value$</td>
<td>(Double) Tolerance level in dB.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>(Double)</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>.1 dB</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td>$rxLevel.Tolerance (1) = .5$ 'Write'</td>
</tr>
<tr>
<td></td>
<td>$value = rxLevel.Tolerance$ 'Read'</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_Tolerance(long port, double* pVal);</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Tolerance(long port, double newVal);</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IReceiverLevelingConfiguration</td>
</tr>
</tbody>
</table>

Last Modified:

13-Feb-2009    MX New topic
**TonePower Property**

**Description**
Sets and returns the power level of the Main Tones. Use with IMD sweep types:

- naIMDToneCWSweep
- naIMDToneCenterFreqSweep
- naIMDDeltaFrequencySweep

When tone power is coupled, setting either F1 or F2 power sets both.

**VB Syntax**

```vbnet
object.TonePower (tone) = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>object</strong></td>
</tr>
<tr>
<td><strong>tone</strong></td>
</tr>
<tr>
<td>0</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td><strong>value</strong></td>
</tr>
</tbody>
</table>

**Return Type**
Double

**Default**
-20 dBm

**Examples**

```vbnet
imd.TonePower(naIMDF1Power) = 0 'Write
value = imd.TonePower(naIMDF2Power) 'Read
```

**C++ Syntax**

```cpp
HRESULT get_TonePower(tagNAIMDTonePowerID naIMDF1, double * pVal);
HRESULT put_TonePower(tagNAIMDTonePowerID naIMDF1, double newVal);
```

**Interface**

ISweptIMD
IMSpectrum

---

Last Modified:

19-Aug-2008   MX New topic
**Write/Read**

**About Swept IMD**

**TonePowerStart Property**

**Description**
Sets and returns the start power level of the Main tones. Use with IMD sweep type=naIMDTonePowerSweep.
When tone power is coupled, setting either F1 or F2 power sets both.

**VB Syntax**
object.TonePowerStart (tone) = value

**Variable**
**Type** - Description

*object* A SweptIMD or IMSpectrum Object

*tone* (Enum as NAIMDTonePowerID) Choose from:

- 0 - naIMDF1Power - F1 tone
- 1 - naIMDF2Power - F2 tone

*value* (Double) Start power in dBm. Choose a value between +30 dBm and -30 dBm.

**Return Type**
Double

**Default**
-20 dBm

**Examples**

```vbnet
imd.TonePowerStart(naIMDF1Power) = 0 'Write
value = imd.TonePowerStart(naIMDF2Power) 'Read
```

**C++ Syntax**

```csharp
HRESULT get_TonePowerStart(tagNAIMDTonePowerID naIMDF1, double * pVal);
HRESULT put_TonePowerStart(tagNAIMDTonePowerID naIMDF1, double newVal);
```

**Interface**
ISweptIMD
IMSpectrum

---

Last Modified:

19-Aug-2008    MX New topic
**TonePowerStop Property**

**Description**
Sets and returns the stop power level of the Main tones. Use with IMD sweep type=naIMDTonePowerSweep.

When tone power is coupled, setting either F1 or F2 power sets both.

**VB Syntax**

```vb
object.TonePowerStop (tone) = value
```

**Variable**

- **Type** - Description
  - **object** A SweptIMD or IMSpectrum Object
  - **tone** (Enum as NAIMDTonePowerID) Choose from:
    - 0 - naIMDF1Power - F1 tone
    - 1 - naIMDF2Power - F2 tone
  - **value** (Double) Stop power in dBm. Choose a value between +30 dBm and -30 dBm.

**Return Type**
Double

**Default**
-20 dBm

**Examples**

```vb
imd.TonePowerStop (naIMDF1Power) = 0 'Write
value = imd.TonePowerStop (naIMDF2Power) 'Read
```

**C++ Syntax**

```c++
HRESULT get_TonePowerStop (tagNAIMDTonePowerID naIMDF1, double * pVal);
HRESULT put_TonePowerStop (tagNAIMDTonePowerID naIMDF1, double newVal);
```

**Interface**
SweptIMD
IMSpectrum

---

Last Modified:
19-Aug-2008    MX New topic
TotalIterations Property

**Description**
Returns the total number of iterations required by the last SMART sweep. Returns number of power points for a 2D sweep.

**VB Syntax**
iter = gca.TotalIterations

**Variable (Type) - Description**
- **iter** (Integer) Variable to store the returned number of iterations.
- **gca** A GainCompression (object)

**Return Type**
Integer

**Default**
Not Applicable

**Examples**
data = gca.TotalIterations

**C++ Syntax**
HRESULT get_TotalIterations();

**Interface**
IGainCompression2

Last Modified:
9-May-2008   MX New topic
TotalNumberOfPoints Property

**Description**
Read the total number of points a complete GCA measurement will generate.

- For 2D modes, this is Frequency * Power points
- For SMART Sweep, this is Frequency points.

The total can NOT exceed the PNA maximum. See Frequency and Power points.

**VB Syntax**
```
value = gca.TotalNumberOfPoints
```

**Variable** *(Type) - Description*
- **value** *(integer)* Variable to store the returned total number of points
- **gca** A GainCompression *(object)*

**Return Type** Integer

**Default** 5226 (201 * 26)

**Example**
```
totPoints = gca.TotalNumberOfPoints 'Read
```

**C++ Syntax**
```
HRESULT get_TotalNumberOfPoints(int* pVal)
```

**Interface** IGainCompression

---

**Last Modified:** 11-Sep-2007

MX New topic
## Touchscreen Property

**Description**
Sets and reads the state of the PNA-X Touchscreen (ON and OFF).
This setting remains until changed again from the front-panel or remote command.

**VB Syntax**
```vbnet
app.Touchscreen = state
```

**Variable**
**Type** - Description

- `app` An Application (object)
- `state` (boolean)
  - **False (0)** - Disables use of Touchscreen
  - **True (1)** - Enables use of Touchscreen

**Return Type**
Boolean

<table>
<thead>
<tr>
<th>False</th>
<th>OFF</th>
</tr>
</thead>
<tbody>
<tr>
<td>True</td>
<td>ON</td>
</tr>
</tbody>
</table>

**Default**
TRUE when shipped from factory.

**Examples**
```vbnet
app.Touchscreen = True 'Write

coupl = app.Touchscreen 'Read
```

**C++ Syntax**
```cpp
HRESULT put_Touchscreen(VARIANT_BOOL bState)
HRESULT get_Touchscreen(VARIANT_BOOL *bState)
```

**Interface**
IApplication12

---

Last Modified:

23-Feb-2007   MX New topic
**TraceMath Property**

**Description**
Performs math operations on the measurement object and the trace stored in memory. (There MUST be a trace stored in Memory to perform math. See [Meas.DataToMemory](#) method.)

**VB Syntax**
```
meas.TraceMath = value
```

**Variable**
- **Type** - Description
  - `meas` A [Measurement](#) (object)
  - `value` (enum NAMathOperation) - Choose from:
    - 0 - naDataNormal
    - 1 - naDataMinusMemory
    - 2 - naDataPlusMemory
    - 3 - naDataDivMemory
    - 4 - naDataTimesMemory

**Return Type**
NAMathOperation

**Default**
Normal (0)

**Examples**
```
meas.TraceMath = naDataMinusMemory 'Write
mathOperation = meas.TraceMath 'Read
```

**C++ Syntax**
```
HRESULT get_TraceMath(tagNAMathOperation* pMathOp)
HRESULT put_TraceMath(tagNAMathOperation mathOp)
```

**Interface**
IMeasurement
## TraceMax Property

**Description**
Maximizes (isolates) or restores the active trace in the active window. When TraceMax is ON, the active trace is the ONLY trace on the display. All other traces are hidden.

**VB Syntax**
```vbnet
meas.TraceMax = state
```

**Variable (Type) - Description**

- **meas**: A Measurement (object)
- **state**: (boolean) - Choose from:
  - **True**: Maximizes / isolates the active trace in the window.
  - **False**: Restores other traces to be viewed in the window.

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
meas.TraceMax = True
state = meas.TraceMax
```

### C++ Syntax

- **HRESULT get_TraceMax(VARIANT_BOOL bState)**
- **HRESULT put_TraceMax(VARIANT_BOOL* bState)**

**Interface**
IMeasurement10

---

**Last Modified:**
26-Aug-2008   MX New topic
## TraceTitle Property

**Description**  
Writes and reads data for the trace title area.  
The trace title is embedded in the [trace status field](#).  
The title is turned ON and OFF using [TraceTitleState](#).

**VB Syntax**  
`meas.TraceTitle = value`

**Variable (Type) - Description**

- `meas` A [Measurement](#) (object)
- `value` (string) - Title to be displayed. Any characters (no spaces), enclosed with quotes.

**Return Type**  
String

**Default**  
Not Applicable

**Examples**  
`meas.TraceTitle = "My new s11 measurement"`

`title = TraceTitle`  
`'Read`

**C++ Syntax**  
`HRESULT get_TraceTitle(BSTR *title );`
`HRESULT put_TraceTitle(BSTR title );`

**Interface**  
IMeasurement8

---

**Last Modified:**

16-Jan-2007    MX New topic
## TraceTitleState Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Turns display of the Trace Title ON or OFF. When turned OFF, the previous trace title returns. Create a trace title using TraceTitle Property</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>meas.TraceTitleState = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type) - Description</em></td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(boolean) - Choose from:</em></td>
</tr>
<tr>
<td><strong>True</strong></td>
<td>Turns the trace title ON</td>
</tr>
<tr>
<td><strong>False</strong></td>
<td>Turns the trace title OFF</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Boolean</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>False</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>meas.TraceTitleState = False</code></td>
</tr>
<tr>
<td></td>
<td><code>title = TraceTitleState 'Read</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_TraceTitleState(VARIANT_BOOL *isTitleON);</code></td>
</tr>
<tr>
<td></td>
<td><code>HRESULT put_TraceTitleState(VARIANT_BOOL isTitleON);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMeasurement8</td>
</tr>
</tbody>
</table>

Last Modified:

18-Jun-2007    MX New topic
Write/Read

About Marker Search

Tracking Property

Description: This property, when on, executes the search function `marker/SearchFunction` every sweep. In effect, turning Tracking ON is the same as executing one of the immediate, one-time, “Search...” methods (such as `SearchMin`, `SearchMax`) for every sweep.

VB Syntax: `mark.Tracking = state`

Variable: (Type) - Description

mark A Marker (object)
state (boolean) - Tracking state. Choose from:
False - Tracking OFF
True - Tracking ON

Return Type: Boolean

Default: False

Examples: `mark.Tracking = False` 'Write
`markTracking = mark.Type` 'Read

C++ Syntax: HRESULT put_Tracking(VARIANT_BOOL bOn)
HRESULT get_Tracking(VARIANT_BOOL * pbOn)

Interface: IMarker
Write/Read

About IM Spectrum

TrackingChannel Property

Description
Sets and returns the IMD channel number to which the IM Spectrum channel is coupled. Use TrackingEnable to enable tracking.

VB Syntax
ims.TrackingChannel = value

Variable (Type) - Description

ims An IMSpectrum Object

value (Integer) Existing IMD channel number to which frequency and power settings are coupled.

Return Type
Long Integer

Default
First existing IMD channel

Examples
ims.TrackingChannel = 1 'Write
value = ims.TrackingChannel 'Read

C++ Syntax
HRESULT get_TrackingChannel(long *pVal)
HRESULT put_TrackingChannel(long newVal)

Interface
IIMSpectrum

Last Modified:
19-Aug-2008 MX New topic
Write/Read

About IM Spectrum

TrackingEnable Property

**Description**  
When an IMD channel exists, allows the IM Spectrum frequency and power setting to track (couple with) the IMD channel settings. Use TrackingChannel to set the channel number to track.

**VB Syntax**  
ims.TrackingEnable = value

**Variable**  
*ims* (Type) - Description

- **ims** An IMSpectrum Object

- **value** (Boolean) Tracking state. Choose from:
  - **True** - IM Spectrum frequency and power settings track the IMD channel settings.
  - **False** - IM Spectrum frequency and power settings are specified in the IMS channel.

**Return Type**  
Boolean

**Default**  
False

**Examples**  
ims.TrackingEnable = True  'Write

value = ims.TrackingEnable  'Read

**C++ Syntax**  
HRESULT get_TrackingEnable(VARIANT_BOOL* bValue)

HRESULT put_TrackingEnable(VARIANT_BOOL newVal)

**Interface**  
IIMSpectrum

Last Modified:

19-Aug-2008   MX New topic
TrackingManualStepEnable Property

**Description**
Sets and returns the step sweep mode for the IM Spectrum channel.

**VB Syntax**

```vbnet
ims.TrackingManualStepEnable = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>ims</code></td>
<td>An <code>IM Spectrum</code> Object</td>
</tr>
<tr>
<td><code>value</code></td>
<td>(Boolean) Choose from:</td>
</tr>
<tr>
<td></td>
<td><strong>False</strong> - Automatic Step</td>
</tr>
<tr>
<td></td>
<td><strong>True</strong> - Manual Step</td>
</tr>
</tbody>
</table>

**Return Type**
Boolean

**Default**
False - Automatic Step

**Examples**

```vbnet
ims.TrackingManualStepEnable = True 'Write
value = ims.TrackingManualStepEnable 'Read
```

**C++ Syntax**

```c
HRESULT get_TrackingManualStepEnable(VARIANT_BOOL *bVal)
HRESULT put_TrackingManualStepEnable(VARIANT_BOOL newVal)
```

**Interface**
`IIM Spectrum`

Last Modified:
19-Aug-2008    MX New topic
TrackingStepIndex Property

Description
When TrackingManualStepEnable = True (Manual step), sets and returns the data point number at which the IM spectrum measurement occurs.

VB Syntax
ims.TrackingStepIndex = value

Variable (Type) - Description
ims An IMSpectrum Object

value (Integer) Existing IMD channel number to which frequency and power settings are coupled.

Return Type
Long Integer

Default
1

Examples
ims.TrackingStepIndex = 50 'Write
value = ims.TrackingStepIndex 'Read

C++ Syntax
HRESULT get_TrackingStepIndex(long *pVal)
HRESULT put_TrackingStepIndex(long newVal)

Interface
IIMSpectrum

Last Modified:
19-Aug-2008   MX New topic
### TriggerDelay Property

**Description**  
Sets and reads the trigger delay for all measurements (GLOBAL). This delay is only applied while in `app.Source = naTriggerSourceExternal` and `trigsetup.Scope = naGlobalTrigger`. After an external trigger is applied, the start of the sweep is delayed for the specified delay value plus any inherent latency.

To apply a trigger delay for a channel only, use `ExternalTriggerDelay Property`.

**VB Syntax**  
```
app.TriggerDelay = value
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>Double</td>
<td>Trigger delay value in seconds. Range is from 0 to 107</td>
</tr>
</tbody>
</table>

**Return Type**  
Double

**Default**  
0

**Examples**

```
app.TriggerDelay = .003  'Write

delay = app.TriggerDelay 'Read
```

**C++ Syntax**  
```
HRESULT get_TriggerDelay(double *delay);
HRESULT put_TriggerDelay(double delay)
```

**Interface**  
IApplication
**TriggerInPolarity Property**

**Description**
Specifies the polarity of the trigger IN signal.

- **AuxTrigger Object** - Sets the polarity to which the rear-panel AuxTrig IN responds.
- **PulseGenerator Object** - Sets the polarity of trigger to which the internal pulse generators will respond when being externally triggered. **Note:** This feature requires DSP version: 4.0 FPGA:34 or higher. Learn more.

**Note:** Used on PNA-X ONLY.

**VB Syntax**

```vbnet
object.TriggerInPolarity = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>object</td>
<td>An AuxTrigger (object) or A PulseGenerator (object)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>(enum NATriggerPolarity) - Choose from:</th>
</tr>
</thead>
<tbody>
<tr>
<td>naTriggerPositive</td>
<td>PNA responds to rising edge or HIGH level</td>
</tr>
<tr>
<td>naTriggerNegative</td>
<td>PNA responds to falling edge or LOW level.</td>
</tr>
</tbody>
</table>

Set Edge or Level triggering using **TriggerInType Property**

**Return Type**
Enum

**Default**

AuxTriggerIn Object - naTriggerNegative
PulseGenerator Object - naTriggerPositive. Also the polarity used when the PNA-X does not have the required DSP hardware

**Exaamples**

```vbnet
obj.TriggerInPolarity = naTriggerPositive 'Write
value = obj.TriggerInPolarity 'Read the value
```

**C++ Syntax**

```c++
HRESULT get_TriggerInPolarity(enum NATriggerPolarity *val);
HRESULT put_TriggerInPolarity(enum NATriggerPolarity val);
```

**Interface**

IAuxTrigger
IPulseGenerator3
9-Dec-2009  Added PulseGenerator Object

5-Sep-2008  Added Note

14-Dec-2006  MX New topic
**TriggerInType Property**

**Description**
Specifies the type of trigger input being supplied to the PNA.

- **AuxTrigger Object** - Sets the type to which the rear-panel AuxTrig IN responds.
- **PulseGenerator Object** - Sets the type of trigger to which the internal pulse generators will respond when being externally triggered. **Note:** This feature requires DSP version: 4.0 FPGA:34 or higher. Learn more.

**Note:** Use on PNA-X ONLY.

**VB Syntax**

```vbnet
obj.TriggerInType = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>obj</td>
<td>An AuxTrigger (object) or PulseGenerator (object)</td>
</tr>
<tr>
<td>value</td>
<td>(enum NATriggerSignalType) Choose from: naTriggerEdge PNA responds to the edge (rising or falling) of a signal naTriggerLevel PNA responds to the level (HIGH or LOW) of a signal Use TriggerInPolarity to set Positive or Negative polarity.</td>
</tr>
</tbody>
</table>

**Return Type**
Enum

**Default**
naTriggerLevel - Also the type used for the PulseGenerator Object when the PNA-X does not have the required DSP hardware

**Examples**

```vbnet
obj.TriggerInType = naTriggerEdge 'Write
value = obj.TriggerInType 'Read the value
```

**C++ Syntax**

```c++
HRESULT get_TriggerInType(enum NATriggerSignalType *val);
HRESULT put_TriggerInType(enum NATriggerSignalType val);
```

**Interface**
IAuxTrigger
IPulseGenerator3
9-Dec-2009   Added PulseGenerator obj.
5-Sep-2008   Added Note
14-Dec-2006  MX New topic
**TriggerMode (ExtendedProperties) Property**

**Description**
Sets and returns the trigger mode for an external source.

**VB Syntax**
```vbnet
extSource.TriggerMode = value
```

**Variable**
- **Type** - Description
  - `extSource` An [ExternalSource Object](#) (object)
  - `value` (enum NAExtDevTriggerMode) - Choose from:
    - 0 - `naExtDevTriggerModeCW`
    - 1 - `naExtDevTriggerModeHW`

**Return Type**
Enum

**Default**
0 - `naExtDevTriggerModeCW`

**Examples**
- `extSource.TriggerMode = naExtDevTriggerModeCW`  
  `'Write`
- `tm = extSource.TriggerMode`  
  `'Read`

**C++ Syntax**
```
HRESULT get_TriggerMode (tagNAExtDevTriggerMode *pMode)
HRESULT put_TriggerMode (tagNAExtDevTriggerMode newMode)
```

**Interface**
IExternalSource

---

**Last Modified:**
- 8-Sep-2009  
  MX New topic
## About Auxiliary Triggering

### TriggerOutDuration Property

**Description**
Specifies the width of the output pulse, which is the time that the Aux trigger output will be asserted.

**VB Syntax**
```vbnet
auxTrig.TriggerOutDuration = value
```

**Variable**
- **Type**: Description
- **auxTrig**: An `AuxTrigger` (object)
- **value**: (single) - Duration value in seconds. Choose a value between 1E-6 and 1.

**Return Type**
Double

**Default**
1E-6 sec

**Examples**
```vbnet
auxTrig.TriggerOutDuration = 1e-3 'Write
value = auxTrig.TriggerOutDuration 'Read the value
```

**C++ Syntax**
```c++
HRESULT get_TriggerOutDuration(double *val);
HRESULT put_TriggerOutDuration(double val);
```

**Interface**
`IAuxTrigger`

---

**Last Modified:**
14-Dec-2006  MX New topic
# TriggerOutInterval Property

**Description**
Specifies how often a trigger output signal is sent.

**VB Syntax**

```vb
auxTrig.TriggerOutInterval = value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>auxTrig</td>
<td>An <code>AuxTrigger</code> (object)</td>
</tr>
<tr>
<td>value</td>
<td>(single) - Choose from:</td>
</tr>
</tbody>
</table>

- **0** - `naTriggerModePoint`: a single data point is measured with each trigger signal the channel receives. Subsequent trigger signals continue to go to the channel in Point mode until the channel measurements are complete. This is effectively the same as trigger point mode.

- **1** - `naTriggerModeMeasurement`: entire traces are swept with a trigger signal, which and how many traces depends on the Scope setting.

**Return Type**
Enum

**Default**

1 - `naTriggerModeMeasurement`

**Examples**

```vb
auxTrig.TriggerOutInterval = naTriggerModeMeasurement 'Write

value = auxTrig.TriggerOutInterval 'Read the value
```

**C++ Syntax**

```c++
HRESULT get_TriggerOutInterval(enum NATriggerMode *val);
HRESULT put_TriggerOutInterval(enum NATriggerMode val);
```

**Interface**
IAuxTrigger

---

Last Modified:

14-Dec-2006   MX New topic
About Auxiliary Triggering

**TriggerOutPolarity Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Specifies the polarity of the trigger output signal being supplied by the PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>auxTrig.TriggerOutPolarity = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>auxTrig</code></td>
<td>An <code>AuxTrigger</code> (object)</td>
</tr>
<tr>
<td><code>value</code></td>
<td><strong>(enum NATriggerPolarity) - Choose from:</strong></td>
</tr>
<tr>
<td></td>
<td><code>naTriggerPositive</code> PNA sends positive going (active HIGH) pulse.</td>
</tr>
<tr>
<td></td>
<td><code>naTriggerNegative</code> PNA sends negative going (active LOW) pulse.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Enum</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td></td>
</tr>
</tbody>
</table>

**Examples**

`auxTrig.TriggerOutPolarity = naTriggerPositive`  `'Write`

`value = auxTrig.TriggerOutPolarity`  `'Read the value`

**C++ Syntax**

`HRESULT get_TriggerOutPolarity(enum NATriggerPolarity *val);`
`HRESULT put_TriggerOutPolarity(enum NATriggerPolarity val);`

**Interface**

`IAuxTrigger`

---

Last Modified:

14-Dec-2006  MX New topic
**Write/Read**

**About Auxiliary Triggering**

**TriggerOutPosition Property**

**Description**
Specifies whether the Aux trigger out signal is sent Before or After the acquisition.

**VB Syntax**

```vbnet
auxTrig.TriggerOutPosition = value
```

**Variable**

- **(Type)**: Description
- **auxTrig**: An `AuxTrigger (object)`
- **value**: (enum NATriggerPosition) Choose from:
  - **naTriggerOutBeforeAcquire**: Use if the external device needs to be triggered before the data is acquired, such as a power meter.
  - **naTriggerOutAfterAcquire**: Use if the external device needs to be triggered just after data has been acquired, such as an external source. This could be more efficient since it allows the external device to get ready for the next acquisition at the same time as the PNA.

**Return Type**
Enum

**Default**
`naTriggerOutAfterAcquire`

**Examples**

```vbnet
auxTrig.TriggerOutPosition = naTriggerOutAfterAcquire 'Write
```

```vbnet
value = auxTrig.TriggerOutPosition 'Read the value
```

**C++ Syntax**

```cpp
HRESULT get_TriggerOutPosition(enum NATriggerPosition *val);
HRESULT put_TriggerOutPosition(NATriggerPosition val);
```

**Interface**
IAuxTrigger

**Last Modified:**

14-Dec-2006    MX New topic
TriggerOutputEnabled Property

Description: Enables the PNA to send trigger signals out the rear-panel TRIGGER OUT connector. For more information, see External triggering.

VB Syntax: `trigsetup.TriggerOutputEnabled = boolean`

Variable: `(Type) - Description`

`trigsetup` A `TriggerSetup2 (object)`

`boolean` Choose from:
- **False** - PNA does NOT send output trigger signals.
- **True** - PNA sends output trigger signals.

Return Type: Boolean

Default: False

Examples:
- `trigsetup.TriggerOutputEnabled = True 'Write`
- `atba = trigsetup.TriggerOutputEnabled 'Read`

C++ Syntax:
- `HRESULT get_TriggerOutputEnabled( BOOL *pVal);`
- `HRESULT put_TriggerOutputEnabled( BOOL newVal);`

Interface: ITriggerSetup2
**TriggerPort Property**

**Description**  
Sets and returns the PNA port through which an external source is to be triggered.

**VB Syntax**  
`extSource.TriggerPort = value`

**Variable**  
**(Type) - Description**

- `extSource`  
  An [ExternalSource Object](object)

- `value`  
  *(enum NAExtDevTriggerPort)*  
  - Choose from:
    - 0 - `naExtDevTriggerPortBNC1` (PNA 'C' models)
    - 1 - `naExtDevTriggerPortAux1` (PNA-X models)
    - 2 - `naExtDevTriggerPortAux2` (PNA-X models)

**Return Type**  
Enum

**Default**  
For PNA 'C' models - BNC1  
For PNA-X models - Aux1

**Examples**  
```
extSource.TriggerPort = 1 'Write
trigpt = extSource.TriggerPort 'Read
```

**C++ Syntax**  
```c++
HRESULT get_TriggerPort (tagNAExtDevTriggerPort *pValue)
HRESULT put_TriggerPort (tagNAExtDevTriggerPort newVal)
```

**Interface**  
IEnternalSource

---

**Last Modified:**  
31-Jul-2009   MX New topic
### TuningIFBW Property

**Description**  
Set the IF Bandwidth for Broadband and Precise tuning sweeps.

**VB Syntax**  
`obj.TuningIFBW = value`

**Variable (Type) - Description**
- **obj**: An [EmbeddedLO](#) (object) or a [ConverterEmbeddedLO](#) (object)
- **value**: (Double) IF Bandwidth

**Return Type**  
(Double)

**Default**  
30 kHz

**Examples**

```vbnet
embedLO.TuningIFBW = 10e3 'write

value = embedLO.TuningIFBW 'read
```

**C++ Syntax**

```c++
HRESULT get_TuningIFBW(double* ifbw);
HRESULT put_TuningIFBW(double ifbw);
```

**Interface**  
IEmbededLO

---

**Last Modified:**
- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
**TuningMode Property**

**Description**
Sets and returns the method used to determine the embedded LO Frequency.

**VB Syntax**
```
obj.TuningMode = value
```

**Variable (Type) - Description**

obj  
An EmbeddedLO (object) or  
A ConverterEmbeddedLO (object)

value  
(Enum as NAEmbeddedLOTuningMode)
Tuning mode. Choose from:

0 - naEmbeddedLOTuningMode_Broadband_And_Precise
1 - naEmbeddedLOTuningMode_Precise_Only
2 - naEmbeddedLOTuningMode_None

**Return Type**
(Enum)

**Default**
0 - naEmbeddedLOTuningMode_Broadband_And_Precise

**Examples**
```
embedLO.TuningMode = naEmbeddedLOTuningMode_None 'write

value = embedLO.TuningMode 'read
```

**C++ Syntax**
```
HRESULT get_TuningMode(enum NAEmbeddedLOTuningMode* mode);
HRESULT put_TuningMode(enum NAEmbeddedLOTuningMode mode);
```

**Interface**
IEembededLO

---

**Last Modified:**
- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
Read/Write

TuningSweepInterval Property

**Description**  Set how often a tuning sweep is performed.

**VB Syntax**

```vbnet
obj.TuningSweepInterval = value
```

**Variable**  **(Type)** - **Description**

- `obj`  An `EmbeddedLO (object)` or an `ConverterEmbeddedLO (object)`
- `value`  **(Long)** Tuning sweep interval.

**Return Type**  **(Long)**

**Default**  1

**Examples**

```vbnet
embedLO.TuningSweepInterval = 3 'write .. tuning is performed every third measurement sweep
```

```vbnet
value = embedLO.TuningSweepInterval 'read
```

**C++ Syntax**

```cpp
HRESULT get_TuningSweepInterval(long* interval);
HRESULT put_TuningSweepInterval(long interval);
```

**Interface**  IEmbededLO

---

Last Modified:

12-Aug-2009  Added ConvEmbedLO object

18-Apr-2007  MX New topic
Write/Read

TwoPointGroupDelayAperture Property

Description
Sets the default group delay aperture setting.

VB Syntax
`pref.TwoPointGroupDelayAperture = value`

Variable (Type) - Description
- `pref` A `Preferences` (object)
- `value` (Boolean) - Choose from:
  - True - Set the default group delay aperture setting to two points.
  - False - Set the default group delay aperture setting to 11 points.

Return Type
Boolean

Default
False

Examples
`pref.TwoPointGroupDelayAperture = True` 'Write
`gda = pref.TwoPointGroupDelayAperture` 'Read

C++ Syntax
`HRESULT get_TwoPointGroupDelayAperture(VARIANT_BOOL* pVal);`
`HRESULT put_TwoPointGroupDelayAperture(VARIANT_BOOL pVal);`

Interface
IPreferences11

Last Modified:
1-Mar-2010   MX New topic
**TriggerMode Property**

**Description**
These settings determine what EACH signal will trigger.

**Note:** Setting Point and EverySweep mode forces `Trigger.Scope = naChannelTrigger`.

**VB Syntax**

`chan.TriggerMode = value`

**Variable (Type) - Description**

- **chan** A `Channel` (object)
- **value** (enum `NATriggerMode`) - Choose from:
  - 0 - `naTriggerModePoint` - Each Manual or External trigger signal causes one data point to be measured.
  - 1 - `naTriggerModeMeasurement` (superseded - still works but replaced with a more descriptive enum)
  - 1 - `naTriggerModeChannel` - Each trigger signal causes ALL traces in that channel to be swept.
  - 2 - `naTriggerModeEverySweep` - Each Manual or External trigger signal causes ALL traces that share a source port to be swept.
  - 3 - `naTriggerModeTrace` - Allowed ONLY when `PointSweepState` is enabled. Each trigger signal causes two identical measurements to be triggered separately - one trigger signal is required for each measurement. Other trigger mode settings cause two identical parameters to be measured simultaneously.

**Return Type**
Long Integer

**Default**
1 - `naTriggerModeChannel`

**Examples**

- `chan.TriggerMode = naTriggerModePoint` 'Write
- `trigtyp = chan.TriggerMode` 'Read

**C++ Syntax**

- `HRESULT get_TriggerMode (tagNATriggerMode *pMode)`
- `HRESULT put_TriggerMode (tagNATriggerMode newMode)`

**Interface**
IChannel

---

**Last Modified:**
6-Nov-2007   Added new sweep mode
**TriggerSignal Property - Superseded**

<table>
<thead>
<tr>
<th>Description</th>
<th><strong>Note:</strong> This command has been replaced by <strong>Source</strong> Property</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sets or returns the trigger source.</td>
<td></td>
</tr>
</tbody>
</table>

**VB Syntax**

```vbnet
app.TriggerSignal = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>app</strong> An Application <em>(object)</em></td>
</tr>
</tbody>
</table>

**value** *(enum NATriggerSignal)* - Choose from:

- **0** - `naTriggerInternal` - free run
- **1** - `naTriggerExternalPositive` - a trigger signal is generated when a TTL high is sensed on the external trigger pin of the Aux IO connector
- **2** - `naTriggerExternalNegative` - a trigger signal is generated when a TTL low is sensed on the external trigger pin of the Aux IO connector.
- **4** - `naTriggerExternalHigh` - a trigger signal is generated when a TTL high is sensed on the external trigger pin of the Aux IO connector
- **5** - `naTriggerExternalLow` - a trigger signal is generated when a TTL low is sensed on the external trigger pin of the Aux IO connector.

**Return Type**

Long Integer

**Default**

`naTriggerInternal`

**Examples**

```vbnet
app.TriggerSignal = naTriggerExternalPositive 'Write
trigsign = app.TriggerSignal 'Read
```

**C++ Syntax**

```cpp
HRESULT get_TriggerSignal(tagNATriggerSignal *pSignal)
HRESULT put_TriggerSignal(tagNATriggerSignal signal)
```

**Interface**

`IApplication`
**TriggerType Property - Superseded**

**Description**  
*Note: This property has been replaced with Scope Property.*

Sets or returns the trigger type which determines the scope of a trigger signal.

**VB Syntax**  
```
app.TriggerType = value
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>app</th>
<th>An Application (object)</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(enum NATriggerType) - Trigger type. Choose from:</td>
</tr>
<tr>
<td>0</td>
<td>naGlobalTrigger - a trigger signal is applied to all triggerable channels</td>
</tr>
<tr>
<td>1</td>
<td>naChannelTrigger - a trigger signal is applied to the current channel. The next trigger signal will be applied to the next channel; not necessarily channel 1-2-3-4.</td>
</tr>
</tbody>
</table>

**Return Type**  
Long Integer

**Default**  
naGlobalTrigger

**Examples**  
```
app.TriggerType = naGlobalTrigger 'Write
trigtyp = app.TriggerType 'Read
```

**C++ Syntax**  
```
HRESULT get_TriggerType(tagNATriggerType *pTrigger)
HRESULT put_TriggerType(tagNATriggerType trigger)
```

**Interface**  
IApplication
## Type (calstd) Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets and Returns the type of calibration standard.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>calstd.Type = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>calstd</code></td>
<td>A CalStandard <em>(object)</em>. Use <code>calKit.GetCalStandard</code> to get a handle to the standard.</td>
</tr>
<tr>
<td><code>value</code></td>
<td><em>(enum NACalStandardType)</em> - Choose from: 0 - <code>naOpen</code> 1 - <code>naShort</code> 2 - <code>naLoad</code> 3 - <code>naThru</code></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Long Integer</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>calstd.Type = naOpen</code> <em>Write</em>  <code>standardtype = calstd.Type</code> <em>Read</em></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_Type(tagNACalStandardType *pVal)</code>  <code>HRESULT put_Type(tagNACalStandardType newVal)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ICalStandard</td>
</tr>
</tbody>
</table>
**About External Testset Control**

### Type (testset) Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns the testset model number.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>tset.Type model</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>model</code> (String)</td>
<td>Variable to return the Test set model</td>
</tr>
<tr>
<td><code>tset</code></td>
<td>A <code>TestsetControl</code> object. Obtained from the <code>ExternalTestsets</code> collection.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>String</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>None</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>testset.type model</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT get_Type(BSTR *pType);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>ITestsetControl</code></td>
</tr>
</tbody>
</table>
## TZImag Property

### Description
Sets and Returns the TZImag value (the Imaginary Terminal Impedance value) for the calibration standard. Only applicable when "Type" is set to `naArbitraryImpedance`.

To set the other resistance values, use `TZReal`.

### VB Syntax
`calstd.TZImag = value`

### Variable (Type) - Description

- **calstd**: A CalStandard `(object)`. Use `calKit.GetCalStandard` to get a handle to the standard.

- **value**: `(single)` - Value for TZImag in Ohms

### Return Type
Single

### Default
Not Applicable

### Examples
```vbnet
  calstd.TZImag = 15 'Write the value of TZImag to 15 Ohms

  imp0 = calstd.TZImag 'Read the value of TZImag
```

### C++ Syntax
```csharp
HRESULT get_TZImag(float *pVal);
HRESULT put_TZImag(float newVal);
```

### Interface
ICalStandard2
### TZReal Property

**Description**
Sets and Returns the TZReal value (the real Terminal Impedance value) for the calibration standard. Only applicable when "**Type**" is set to **naArbitraryImpedance**.

To set the other resistance values, use **TZImag**

**VB Syntax**
```vbnet
calstd.TZReal = value
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>calstd</td>
<td>(object)</td>
<td>A CalStandard. Use calKit.GetCalStandard to get a handle to the standard.</td>
</tr>
<tr>
<td>value</td>
<td>(single)</td>
<td>Value for TZReal in Ohms</td>
</tr>
</tbody>
</table>

**Return Type**
Single

**Default**
Not Applicable

**Examples**
```
calstd.TZReal = 15 'Write the value of TZReal to 15 Ohms
imp0 = calstd.TZReal 'Read the value of TZReal
```

**C++ Syntax**

```
HRESULT get_TZReal(float *pVal);
HRESULT put_TZReal(float newVal);
```

**Interface**
ICalStandard2
### UnusedChannelNumbers Property

**Description**
Returns an array of channel numbers that are NOT in use. An unused channel has NO measurements subscribed to it.

**VB Syntax**
```
chanNumbers = chans.UnusedChannelNumbers (NumberOfChannels)
```

**Variable**

- **chanNumbers** Variable array to store the returned channel numbers
- **chans** A Channel collection (object)

- **NumberOfChannels** (Long Integer) Number of channels that you are requesting.

**Return Type**
One-dimensional array of long integers. The size of the array is specified by the `NumberOfChannels` parameter.

- **Default** Not Applicable

**Examples**
```
chanNumbers = chans.UnusedChannelNumbers(5)
```

**C++ Syntax**
```
HRESULT get_UnusedChannelNumbers(long numberRequested,VARIANT* channelNumbers);
```

**Interface**
IChannels2
**USBPowerMeterCatalog Property**

**Description**
Returns the ID string of power meters / sensors that are connected to the PNA USB. Use the list to select a power sensor for a source power cal.

**VB Syntax**

```vbnet
list = pwrCal.USBPowerMeterCatalog
```

**Variable**

(Type) - Description

- **list** (String) Variable to store the returned list of USB power meters.
- **pwrCal** (object) – A SourcePowerCalibrator (object)

**Return Type**
Comma-delimited strings. Two sensor strings are separated by a semicolon.

**Default**
Not Applicable

**Examples**

```vbnet
Set pwrCal = pna.SourcePowerCalibrator
list = pwrCal.USBPowerMeterCatalog
```

**C++ Syntax**

```cpp
HRESULT get_USBPowerMeterCatalog(BSTR *pUSBList);
```

**Interface**
ISourcePowerCalibrator6

---

**Last Modified:**

- **24-Feb-2009**   Updated with semicolon
- **11-Jul-2007**   MX New topic
UseCalWindow Property

**Description**
Turns Calibration window ON or OFF during a calibration. [Learn more.]

**VB Syntax**
`guidedCal.UseCalWindow = value`

**Variable (Type) - Description**
- `guidedCal` GuidedCalibration (object)
- `value` (Boolean)
  - **True**  Show calibration window
  - **False**  Hide calibration window

**Return Type**
Boolean

**Default**
True

**Example**
`guided.UseCalWindow = True`

**C++ Syntax**
HRESULT get_UseCalWindow(VARIANT_BOOL* val);
HRESULT put_UseCalWindow(VARIANT_BOOL newVal);

**Interface**
IGuidedCalibration

---

Last Modified:
12-Sep-2007  MX New topic
**Read-only**

**UsedChannelNumbers Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns an array of channel numbers that are in use. A used channel has at least one measurement subscribed to it</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td>$chanNumbers = chans.UsedChannelNumbers$</td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td>$chanNumbers$</td>
<td>Variable array to store the returned channel numbers</td>
</tr>
<tr>
<td>$chans$</td>
<td>A <strong>Channel collection</strong> <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>One-dimensional array of long integers</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td>$chanNumbers = chans.UsedChannelNumbers$</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT get_UsedChannelNumbers(VARIANT* channelNumbers);</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IChannels2</td>
</tr>
</tbody>
</table>
UseMultipleSensors Property

**Description**
Enable and disable the use of multiple power sensors during a guided calibration.

**VB Syntax**
```
guidPwrSensors.UseMultipleSensors = value
```

**Variable** *(Type) - Description*

- `guidPwrSensors` *(GuidedCalibrationPowerSensors Collection)*
- `value` *(Boolean)*
  - True  Use multiple power sensors
  - False  Do NOT use multiple power sensors

**Return Type**  Boolean

**Default**  False

**Examples**
```
guidedPwrSensors.UseMultipleSensors = True
value = guidedPwrSensors.UseMultipleSensors
```

**C++ Syntax**
```
HRESULT get_UseMultipleSensors(VARIANT_BOOL* val);
HRESULT put_UseMultipleSensors(VARIANT_BOOL newVal);
```

**Interface**  IGuidedCalibrationPowerSensors

---

Last Modified:

8-Feb-2011  New topic
# UsePowerLossSegments Property

**Description**
Specifies if subsequent power readings will use of the loss table. (PowerLossSegments).

**VB Syntax**
```vbnet
pwrCal.UsePowerLossSegments = value
```

**Variable**
- **(Type)** - Description
- **pwrCal**
  - A [SourcePowerCalibrator](#) (object)
  - A [PowerSensorAsReceiver](#) (Object)
- **value**
  - **(boolean)**
  - **False** – Do not use loss table
  - **True** – Use loss table

**Return Type**
Boolean

**Default**
False

**Examples**
```vbnet
pwrSens.UsePowerLossSegments = True 'Write
lossTableState = pwrSens.UsePowerLossSegments 'Read
```

**C++ Syntax**
```c++
HRESULT put_UsePowerLossSegments(VARIANT_BOOL bState);
HRESULT get_UsePowerLossSegments(VARIANT_BOOL *bState);
```

**Interface**
- ISourcePowerCalibrator
- IPowerSensorAsReceiver

---

**Last Modified:**
25-Aug-2009   MX New topic
UsePowerSensorFrequencyLimits Property

Description
Specifies if subsequent calls to the AcquirePowerReadings method will observe frequency values of the MinimumFrequency and MaximumFrequency properties.

VB Syntax
`pwrCal.UsePowerSensorFrequencyLimits = value`

Variable (Type) - Description
- `pwrCal (object)` – A SourcePowerCalibrator (object)
- `value (boolean)` -
  - **False** – Do not use power sensor frequency limits. An acquisition will use just one power sensor for the entire sweep, regardless of frequency.
  - **True** – Use power sensor frequency limits. A requested acquisition will only succeed for those frequency points which fall between the MinimumFrequency and MaximumFrequency values of that PowerSensor. An acquisition will pause in mid-sweep if the frequency is about to exceed the MaximumFrequency value. When the sweep is paused in this manner, a sensor connected to the other channel input of the power meter can be connected to the measurement port in place of the previous sensor, and then the sweep completed by another call to AcquirePowerReadings. However, the MaximumFrequency specified for the second sensor would need to be sufficient for the sweep to complete.

Return Type
Boolean

Default
False

Examples
```vbnet
Set powerCalibrator = pna.SourcePowerCalibrator
powerCalibrator.UsePowerSensorFrequencyLimits = True 'Write
FreqCheck = powerCalibrator.UsePowerSensorFrequencyLimits 'Read
```

C++ Syntax
HRESULT put_UsePowerSensorFrequencyLimits(VARIANT_BOOL bState);
HRESULT get_UsePowerSensorFrequencyLimits(VARIANT_BOOL *bState);

Interface
ISourcePowerCalibrator
Write/Read

**UserRange Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Assigns the marker to the specified User Range. This restricts the marker’s x-axis travel to the User Range span, specified with <strong>Start</strong> and <strong>Stop</strong> values.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>- Each channel has 16 user ranges.</td>
</tr>
<tr>
<td></td>
<td>- Markers and trace statistics can be restricted to any user range.</td>
</tr>
<tr>
<td></td>
<td>- More than one marker can occupy a user range.</td>
</tr>
<tr>
<td></td>
<td>- User ranges can overlap. For example:</td>
</tr>
<tr>
<td></td>
<td>- User range 1: 3 GHz to 5 GHz</td>
</tr>
<tr>
<td></td>
<td>- User range 2: 4 GHz to 6 GHz</td>
</tr>
</tbody>
</table>

**Note:** User ranges are especially useful in restricting marker searches to specific areas of the measurement.

**VB Syntax**

```vbnet
mark.UserRange = value
```

**Variable**

<table>
<thead>
<tr>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>mark</strong></td>
</tr>
<tr>
<td><strong>value</strong></td>
</tr>
</tbody>
</table>

**Return Type**

Long Integer

**Default**

0 - Full Span

**Examples**

```vbnet
mark.UserRange = 1 'Write
UserRange = mark.UserRange 'Read
```

**C++ Syntax**

```cpp
HRESULT get_UserRange(long *pRangeNumber)
HRESULT put_UserRange(long lRangeNumber)
```

**Interface**

IMarker
UserRangeMax Property

### Description

**Note:** This property on the Channel Object is superseded by the same property on the Measurement Object.

Sets the stimulus stop value for the specified User Range.

### VB Syntax

```vbnet
chan.UserRangeMax(domainType, Rnum) = value - Superseded
meas(UserRangeMax(Rnum) = value
mark.UserRangeMax(Rnum) = value
```

### Variable (Type) - Description

- **chan** A Channel (object) - Superseded
- **meas** A Measurement (object)
- **mark** A Marker (object)

To assign a marker to a User Range, use the UserRange Property.

- **domainType** (enum NADomainType) - Choose from:
  0 - naDomainFrequency
  1 - naDomainTime
  2 - naDomainPower

- **Rnum** (long integer) - User Range number. Choose any number between 1 and 16 (0=Full Span)

- **value** (double) - Stop value. Choose any number within the full span of the channel.

### Return Type

Double

### Default

The current stimulus setting for the channel

### Examples

```plaintext
mark.UserRangeMax(1) = 3e9 'Write
meas.UserRangeMax(1) = 3e9 'Write
UserRangeMax = mark.UserRangeMax(1) 'Read
UserRangeMax = meas.UserRangeMax(2) 'Read
```

### C++ Syntax

```c
HRESULT put_UserRangeMax(long rangeNumber, double maxValue)
HRESULT get_UserRangeMax(long rangeNumber, double *maxValue)
```

1631
Interface
  IMeasurement
  IMarker

Last Modified:

  3-May-2011  Updated for measurement object
## UserRangeMin Property

### Description
Sets the stimulus start value for the specified User Range. This property uses different arguments for the channel and marker objects.

### VB Syntax
```
chan.UserRangeMin(domainType, range) = value
or
mark.UserRangeMin(range) = value
```

### Variables
- `chan` A Channel (object)
- `mark` A Marker (object)

To assign a marker to a User Range, use the UserRange Property.

- `domainType` (enum NADomainType) Type of sweep currently implemented on the channel - Choose from:
  - 0 - naDomainFrequency
  - 1 - naDomainTime
  - 2 - naDomainPower
  - 3 - naDomainPhase

- `range` (long) - User Range number. Choose any number between 1 and 16 (0=Full Span)

- `value` (double) - Start value. Choose any number within the full span of the analyzer

### Return Type
Double

### Default
The current stimulus setting for the channel

### Examples
```
mark.UserRangeMin(1) = 3e9 'Write
chan.UserRangeMin(naDomainFrequency, 1) = 3e9 'Write

UserRngeMin = mark.UserRangeMin 'Read
UserRngeMin = chan.UserRangeMin 'Read
```

### C++ Syntax
- HHRESULT put_UserRangeMin(tagNADomainType domain, long rangeNumber, double minValue)
- HHRESULT get_UserRangeMin(tagNADomainType domain, long rangeNumber, double *minValue)

Interface
- IChannel

Last Modified:
- 3-May-2011 Updated for phase
**UserDescriptionOfPNA Property**

**Description**
Sets and reads a description of the PNA used to perform the User Characterization. This description is stored with the characterization in the ECal module. Set this description before sending [Initialize](#) or the default (empty string) will be used.

**VB Syntax**
```
userChar.UserDescriptionOfPNA = value
```

**Variable**
*userChar* (Type) - Description

- **userChar**: An ECalUserCharacterizer Object
- **value**: (String) Descriptive text, limited to 14 characters maximum.

**Return Type**
String

**Default**
"" (Empty String)

**Examples**
```
userChar.UserDescriptionOfPNA = "My PNA"
```

**C++ Syntax**
```
HRESULT get_UserDescriptionOfPNA(BSTR *info);
HRESULT put_UserDescriptionOfPNA(BSTR info);
```

**Interface**
IECalUserCharacterizer

---

Last Modified:

2-Nov-2008 New topic (8.33)
UserName Property

**Description**
Sets and reads the description of the person and/or company who is producing the ECaI user characterization. This description is stored with the characterization in the ECaI module.

Set this description before sending Initialize or the default (empty string) will be used.

**VB Syntax**
```
userChar.UserName = value
```

**Variable**
*Type* - Description

*userChar* An ECaIUserCharacterizer Object

*value* (String) Descriptive text, limited to 19 characters maximum.

**Return Type**
String

**Default**
"" (Empty String)

**Examples**
```
userChar.UserName = "John Doe, Acme Inc."
```

**C++ Syntax**
```
HRESULT get_UserName(BSTR *name);
HRESULT put_UserName(BSTR name);
```

**Interface**
IECalUserCharacterizer
**UserPresetEnable Property**

**Description**
'Checks' and 'clears' the enable box on the User Preset dialog box. This only affects subsequent Presets from the front panel user interface.

Regardless of the state of the User Preset Enable checkbox, the `app.Preset` command will always preset the PNA to the factory preset settings, and `app.UserPreset` will always perform a User Preset.

**VB Syntax**

```vbnet
app.UserPresetEnable = state
```

**Variable (Type) - Description**

- `app` An `Application` (object)
- `state` (boolean) Front Panel User Preset State. Choose from:
  - False – User Preset OFF
  - True – User Preset ON

**Return Type**
Boolean

**Default**
False

**Examples**

```vbnet
app.UserPresetEnable = True 'Write
upreset = app.UserPresetEnable 'Read
```

**C++ Syntax**

```cpp
HRESULT get_UserPresetEnable(VARIANT_BOOL *pVal)
HRESULT put_UserPresetEnable(VARIANT_BOOL newVal)
```

**Interface**
IApplication6
### Valid Property

**Description**
Returns a boolean value to indicate if the current equation on the measurement is valid. For equation processing to occur, the equation must be valid and ON.

**VB Syntax**
```vbnet
IsValid = eq.Valid
```

**Variable**
- **(Type) - Description**
  - `good` *(Boolean)* Variable to store the returned value.
    - **True** (1) - equation is valid
    - **False** (0) - equation is NOT valid
  
  - `eq` *(Equation (object))*

**Return Type**
Boolean

**Default**
Not Applicable

**Examples**
```vbnet
IsValid = eq.Valid 'Read
```

**C++ Syntax**
```cpp
HRESULT get_Valid(Boolean *equation)
```

**Interface**
IEquation

---

**Last Modified:**
18-Jun-2007

New topic
ValidConnectorTypes Property

Description

Returns a list of all connector types for which there are calibration kits. Looks for connector types in mechanical cal kits, within PNA disk memory, and within the attached ECal Module memory.

Here are the more common connector types:

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>W-band waveguide</td>
<td>Type B</td>
<td>1.00 mm female</td>
</tr>
<tr>
<td>V-band waveguide</td>
<td>Type A (50) female</td>
<td>1.00 mm male</td>
</tr>
<tr>
<td>U-band waveguide</td>
<td>Type A (50) male</td>
<td>1.85 mm male</td>
</tr>
<tr>
<td>R-band waveguide</td>
<td>Type F (75) female</td>
<td>1.85 mm female</td>
</tr>
<tr>
<td>Q-band waveguide</td>
<td>Type F (75) male</td>
<td>2.92 mm female</td>
</tr>
<tr>
<td>K-band waveguide</td>
<td>Type N (75) female</td>
<td>2.92 mm male</td>
</tr>
<tr>
<td>P-band waveguide</td>
<td>Type N (75) male</td>
<td>APC 2.4 female</td>
</tr>
<tr>
<td>X-band waveguide</td>
<td>Type N (50) female</td>
<td>APC 2.4 male</td>
</tr>
<tr>
<td>7-16 female</td>
<td>Type N (50) male</td>
<td>APC 3.5 female</td>
</tr>
<tr>
<td>7-16 male</td>
<td>APC 3.5 male</td>
<td>APC 7</td>
</tr>
</tbody>
</table>

VB Syntax

value = obj.ValidConnectorTypes

Variable (Type) - Description

value (Variant) List of connector types

obj Any of the following:

- ECalUserCharacterizer (object)
- GuidedCalibration (object)
- SMCType (object)
- VMCType (object)

Return Type Variant

Default Not Applicable

Examples

value = SMC.ValidConnectorTypes

C++ Syntax

HRESULT get_ValidConnectorTypes(VARIANT* connectorTypes);
Interface: IGuidedCalibration
   SMCType
   VMCType

Last Modified:

23-Feb-2011   Added list

1-Sep-2009    Added ECal User and within disk memory (A.09.00)
## Value Property

**Description**  
Write or read a value (setting) for the current element.  
See a [list of configurable elements and settings](#) for various PNA models.

**VB Syntax**  
`pathElement.Value = value`

**Variable**  
*Type* - Description

- `pathElement`  
  A [PathElement](#) (object)

- `value`  
  *(String)* Value for the element. Use `pathElement.Values` to return a list of valid settings for this element.

**Return Type**  
String

**Default**  
Not Applicable

**Examples**  
Set the "Combiner" element to value "Reversed"

```vbnet
chan.PathConfiguration.Element("Combiner").Value = "Reversed"
setting=pathElement.Value
```

**C++ Syntax**  
```cpp
HRESULT get_Value( BSTR* pValue );
HRESULT put_Value( BSTR value );
```

**Interface**  
IPathElement

---

Last Modified:  
14-Dec-2006  
MX New topic
Values Property

Description
Returns an array of valid settings that can be used with the element object.
See a list of configurable elements and settings for various PNA models.

VB Syntax
values = pathElement.Values

Variable (Type) - Description
values (Variant array) Variable to store the array of valid settings for the element.

pathElement A PathElement (object)

Return Type Variant array

Default Not Applicable

Examples settings=pathElement.Values

C++ Syntax
HRESULT Values( VARIANT* pValues );

Interface IPathElement

Last Modified:
14-Dec-2006 MX New topic
### VelocityFactor Property

**Description**
Sets the velocity factor to be used with Electrical Delay, Port Extensions, and Time Domain marker distance calculations.

**VB Syntax**
```vbnet
app.VelocityFactor = value
```

**Variable**

- **app** (An Application (object))
- **value** (double) - Velocity factor. Choose a number between: 0 and 10 (.66 polyethylene dielectric; .7 PTFE dielectric)

**Return Type**
Double

**Default**
1

**Examples**
```vbnet
app.VelocityFactor = .66 'Write
RelVel = app.VelocityFactor 'Read
```

**C++ Syntax**
```cpp
HRESULT get_VelocityFactor(double *pVal)
HRESULT put_VelocityFactor(double newVal)
```

**Interface**
IApplication
**View Property**

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets (or returns) the type of trace displayed on the screen.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>meas.View = value</code></td>
</tr>
<tr>
<td><strong>Variable</strong> (Type) - Description</td>
<td></td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A measurement (object)</td>
</tr>
<tr>
<td><code>value</code> (enum NAView) - Type of trace. Choose from:</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>naData</td>
</tr>
<tr>
<td>1</td>
<td>naDataAndMemory</td>
</tr>
<tr>
<td>2</td>
<td>naMemory</td>
</tr>
<tr>
<td>3</td>
<td>naNoTrace</td>
</tr>
</tbody>
</table>

**Note:** The naData trace may reflect the result of a TraceMath operation.

<table>
<thead>
<tr>
<th>Return Type</th>
<th>NAView</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Default</strong></td>
<td>naData</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>meas.View = naData 'Write</code></td>
</tr>
<tr>
<td></td>
<td><code>trcview = meas.View 'Read</code></td>
</tr>
</tbody>
</table>

**C++ Syntax**

HRESULT get_View(tagNAView* pView)
HRESULT put_View(tagNAView newView)

**Interface**

IMeasurement
Write/Read

Visible Property

<table>
<thead>
<tr>
<th>Description</th>
<th>Makes the Network Analyzer application visible or not visible. In the Not Visible state, the analyzer cycle time for making measurements can be significantly faster because the display does not process data.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>app.Visible = state</code></td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean)</td>
</tr>
<tr>
<td>False</td>
<td>Network Analyzer application NOT visible</td>
</tr>
<tr>
<td>True</td>
<td>Network Analyzer application IS visible</td>
</tr>
<tr>
<td>Return Type</td>
<td>Boolean</td>
</tr>
<tr>
<td>Default</td>
<td>True</td>
</tr>
<tr>
<td>Examples</td>
<td><code>app.Visible = False</code> 'Write</td>
</tr>
<tr>
<td><code>vis = app.Visible</code></td>
<td>'Read</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT get_Visible(VARIANT_BOOL * bVisible)</td>
</tr>
<tr>
<td></td>
<td>HRESULT put_Visible(VARIANT_BOOL bVisible)</td>
</tr>
<tr>
<td>Interface</td>
<td>IApplication</td>
</tr>
</tbody>
</table>
### WGCutoffFreq Property

**Description**
Sets or returns the value of the waveguide cut off frequency.

**VB Syntax**
`meas.WGCutoffFreq = value`

**Variable**
- **(Type)** - Description
  - *meas* A Measurement *(object)*
  - *value* *(double)* - Frequency in Hertz.

**Return Type**
Double

**Default**
Not Applicable

**Examples**
`Print meas.WGCutoffFreq` 'prints the value of the waveguide cut off frequency'

**C++ Syntax**
- `HRESULT get_WGCutoffFreq(double *pVal);`
- `HRESULT put_WGCutoffFreq(double newVal);`

**Interface**
`IMeasurement2`
WideBandDectionState Property

Description: Set and read the pulse mode detection method.

VB Syntax: `pulseMeas.WideBandDectionState = bool`

Variable (Type) - Description:
- `pulseMeas` A PulseMeasurementControl (object)

- `bool` False - Narrowband mode.
  True - Wideband mode

Return Type: Boolean

Default: Based on pulse width.

Examples:
- `pulse.WideBandDectionState = True` Write
- `value = pulse.WideBandDectionState` Read

C++ Syntax:
- `HRESULT get_WideBandDectionState(VARIANT_BOOL *pVal);`
- `HRESULT put_WideBandDectionState(VARIANT_BOOL newVal);`

Interface: IPulseMeasurementControl

Last Modified:
- 11-Mar-2010 New topic
Write/Read

About PNA-X Pulsed Capabilities

Width Property

Description
Sets the pulse width - the amount of time that the pulse is ON.

VB Syntax
pulse.Width (n) = value

Variable (Type) - Description

pulse A PulseGenerator (object)

n (Integer) Pulse generator number. Choose from 0 to 4.
0 is the generator that pulses the ADC.

value (Double) Pulse width in seconds. Choose a value from about 33ns to about 70 seconds.

Return Type Double

Default 1e-4 sec

Examples
pulse.Width = 1ms 'Write
value = pulse.Width 'Read

C++ Syntax
HRESULT get_Width (integer pulse, double* width);
HRESULT put_Width (integer pulse, double width);

Interface IPulseGenerator

Last Modified: 2-Jan-2007  MX New topic
**Read-only**

**WindowNumber Property**

**Description**
Returns the window number. You might use this property to identify a particular window so that you can create a new Measurement in that window.

**VB Syntax**
`value = win.WindowNumber`

**Variable**
(Type) - Description

- `win`  A NAWindow (object)

- `value`  (long integer) - Variable to store the returned window number

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
`value = app.ActiveNAWindow.WindowNumber`

**C++ Syntax**
`HRESULT (long* windowNumber);`

**Interface**
INAWindow
WindowState Property

Description Sets or returns the window setting of Maximized, Minimized, or Normal. To arrange all of the windows, use app.ArrangeWindows.

VB Syntax object.WindowState = value

Variable (Type) - Description

object An Application (object) - main window
or
A NaWindow (object) - data windows

value (enum NAWindowStates) - The window state. Choose from:
0 - naMinimized - Minimizes the window to an Icon on the lower toolbar
1 - naMaximized - Maximizes the window
2 - naNormal - changes the window size to the user defined setting (between Max and Min).

Return Type Long Integer

Default naMaximized

Examples appWindowState = naMinimized 'changes the Network Analyzer application window to an icon. -Write
winWindowState = naNormal 'changes the window defined by the win object variable to user defined settings. -Write

Winstate = appWindowState 'Read

C++ Syntax HRESULT get_WindowState(tagNAWindowStates *pVal)
HRESULT put_WindowState(tagNAWindowStates newVal)

Interface INAWindow
IApplication
## XAxisAnnotation Property

**Description**
Returns the X-Axis annotation of the specified tuning sweep.

**VB Syntax**
```
value = embedLODiag.XAxisAnnotation (n)
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(String) Variable to store the returned data.</td>
</tr>
</tbody>
</table>

**embedLODiag**
An **EmbeddedLODiagnostic** (object)

<table>
<thead>
<tr>
<th>n</th>
<th>(Long) Tuning sweep number. Use NumberOfSweeps to find the number of sweeps taken.</th>
</tr>
</thead>
</table>

**Default**
Not Applicable

**Examples**
```
data= embedLO.XAxisAnnotation 3 'read
```

**C++ Syntax**
```
HRESULT XAxisAnnotation (long sweep, BSTR* annotation);
```

**Interface**
IEembedLODiagnostic

---

Last Modified:

13-Apr-2007  MX New topic
XAxisPointSpacing Property

**Description**
Sets X-axis Point Spacing for the display traces measured with segment sweeps on the active channel.

**VB Syntax**
`chan.XAxisPointSpacing = value`

**Variable**  
**Type** - Description
- `chan` A Channel (object)
- `value` (Enum as `naStates`) - Choose from:
  - 0 - `naOFF` - Turns X-axis Point Spacing OFF
  - 1 - `naON` - Turns X-axis Point Spacing ON

**Return Type**
Enum

**Default**
0 - `naOFF`

**Examples**
- `chan.XAxisPointSpacing = naOFF`  'Write
- `xspac = chan.XAxisPointSpacing`  'Read

**C++ Syntax**
```cpp
HRESULT get_XAxisPointSpacing (tagNAStates *pState);
HRESULT put_XAxisPointSpacing (tagNAStates newState);
```

**Interface**
IChannel2
**XAxisStart Property**

**Description**
Returns the X-Axis start value of the specified tuning sweep.

**VB Syntax**

```vbnet
value = embedLODiag.XAxisStart(n)
```

**Variable**

- **value** (Double) Variable to store the returned data.
- **embedLODiag** An EmbeddedLODiagnostic (object)
- **n** (Long) Tuning sweep number. Use NumberOfSweeps to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**

```vbnet
data= embedLO.XAxisStart 3   'read
```

**C++ Syntax**

```c++
HRESULT XAxisStart(long sweep, double* start);
```

**Interface**
IEembedLODiagnostic

---

Last Modified:

13-Apr-2007  MX New topic
### XAxisStop Property

**Description**

Returns the X-Axis stop value of the specified tuning sweep.

**VB Syntax**

```
value = embedLODiag.XAxisStop (n)
```

**Variable (Type) - Description**

- **value** (Double) Variable to store the returned data.
- **embedLODiag** An `EmbeddedLODiagnostic` (object)
- **n** (Long) Tuning sweep number. Use `NumberOfSweeps` to find the number of sweeps taken.
- **Default** Not Applicable

**Examples**

```
data= embedLO.XAxisStop 3 'read
```

**C++ Syntax**

```
HRESULT XAxisStop (long sweep, double* start);
```

**Interface**

`IEmbededLODiagnostic`

---

Last Modified:

13-Apr-2007   MX New topic
### YAxisAnnotation Property

**Description**
Returns the Y-Axis annotation of the specified tuning sweep.

**VB Syntax**
```vbnet
value = embedLODiag.YAxisAnnotation(n)
```

**Variable (Type) - Description**

- **value (String)** Variable to store the returned data.
- **embedLODiag (An EmbeddedLODiagnostic (object))**
  
  - **n (Long)** Tuning sweep number. Use `NumberOfSweeps` to find the number of sweeps taken.

**Default**
Not Applicable

**Examples**
```vbnet
data= embedLO.YAxisAnnotation 3 'read
```

**C++ Syntax**
```cpp
HRESULT YAxisAnnotation (long sweep, BSTR* annotation);
```

**Interface**
IEembedLODiagnostic

---

Last Modified: 13-Apr-2007  MX New topic
YScale Property

Description
Sets or returns the Y-axis Per-Division value of the active trace.

VB Syntax
```
trace.YScale = value
```

Variable
- **trace** - A Trace (object)
- **value** - (double) - Scale /division number. Units and range depend on the current data format.

Return Type
Double

Default
10 (db)

Examples
```
trac.YScale = 5 'Write
yscl = trac.YScale 'Read
```

C++ Syntax
```
HRESULT get_YScale(double *pVal)
HRESULT put_YScale(double newVal)
```

Interface
ITrace
# Z0 Property

**Description**  
Sets and Returns the characteristic impedance for the calibration standard.

**VB Syntax**  
```
calstd.Z0 = value
```

**Variable**  
- **calstd**: A CalStandard object. Use calKit.GetCalStandard to get a handle to the standard.
- **value**: (single) - Impedance in Ohms

**Return Type**  
Single

**Default**  
Not Applicable

**Examples**  
```
calstd.Z0 = 50 'Write
impedance = calstd.Z0 'Read
```

**C++ Syntax**  
- HRESULT get_Z0(float *pVal)
- HRESULT put_Z0(float newVal)

**Interface**  
ICalStandard
Abort Method

Description
Ends the current measurement sweep on the channel.

VB Syntax
chan.Abort [sync]

Variable (Type) - Description

chan (object) - A Channel object
sync (boolean) - wait (or not) for the analyzer to stop before processing subsequent commands. Optional argument; if unspecified, value is set to False. Choose from: True - synchronize - the analyzer will not process subsequent commands until the current measurement is aborted. False - continue processing commands immediately

Return Type
None

Default
None

Examples
chan.abort True
chan.abort

C++ Syntax
HRESULT Abort(VARIANT_BOOL bSynchronize);

Interface IChannel
AbortPowerAcquisition Method

**Description**
Aborts a source power cal acquisition sweep that is currently in progress.

**VB Syntax**
`powerCalibrator.AbortPowerAcquisition`

**Variable**
`powerCalibrator` *(object)* - A `SourcePowerCalibrator` object

**Return Type**
None

**Default**
Not Applicable

**Examples**
`powerCalibrator.AbortPowerAcquisition`

**C++ Syntax**
`HRESULT AbortPowerAcquisition();`

**Interface**
`ISourcePowerCalibrator`
Write-only

AcquireCalStandard Method - Superseded

About Calibration Standards

Description  
Note: This command has been replaced by AcquireCalStandard2 Method, which provides for acquisition of sliding load standards. All other functionality is identical.

VB Syntax  
`cal.AcquireCalStandard std[,index]`

Variable  
(Type) - Description

cal  
A Calibrator (object)

std  
(enum NACalClass) Standard to be measured. Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

SOLT Standards
1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

TRL Standards
1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
5 - naTRL_Isolation

index  
(long integer) number of the standard. Optional argument - Used if there is more than one standard required to cover the necessary frequency range. If unspecified, value is set to 1.
**Note** The behavior has changed with PNA revisions as follows:

- Before 6.01: Accepted 0 and changed it to 1
- 6.01 to 6.04: Did NOT accept 0
- 6.04.11 and higher: Accepts 0 and changes it to 1

**Return Type** None

**Default** Not Applicable

**Examples**
```
Cal.AcquireCalStandard naSOLT_Thru 'Write
```

**C++ Syntax**
```
HRESULT AcquireCalStandard(tagNACalClass enumClass, short standardNumber)
```

**Interface** ICaliibrator

Last modified:

10/05/06 Modified Index argument.
AcquireCalStandard2 Method

**Description**
Measures the specified standard from the selected calibration kit. The calibration kit is selected using `app.CalKitType`.

For 2-port calibration, it is also necessary to specify direction with `AcquisitionDirection`.

To omit Isolation from a 2-port calibration, do not Acquire a cal standard for `naSOLT_Isolation`.

For using two sets of standards, see `Simultaneous2PortAcquisition Property`.

**Note:** This command replaces `AcquireCalStandard`. This command provides for the acquisition of a sliding load cal. All other functionality is identical.

**VB Syntax**
```
cal.AcquireCalStandard2 std[,index][,slide]
```

**Variable (Type) - Description**

- `cal` A `Calibrator (object)`
- `std` (enum `NACalClass`) Standard to be measured. Choose from:
  1. `naClassA`
  2. `naClassB`
  3. `naClassC`
  4. `naClassD`
  5. `naClassE`
  6. `naReferenceRatioLine`
  7. `naReferenceRatioThru`

**SOLT Standards**

- 1. `naSOLT_Open`
- 2. `naSOLT_Short`
- 3. `naSOLT_Load`
- 4. `naSOLT_Thru`
- 5. `naSOLT_Isolation`

**TRL Standards**

- 1. `naTRL_Reflection`
- 2. `naTRL_Line_Reflection`
- 3. `naTRL_Line_Tracking`
4 - naTRL_Thru

5 - naTRL_Isolation

[index] (long  integer) Number of the standard. Optional argument - Used if there is more than one standard required to cover the necessary frequency range. If unspecified, value is set to 1.

**Note** The behavior has changed with PNA revisions as follows:

- Before 6.01: Accepted 0 and changed it to 1
- 6.01 to 6.04: Did NOT accept 0
- 6.04.11 and higher: Accepts 0 and changes it to 1

[slide] (enum as NACalStandardSlidingState) Optional argument. State of the sliding load. The slide should be set a minimum of five times. Seven is the maximum that can be stored. Choose from:

0 - naNotSlidingStd - not using a sliding load - Default if not specified.
1 - naSlidelsSet - slide is set for acquisition
2 - naSlidelsDone - this next acquisition will be the last. Calculations will then be performed.

**Return Type** None

**Default** Not Applicable

**Examples**

Cal.AcquireCalStandard2 naSOLT_Thru
Cal.AcquireCalStandard2 naSOLT_Thru,2,naNotSlidingStd

'measures the second standard listed in the class of naSOLT_Thru'

**C++ Syntax**

HRESULT AcquireCalStandard2(NACalClass enumClass, long standardPosition, NACalStandardSlidingState slidingStandardState)

**Interface** ICalibrator

---

Last modified:

10/05/06 Modified Index argument.
AcquireCalConfidenceCheckECALEx Method

Description
This method replaces AcquireCalConfidenceCheckECAL. Transfers confidence data from the specified ECal module into the measurement's memory trace. The data is transferred to the specified S-parameter on the same channel as this Calibrator object.

The characterization within the ECal module that the confidence data will be read from is specified by ECALCharacterizationEx. The default value is 0.

Note: A confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

VB Syntax
`cal.AcquireCalConfidenceCheckECALEx Sparam [,ecalModule]`

Variable (Type) - Description
- **cal** (object) A Calibrator
- **Sparam** (String) S-parameter to transfer confidence data to. This parameter must be present on the same channel as the calibrator object.
- **ecalModule** (Integer) – Optional argument. ECal module. Choose from modules 1 through 8. Use IsECALModuleFoundEx to determine the number of modules connected to the PNA. Use GetECALModuleInfoEx to return the model and serial number of each module.

Return Type None

Default Not applicable

Examples
`Cal.AcquireCalConfidenceCheckECALEx "S11", 2`

C++ Syntax
`HRESULT AcquireCalConfidenceCheckECALEx(BSTR strParameter, long moduleNumber = 1);`

Interface ICalibrator4

Last Modified: 15-Jun-2010

Added Note
### About Source Power Cal

**AcquirePowerReadingsEx Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>This command replaces AcquirePowerReadings Method. Initiates a source power cal acquisition.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>powerCalibrator.AcquirePowerReadingsEx calMethod, acqdevice [,sync]</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>powerCalibrator</code></td>
<td>(object) - A <code>SourcePowerCalibrator</code> object</td>
</tr>
<tr>
<td><code>calMethod</code></td>
<td>(enum <code>NASourcePowerCalMethod</code>) Selects the calibration method to be used for the source power cal acquisition.</td>
</tr>
<tr>
<td>0 – <code>naPowerMeter</code></td>
<td>Use power meter for all readings.</td>
</tr>
<tr>
<td>1 - <code>naPowerMeterAndReceiver</code></td>
<td>Power meter for the first iteration; then use the reference receiver for remaining readings if necessary.</td>
</tr>
<tr>
<td>2 - <code>naReceiver</code></td>
<td>Use PNA measurement receiver for all readings.</td>
</tr>
<tr>
<td><code>acqdevice</code></td>
<td>(String) The specific acquisition device to be used. NOT case sensitive. Choose from: If <code>calMethod = naPowerMeter</code> or <code>naPowerMeterAndReceiver</code>, choose from:</td>
</tr>
<tr>
<td></td>
<td>- “ASEN” -- Sensor on power meter channel A.</td>
</tr>
<tr>
<td></td>
<td>- “BSEN” -- Sensor on power meter channel B.</td>
</tr>
<tr>
<td></td>
<td>- To use the sensor that currently corresponds to the frequency of interest, use the value from the <code>PowerAcquisitionDevice</code> property.</td>
</tr>
<tr>
<td></td>
<td>If <code>calMethod = naReceiver</code>, choose from:</td>
</tr>
<tr>
<td></td>
<td>- The receiver names for your specific PNA using either physical receiver notation or logical receiver notation. For example, &quot;a1&quot; or &quot;A&quot;.</td>
</tr>
<tr>
<td></td>
<td>- Any configured PMAR device name. Learn more about PMAR Devices. See PMAR commands.</td>
</tr>
<tr>
<td><code>[sync]</code></td>
<td>(boolean) Optional argument. If not specified, value is set to False. Choose from:</td>
</tr>
<tr>
<td>True (1)</td>
<td>The method does not return until this acquisition has completed (the program calling this method is halted while waiting for the method to return).</td>
</tr>
<tr>
<td>False (0)</td>
<td>The method initiates an acquisition then returns immediately (while the acquisition still proceeds). The program calling this method can then perform other operations during the acquisition.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>None</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>
Examples

```csharp
powerCalibrator.AcquirePowerReadingsEx naPowerMeter, "asen", True

powerCalibrator.AcquirePowerReadingsEx naReceiver, "b2"

powerCalibrator.AcquirePowerReadingsEx naReceiver, "MyPMAR"
```

**C++ Syntax**

```c++
HRESULT AcquirePowerReadingsEx (tagNASourcePowerCalMethod enumCalMethod, BSTR bstrAcqDevice, VARIANT_BOOL bSync);
```

**Interface**

ISourcePowerCalibrator4

---

Last modified:

- 10-Feb-2011  Added PMAR
- 9/12/06  MQ New command to accommodate receiver only SPC
AcquireStep Method

**Description**
Acquire the measurement data for the specified step in the calibration process. For an ECal User characterization this measures the ECal module.

*Note:* Guided Cal allows you to measure standards in any order. See an example.

**VB Syntax**

```vbnet
obj.AcquireStep (n)
```

**Variable (Type) - Description**

- **obj**
  Any of the following:
  - `GuidedCalibration` (object)
  - `SMCType` (object)
  - `VMCType` (object)
  - `ECalUserCharacterizer` (object) - Currently, only ONE step is required to measure the ECal module.

- **n**
  Step number in the calibration process.
  Use `GenerateSteps` to determine the total number of steps.
  Use `GetStepDescription` to read the description of each step.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vbnet
VMC.AcquireStep (3)
```

**C++ Syntax**

```cpp
HRESULT put_AcquireStep(long step);
```

**Interface**

- `SMCType`
- `VMCType`
- `IGuidedCalibration`
- `IECalUserCharacterizer`

---

**Last Modified:**

- 4-Nov-2008  Added ECalUserChar object
- 20-Jan-2007  Added any order note.
Write-only

Activate Method

**Description**
Makes an object the Active Object. When making a measurement active, the channel and window the measurement is contained in becomes the active channel and active window.

In order to change properties on any of the active objects, you must first have a "handle" to the active object using the **Set** command. For more information, See [Getting a Handle to an Object](#).

You do not have to make an object "Active" to set or read its properties remotely. But an object must be "Active" to change its values from the front panel.

**VB Syntax**
```
object.Activate
```

**Variable**

**Type** - **Description**

- `object` **Measurement** *(object)*
- or
- `Marker` *(object)*

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
meas.Activate
mark.Activate
```

**C++ Syntax**
```
HRESULT Activate()
```

**Interface**
IMeasurement
IMarker
Write-only

ActivateMarker Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Makes a marker the Active Marker. Use <code>meas.ActiveMarker</code> to read the number of the active marker.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>meas.ActivateMarker(Mnum)</code></td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement (object)</td>
</tr>
<tr>
<td><code>Mnum</code></td>
<td>(long integer) - the number of the marker to make active. Choose any marker number from 1 to 9.</td>
</tr>
<tr>
<td>Return Type</td>
<td>None</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>meas.ActivateMarker(1)</code> 'Write`</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT ActivateMarker(long IMarkerNumber)</td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement</td>
</tr>
<tr>
<td>Remarks</td>
<td>Use <code>ReferenceMarkerState</code> to control the Reference marker.</td>
</tr>
</tbody>
</table>
ActivateWindow Method

Description  Makes a window object the Active Window.

In order to change properties on any of the active objects, you must first have a "handle" to the active object using the Set command. For more information, See Programming the Analyzer Object Model.

You do not have to make an object "Active" to set or read its properties remotely. But an object must be "Active" to change its values from the front panel.

VB Syntax  

`app.ActivateWindow n`

Variable  

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An <strong>Application</strong> (object)</td>
</tr>
<tr>
<td><code>n</code></td>
<td>(long) Number of the window to make active</td>
</tr>
</tbody>
</table>

Return Type  Window Object

Default  Not Applicable

Examples  

`app.ActivateWindow 4`

C++ Syntax  

`HRESULT ActivateWindow(long WindowNumber)`

Interface  IApplication

See the PNA Object Model
### Add (channels) Method

**Description**  
Creates a channel and returns a handle to it. If the channel already exists, it returns the handle to the existing channel.

**VB Syntax**  
`chans.Add (item)`

**Variable** (Type) - Description

- `chans` A Channel collection (object)
- `item` (variant) - Channel number.

**Return Type**  
Channel

**Default**  
Not Applicable

**Examples**  
`chans.Add 3 'Creates channel 3`

**C++ Syntax**  
`HRESULT Add(VARIANT numVal, IChannel** pChannel)`

**Interface**  
IChannels
Write-only

Add (measurement) Method

**Description**  Adds a Measurement to the collection.

**VB Syntax**  `meas.Add channel,param,source[,window]`

- `meas`  A **Measurements** collection **(object)**
- `channel`  **(long)** - Channel number of the new measurement.

**For S-parameters:**

- **Any S-parameter that can be measured by your PNA.**
  - Single-digit port numbers can be separated by "_" (underscore). For example: "S21" or "S2_1"
  - Double-digit port numbers MUST be separated by underscore. For example: "S10_1"

**For Ratioed measurements:**

- Any two receivers in your PNA separated by "/". For example: "A/R1"
  - See the [block diagram](#) showing the receivers in YOUR PNA.

**For Unratioed (absolute power) measurements:**

- Any receiver in the PNA. For example: "A"
  - See the [block diagram](#) showing the receivers in YOUR PNA

With PNA Rev 6.2, **Ratioed** and **Unratioed** measurements can also use **logical receiver notation** to refer to receivers. This notation makes it easy to refer to receivers with an **external test set** connected to the PNA. You do not need to know which physical receiver is used for each test port. [Learn more](#).

**For ADC measurements**

- Any ADC receiver in the PNA followed by a comma, then the source port.
  - For example: "AI1_2" indicates the Analog Input1 with source port of 2.
  - [Learn more about ADC receiver measurements](#).

**For Balanced S-parameter measurements:**

- "*topology:Sabxy*"
  - `topology` - Choose from:
  - `sbal` - single-ended to balanced
  - `ssb` - single-ended / single-ended to balanced
  - `bbal` - balanced to balanced
Sabxy -

Where

a - device output (receive) mode
b - device input (source) mode

(choose from the following for both a and b):

- d - differential
- c - common
- s - single ended

x - device output (receive) logical port number
y - device input (source) logical port number

For example: "sbal:sdd42"

See an example program

For Imbalance and Common Mode Rejection measurements:

"topology:parameter" Choose from:

<table>
<thead>
<tr>
<th>Choose this:</th>
<th>To get this:</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;SBAL:IMBSB&quot;</td>
<td>single-ended to balanced</td>
<td>imbalance</td>
</tr>
<tr>
<td>&quot;SBAL:CMRRSB1&quot;</td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds21/Scs21)</td>
</tr>
<tr>
<td>&quot;SBAL:CMRRSB2&quot;</td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Ssd12/Ssc12)</td>
</tr>
<tr>
<td>&quot;SSB:IMB1SSB&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 1</td>
</tr>
<tr>
<td>&quot;SSB:IMB2SSB&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 2</td>
</tr>
<tr>
<td>&quot;SSB:CMRRSSB1&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds31/Scs31)</td>
</tr>
<tr>
<td>&quot;SSB:CMRRSSB2&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds32/Scs32)</td>
</tr>
<tr>
<td>&quot;BBAL:IMB1BB&quot;</td>
<td>balanced to balanced</td>
<td>imbalance 1</td>
</tr>
<tr>
<td>&quot;BBAL:IMB2BB&quot;</td>
<td>balanced to balanced</td>
<td>imbalance 2</td>
</tr>
<tr>
<td>&quot;BBAL:CMRRBB&quot;</td>
<td>balanced to balanced</td>
<td>common mode rejection (Sdd21/Scc21)</td>
</tr>
</tbody>
</table>

source (long integer) - Source port number; if unspecified, value is set to 1. Only used for non-s-parameter measurements; ignored if s-parameter.

window (long integer) - Optional argument. Window number of the new measurement. If unspecified, the S-Parameter will be created in the Active Window. Choose between 1 and the maximum number of windows allowed on the PNA. If unspecified, the measurement will be created in the Active Window.

See also Traces, Channels, and Windows on the PNA

Return Type None

Default None

Examples meass.Add 3,"A/R1",1,1 ' Adds A/R1 measurement to channel 3 in window 1

C++ Syntax HRESULT Add(long ChannelNum, BSTR strParameter, long srcPort, VARIANT_BOOL bNewWindow)

Interface IMeasurements

Last modified:

9/12/06 MQ Added logical receiver notation and number of windows.
Add (NAWindows) Method

**Description**
Add a window to the display. Does not add a measurement. The window number must not already exist.

**VB Syntax**
```
wins.Add [item]
```

**Variable (Type) - Description**
- `wins` A NAWindow collection (object)
- `item` (variant) - Window number. Choose between 1 and the maximum number of windows allowed on the PNA.

See also Traces, Channels, and Windows on the PNA.

**Return Type**
Object

**Default**
Not Applicable

**Examples**
```
wins.Add 3 'Creates a window number 3
```

**C++ Syntax**
```
HRESULT Add(long windowNumber )
```

**Interface**
INAWindows

---

Last modified:
- 4-Mar-2009  Removed optional argument
- 9/12/06      Modified for number of windows
Add (PowerLossSegment) Method

**Description**
Adds a PowerLossSegment to the PowerLossSegments collection. Also, adds a PowerLossSegmentPMAR to the PowerLossSegmentsPMAR collection. To ensure predictable results, it is best to remove all segments before defining a new list of segments. For each segment in the collection, do a seg.Remove.

**VB Syntax**
```
segs.Add (item [ size])
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>segs</td>
<td>A PowerLossSegments collection</td>
</tr>
<tr>
<td></td>
<td>A PowerLossSegmentsPMAR collection.</td>
</tr>
<tr>
<td>item</td>
<td>(variant) - Number of the new segment. If it already exists, a new segment is inserted at the requested position.</td>
</tr>
<tr>
<td>size</td>
<td>(long integer) - Optional argument. The number of segments to add, starting with item. If unspecified, value is set to 1.</td>
</tr>
</tbody>
</table>

**Return Type**
None

**Default**
Not Applicable

**Examples**
```
segs.Add 1, 4 'Adds segments 1, 2, 3 and 4
```

**C++ Syntax**
```
HRESULT Add(VARIANT index, long size);
```

**Interface**
IPowerLossSegments

---

Last Modified:

25-Aug-2009  Added PMAR
## Add (PowerSensorCalFactorSegment) Method

**Description**  Adds a PowerSensorCalFactorSegment to the CalFactorSegments collection. Also adds a PowerSensorCalFactorSegmentPMAR to the CalFactorSegmentsPMAR collection.

To ensure predictable results, it is best to remove all segments before defining a new list of segments. For each segment in the collection, do a `seg.Remove`.

**VB Syntax**  

```vbnet
segs.Add (item [ size])
```

**Variable (Type) - Description**

- **segs**  A `CalFactorSegments` (collection) or a `CalFactorSegmentsPMAR` (collection)

- **item**  (variant) - Number of the new segment. If it already exists, a new segment is inserted at the requested position.

- **size**  (long integer) - Optional argument. The number of segments to add, starting with item. If unspecified, value is set to 1.

**Return Type**  None

**Default**  Not Applicable

**Examples**  

```plaintext
segs.Add 1, 4  'Adds segments 1,2,3 and 4
```

**C++ Syntax**  

```c++
HRESULT Add(VARIANT index, long size);
```

**Interface**  

- `ICalFactorSegments`
- `ICalFactorSegmentsPMAR`

---

Last Modified:

25-Aug-2009  Added PMAR
Add (segment) Method

Description
Adds segments to the Segments collection, but does not turn the segments ON.

VB Syntax
`segs.Add (item, [size])`

*segs* A segments collection *(object)*

*item* *(variant)* Number of the new segment. If it already exists, a new segment is inserted at the requested position.

*size* *(long integer)* Optional argument. The number of segments to add, starting with *item*. If unspecified, value is set to 1.

Return Type
None

Default
None

Examples
```vbnet
Segs.Add 1, 4 'Adds segments 1,2,3,and 4. (does NOT automatically turn segments ON)
```

C++ Syntax
`HRESULT Add(VARIANT index, long size);`

Interface
ISegments

Remarks
To ensure predictable results, it is best to remove all segments before defining a segment list. For each segment in the collection, do a seg.Remove.
**Add (External Device) Method**

**Description**

Adds an external device to the system. This is the same as clicking the **New** button and editing the name on the Configure an External Device dialog.

Upon creation, all settings on the new device are set to the defaults. The device is not active until set using `Ext.Dev.Active`.

**VB Syntax**

```
extDevices.Add name
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>extDevices</code></td>
<td>An <code>ExternalDevices</code> (collection)</td>
</tr>
<tr>
<td><code>name</code></td>
<td>(String) - Name of the new external device.</td>
</tr>
</tbody>
</table>

**Return Type**

Not Applicable

**Default**

Not Applicable

**Examples**

```
extDevices.Add 'MySource' 'Creates a new external device'
```

**C++ Syntax**

```
HRESULT Add (BSTR name)
```

**Interface**

`IExternalDevices`

---

Last Modified:

31-Jul-2009  MX New topic
Add (GuidedPowerSensors) Method

**Description**
Adds a power sensor name and item number to be used during a source power calibration. Use when multiple power sensors are to be used to calibrate the entire frequency span. The Name is used to recognize the sensor in the User Interface.

Item numbers in the **GuidedCalibrationPowerSensors** collection are used to refer to the power sensor remotely. Use the **Count Property** to return the number of power sensor items that are configured for use on the channel.

The port number to be calibrated is set using the **PerformPowerCalibration Property**.

[Learn about using multiple power sensors](#)

**VB Syntax**
sensors.Add(name)

**Variable**
- **(Type) - Description**
  
  *sensors* - A **GuidedCalibrationPowerSensors** (collection)

  *name* - (String) - Name of the power sensor to add. The power sensor must be already configured as a PMAR device using this name. [Learn how to remotely configure a PMAR device](#).

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
sensors.Add "pmar2"

[See Example program](#)

**C++ Syntax**
HRESULT Add(BSTR name);

**Interface**
IGuidedCalibrationPowerSensors

---

Last Modified:

8-Feb-2011  New topic
Add (Testset) Method

**Description**
Adds a testset to the ExternalTestsets Collection and loads the configuration file.

**VB Syntax**
```vbnet
testsets.Add (model,address)
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>testsets</td>
<td>An ExternalTestsets (collection)</td>
</tr>
<tr>
<td>model</td>
<td>(String) Model of the testset to be added, NOT case-sensitive. There is no COM command to read a list of currently-supported test sets. However, the following SCPI command can be used with the following format:</td>
</tr>
<tr>
<td>address</td>
<td>(Integer) Address of the testset to be added.</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
testsets.Add("Z5623AK66",12) ' add Z5623AK66 test at address 12 to testsets collection
```

See an example program

**C++ Syntax**
```cpp
HRESULT Add(BSTR typename, long address)
```

**Interface**
IExternalTestsets
## AddSegment Method

**Description**  Adds the specified number of segments to the scratch mixer at the index position. All segments are added with default settings.

**VB Syntax**  

```vbnet
conv.AddSegment index,count
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>conv</code></td>
<td>A Converter Object</td>
</tr>
<tr>
<td><code>index</code></td>
<td>(Long integer) Position at which to add segments. Valid index range is between 1 and the current segment count +1. Using count +1 adds segments to the end of the segment table. Use SegmentCount Property to read the current count in the Applied Mixer.</td>
</tr>
<tr>
<td><code>count</code></td>
<td>(Long integer) Optional argument. Number of segments to add. If unspecified, 1 segment is added.</td>
</tr>
</tbody>
</table>

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  

```vbnet
mxr.AddSegment 1,5 'Adds 5 segments beginning at the first position.
```

**See example program**

**C++ Syntax**  

```cpp
HRESULT AddSegment(long index, long count);
```

**Interface**  IConverter5

---

**Last Modified:** 26-Oct-2010

New command (A.09.33)
# AllowAllEvents Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets event filtering to monitor all events in the analyzer. This is the default setting when subscribing to events. This could slow the measurement speed of the analyzer significantly.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.AllowAllEvents</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.AllowAllEvents</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT AllowAllEvents()</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IApplication</td>
</tr>
</tbody>
</table>
AllowChannelToSweepDuringCalAcquisition Method

Description
Specifies the channel to sweep during a Calibration.

When this command is sent, the SwpChan channel is ‘flagged’ to be swept during calibration. The flag is cleared when the channel is deleted, if the Measurement Class is changed, or if all measurements are deleted from the channel. If the same channel number is recreated, this command must be sent again to sweep the channel during a calibration. The flag is NOT saved with an instrument state.

A Preset or Instrument State Recall deletes the channel.

VB Syntax
```
calMgr.AllowChannelToSweepDuringCalAcquisition (CalChan, SwpChan, State)
```

Variable
( Type ) - Description

- **calMgr** (object) - A CalManager object
- **CalChan** (long) - Channel to be calibrated.
- **SwpChan** (long) - The channel to sweep when waiting to measure a standard.
  
  This channel must already exist with at least one measurement in the channel. If this channel is in continuous sweep mode, it must have the same attenuator settings and path configuration (PNA-X only).

- **state** (Boolean) - Channel sweep state. Choose from:
  - **True** - Sweep the channel during calibration.
  - **False** - Do NOT sweep the channel during calibration.

Return Type
Not Applicable

Default
Not Applicable

Example
```
calMgr.AllowChannelToSweepDuringCalAcquisition 2,1,True
```

See example using this command

C++ Syntax
```
HRESULT AllowChannelToSweepDuringCalAcquisition ( long CalChannel, long SwpChannel, VARIANT_BOOL bVal);
```

Interface
ICalManager5

Last Modified:

- 23-Jan-2009  Added ‘if measurements are deleted’.
- 8-Nov-2007  MX New topic
### AllowEventCategory Method

**Description**  
Sets event filtering to monitor a category of event.

**VB Syntax**  
`app.AllowEventCategory, category, state`

**Variable**  
(Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An <code>Application</code> (object)</td>
<td></td>
</tr>
<tr>
<td><code>category</code></td>
<td>Category to monitor. Choose from list in <a href="#">Working with the Analyzer's Events</a></td>
<td></td>
</tr>
</tbody>
</table>
| `state`  | (boolean)  | True - monitor  
False - do not monitor                                                      |

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`app.AllowEventCategory`

**C++ Syntax**  
`HRESULT AllowEventCategory(tagNAEventCategory category, VARIANT_BOOL bAllow )`

**Interface**  
IAplication
### AllowEventMessage Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Sets event filtering to monitor specific events.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.AllowEventMessage event</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application <em>(object)</em></td>
</tr>
<tr>
<td><code>event</code></td>
<td>Event to monitor. Refer to list in <a href="#">Working with the Analyzer's Events</a></td>
</tr>
<tr>
<td><code>state</code></td>
<td><em>(boolean)</em></td>
</tr>
<tr>
<td><code>True</code></td>
<td>- monitor</td>
</tr>
<tr>
<td><code>False</code></td>
<td>- do not monitor</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.AllowEventMessage</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT AllowEventMessage(tagNAEventID eventID, VARIANT_BOOL bAllow)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IApplication</td>
</tr>
</tbody>
</table>
AllowEventSeverity Method

Description
Sets event filtering to monitor levels of severity.

VB Syntax
`app.AllowEventSeverity severity, state`

Variable (Type) - Description
- `app` An Application (object)
- `severity` (enum `naEventSeverity`) Choose from: `naEventSeverityERROR` `naEventSeverityINFORMATIONAL` `naEventSeveritySUCCESS` `naEventSeverityWARNING`
- `state` (boolean)
  - `True` - monitor
  - `False` - do not monitor

Return Type
Not Applicable

Default
Not Applicable

Examples
`app.AllowEventSeverity`
## Write only

### Apply Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Applies the mixer setup and turns the channel ON. (Performs the same function as the Apply button on the mixer setup dialog box.) Learn about the Scratch and Applied mixer properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>obj.Apply</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>Type</strong> - Description</td>
</tr>
<tr>
<td></td>
<td><code>obj</code> A Mixer Interface pointer to the Measurement (object) Or A Converter Object</td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>mrx.Apply</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT Apply()</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMixer3 IConverter</td>
</tr>
</tbody>
</table>

---

**Last Modified:** 2-Feb-2009  Added converter object
Write-only

ApplyDeltaMatchFromCalSet Method

Description
Specifies a Cal Set as a source of delta match correction.

If ‘GUID’ is not supplied then the Global Delta Match Cal Set is assumed. An error is returned if the specified Cal Set does not meet the following Delta Match criteria. The Global Delta Match Cal can ALWAYS be applied.

- Must have been performed using ECal or as a guided mechanical cal (not Unguided).
- Must have the same start freq, stop freq, and number of points as the channel being calibrated.
- Must calibrate the ports that are required by the TRL or Unknown Thru Cal as indicated by PortsNeedingDeltaMatch Property.

Learn more about Delta match calibration.
See example of a complete Delta Match calibration.

VB Syntax

```
guided.ApplyDeltaMatchFromCalSet [GUID]
```

Variable (Type) - Description

- **guided** GuidedCalibration (object)
- **GUID** Optional Argument. GUID of the Cal Set to use. If unspecified, the Global Delta Match Cal Set is used.

Return Type
Not Applicable

Default
Not Applicable

Examples

```
guided.ApplyDeltaMatchFromCalSet "{2B893E7A-971A-11d5-8D6C-00108334AE96}"
```

C++ Syntax

```
HRESULT ApplyDeltaMatchFromCalSet(BSTR calsetGUID);
```

Interface
IGuidedCalibration2
## ApplyPowerCorrectionValuesEx Method

**Description**
This command replaces `ApplyPowerCorrectionValues Method`. Applies the array of power correction values to the channel memory and turns correction ON. Perform after completing a source power cal acquisition sweep.

This command does NOT save the correction values. To save correction values, save an instrument / calibration state (*.cst file) after performing a source power cal.

Optionally, as part of the source power calibration, perform calibration of the reference receiver used in the power calibration. Learn more.

**VB Syntax**
```
powerCalibrator.ApplyPowerCorrectionValuesEX [rRec]
```

**Variable**
- `powerCalibrator` *(object)* - A `SourcePowerCalibrator` object
- `rRec` *(Enum as NASourcePowerApplyCorrectionOption)* Optional argument. Choose from:
  - 0 - `naSourcePowerApplyCorrectionDefault` Do NOT perform and save a calibration of the reference receiver. (Default if not specified).
  - 1 - `naIncludeReferenceReceiverPowerCal` Perform and save a calibration of the reference receiver. The Cal Set, which includes only the reference receiver cal, is saved to the destination specified by `RemoteCalStoragePreference`.

**Return Type**
None

**Default**
Not Applicable

**Examples**
```
powerCalibrator.ApplyPowerCorrectionValuesEX
powerCalibrator.ApplyPowerCorrectionValuesEX (naIncludeReferenceReceiverPowerCal)
```

**C++ Syntax**
```
HRESULT ApplyPowerCorrectionValuesEx(enum NASourcePowerApplyCorrectionOption option);
```

**Interface**
`ISourcePowerCalibrator5`

---

Last Modified:

23-Apr-2007    MX New topic
ApplySourcePowerCorrectionTo Method

Description
Copies and applies an existing Source Power Calibration to another channel.

VB Syntax
`chan.ApplySourcePowerCorrectionTo (fromPortNum, targetChan, targetPortNum);`

Variable (Type) - Description

- **chan** A Channel (object)
- **fromPortNum** (Long) Port number of the existing source power correction.
- **targetChan** (Long) Channel number to which the source power correction will be copied.
- **targetPortNum** (Long) Port number to which the source power correction will be applied.

Return Type
Not Applicable

Default
Not Applicable

Examples
`chan.ApplySourcePowerCorrectionTo 1,2,1`

C++ Syntax
`HRESULT ApplySourcePowerCalibrationTo (long fromPortNumber, long otherChannelNumber, long portNumber);`

Interface
IChannel11

Last Modified:
20-Jul-2007   MX New topic
AssignSourceToRole Method - Superseded

Description
This command is replaced by: `chan.RoleDevice`
Assigns a configured source to the specified role.

VB Syntax
`conv.AssignSourceToRole role, source`

Variable (Type) - Description
- `conv` A `Converter Object`
- `role` (String) Role to which the external source is assigned. Choose from:
  - For IMDX and IMSX, choose from:
    - "RF2"
    - "LO1"
    - "LO2"
  - For all other converter applications, choose from:
    - "LO1"
    - "LO2"
- `source` (String) Source name from `Source Configuration dialog`.

Return Type
Not Applicable

Default
Not Applicable

Examples
`conv.AssignSourceToRole "LO1", "LO1Name"`

C++ Syntax
`HRESULT AssignSourceToRole(BSTR roleID, BSTR deviceName);`

Interface
IConverter

Last Modified:
- 3-May-2011 Superseded
- 22-Feb-2011 Edited for FCA2
- 2-Feb-2009 New topic
# AutoOrient Method

**Description**
Returns the ECal port that is connected to the specified PNA port. A calibration does not have to be in process.

**VB Syntax**
```vba
ecalPortNumber = ecal.AutoOrient(chanNum, pnaPort, ecalCharNum)
```

**Variable**
<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ecalPortNumber</td>
<td>Long</td>
<td>Variable to store the returned ECal port number that is connected to the specified PNA port number. The returned ECal port number is a 1-based number: 1 = Port A, 2 = Port B, 3 = Port C, 4 = Port D. Zero (0) is returned when the auto-orientation routine is unable to resolve the orientation.</td>
</tr>
<tr>
<td>ecal</td>
<td>ECalModule Object (object)</td>
<td>A ECalModule Object (object)</td>
</tr>
<tr>
<td>chanNum</td>
<td>Long</td>
<td>Channel number that contains the frequency range that will be calibrated.</td>
</tr>
<tr>
<td>pnaPort</td>
<td>Long</td>
<td>PNA port number.</td>
</tr>
<tr>
<td>ecalCharNum</td>
<td>Long</td>
<td>User Characterization number that matches the physical adapters/fixtures that are on the ECal module. This aids in determining the orientation of the ECal module. Choose from: 0 Factory characterization (no adapters - data that was stored in the ECal module by Agilent) 1 User characterization #1 2 User characterization #2 ...and so forth up to: 12 User characterization #12</td>
</tr>
</tbody>
</table>

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```vba
Dim app As AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application", <analyzerName>)
Dim pna
pna.Preset
Const chanNum = 1
pna.Channels(chanNum).StopFrequency = 20E9 ' for a 20 GHz ECal mod
```
Const pnaPortNumber = 1
Const ecalCharacterizationNum = 0
Dim calMgr
Set calMgr = pna.GetCalManager
Dim ecalPortNumber ' The returned ECal port number is a 1-based number
' (1 = Port A, 2 = Port B, etc)
ecalPortNumber = calMgr.ECalModules(1).AutoOrient(chanNum, pnaPortNumber, ecalCharacterizationNum)
MsgBox "ECal port number attached to PNA port 1 = " & ecalPortNumber

**C++ Syntax**

```cpp
HRESULT AutoOrient(long channel, long pnaPortNumber, long characterization, long *pECalPortNumber);
```

**Interface**

IECalModule

---

**Last Modified:**

- 11-Apr-2011  Edited userChar verbage
- 6-Mar-2009  MX New topic
# AutoPortExtMeasure Method

**Description**
Measures either an OPEN or SHORT standard. When this command is sent, the PNA acquires the measurement with which to set automatic port extensions. [Learn more about choosing which standard to measure.](#)

**VB Syntax**
```vb
fixture.AutoPortExtMeasure value
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing (object)</td>
</tr>
<tr>
<td>value</td>
<td>(Enum as NAAutoPortExtMeasure)</td>
</tr>
<tr>
<td></td>
<td>0 - naAPEM_OPEN - Measure OPEN</td>
</tr>
<tr>
<td></td>
<td>1 - naAPEM_SHORT - Measure SHORT</td>
</tr>
</tbody>
</table>

**Return Type**
ENUM

**Default**
Not Applicable

**Examples**
```vb
fixture.AutoPortExtMeasure naAPEM_OPEN
```

**C++ Syntax**
```cpp
HRESULT get_AutoPortExtMeasure(tagNAAutoPortExtMeasure *pVal );
```

**Interface**
IFixturing2
AutoPortExtReset Method

**Description**
Clears old port extension delay and loss data in preparation for acquiring new data. Send this command prior to sending a new series of measurements using `AutoPortExtMeasure Method`. If acquiring both OPEN and SHORT standards, do not send this command between those acquisitions.

**VB Syntax**

```vb
fixture.AutoPortExtReset
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixture</td>
<td>A Fixturing (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vb
fixture.AutoPortExtReset
```

**C++ Syntax**

```cpp
HRESULT AutoPortExtReset();
```

**Interface**

IFxturing2
# Autoscale Method

**Description**

**Trace Object** - Autoscales only the ONE trace on which Autoscale is being called.

**NAWindow Object** - Scales ALL of the traces to fit in the same window. This is equivalent to "Autoscale All" from the front panel.

Autoscale (both trace and window) behaves differently when scale coupling is enabled. How it behaves depends on the scale coupling method. [Learn more.](#)

**VB Syntax**

`object.Autoscale`

**Variable**

*(Type) - Description*

- `object Trace (object)`
- `object NAWindow (object)`

**Return Type**

Not Applicable

**Default**

Not Applicable

**Examples**

- `Trac.Autoscale 'Autoscales the trace`
- `Win.Autoscale 'Autoscales all the traces in the window -Write`

**C++ Syntax**

`HRESULT AutoScale()`

**Interface**

INAWindow

ITrace

---

**Last Modified:**

15-Sep-2010  Added links to scale coupling
### AveragingRestart Method

**Description**
Clears and restarts averaging of the measurement data.

**VB Syntax**
```
chan.AveragingRestart
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chan</td>
<td>A Channel (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
chan.AveragingRestart
```

**C++ Syntax**
```
HRESULT AveragingRestart()
```

**Interface**
IChannel
**BuildHybridKit Method**

**Description**
Use this method when you have different port connectors. This is a convenient way to combine two kits that match the connectors on your DUT.

**VB Syntax**
```
app.BuildHybridKit port1Kit, p1sex, port2Kit, p2sex, adapter, user kit
```

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>Application (object)</td>
<td>An Application (object)</td>
</tr>
<tr>
<td>port1Kit</td>
<td>(enum NACalKit)</td>
<td>Specifies the two kits to be used to build the hybrid kit. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naCalKit_85032F_N50</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85033E_3_5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85032B_N50</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85033D_3_5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85038A_7_16</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85052C_3_5TRL</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User7</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User9</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User10</td>
<td></td>
</tr>
<tr>
<td>port2Kit</td>
<td>(enum NACalKit)</td>
<td>Specifies the two kits to be used to build the hybrid kit. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naCalKit_85032F_N50</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85033E_3_5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85032B_N50</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85033D_3_5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85038A_7_16</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_85052C_3_5TRL</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User7</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User9</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naCalKit_User10</td>
<td></td>
</tr>
<tr>
<td>p1sex</td>
<td>(enum NAPortSex)</td>
<td>Specifies the sex of the connector at that port. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naMale</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naFemale</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naDon'tCare</td>
<td></td>
</tr>
<tr>
<td>p2sex</td>
<td>(enum NAPortSex)</td>
<td>Specifies the sex of the connector at that port. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naMale</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naFemale</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naDon'tCare</td>
<td></td>
</tr>
<tr>
<td>adapter</td>
<td>(enum NAAdapter)</td>
<td>Choose from:</td>
</tr>
<tr>
<td></td>
<td>naUserkit - the electrical length of the adapter in the userKit specifications</td>
<td></td>
</tr>
<tr>
<td></td>
<td>naZeroLength - no adapter</td>
<td></td>
</tr>
<tr>
<td>userKit</td>
<td>(enum NACalKit)</td>
<td>The Hybrid kit - Choose from the previous list of kits</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
app.BuildHybridKit naCalKit_85033E_3_5, naMale, naCalKit_85038A_7_16, naFemale, naUserkit, naCalKit_User8
```

**C++ Syntax**
```
HRESULT BuildHybridKit(tagNACalKit port1Kit, tagNAPortSex port1Sex, tagNACalKit port2Kit, tagNAPortSex port2Sex, tagNAAdapter adapter, tagNACalKit userKit)
```

**Interface**
IApplication
**CalculateErrorCoefficients Method**

**Description**
This method is the final call in a calibration process. It calculates error-correction terms, turns error-correction ON and saves the error-correction terms to the channel’s Cal Register or a User Cal Set.

Do NOT use this command during an ECAL.

*Note:* The destination (Cal Register or User Cal Set) is determined by the setting of the [RemoteCalStoragePreference](#) property.

**VB Syntax**
cal.CalculateErrorCoefficients

**Variable**
(Type) - Description

cal Calibrator (object)

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
Cal.CalculateErrorCoefficients

**C++ Syntax**
HRESULT CalculateErrorCoefficients()

**Interface**
ICalibrator

---

Last Modified:

16-Apr-2007    MX Added link to Remote...
Write only

Calculate Method

Description
Calculates the Input or Output frequencies of the mixer setup, applies the mixer setup to the mixer object, and turns the channel ON.

Note: There is also a Calculate Method on the Converter Object

VB Syntax
obj.Calculate (port)

Variable (Type) - Description

obj  A Mixer Interface pointer to the Measurement (object)

port (enum as MixerCalculation) Port of the mixer for which to calculate start and stop frequencies. Choose from:

<table>
<thead>
<tr>
<th>enum</th>
<th>1st or only stage requires:</th>
<th>In addition, 2nd stage requires:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>mixCalculateINPUT</td>
<td>• Output Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LO frequency</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Output sideband (High or Low)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• IF Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2nd LO frequency</td>
</tr>
<tr>
<td>1</td>
<td>mixCalculateINPUT AndOUTPUT</td>
<td>NA</td>
</tr>
<tr>
<td></td>
<td>(2 stage mixers ONLY)</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>mixCalculateOUTPUT</td>
<td>• Input Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LO frequency</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Output sideband (High or Low)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• IF start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2nd LO frequency</td>
</tr>
<tr>
<td>3</td>
<td>mixCalculateLO1</td>
<td>• Input Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Output frequency</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Output sideband (High or Low)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• IF sideband (High or Low)</td>
</tr>
<tr>
<td>4</td>
<td>mixCalculateLO2</td>
<td>NA</td>
</tr>
</tbody>
</table>
• 1st LO start and stop frequencies
• Output frequency
• IF sideband(High or Low)
• Output sideband(High or Low)

<table>
<thead>
<tr>
<th>Return Type</th>
<th>Not Applicable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>obj.Calculate (mixCalculateOUTPUT)</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT Calculate()</td>
</tr>
<tr>
<td>Interface</td>
<td>IMixer</td>
</tr>
</tbody>
</table>

Last Modified:

2-Feb-2009  Added converter
ChangeParameter Method

Description
Changes the parameter of the measurement.

VB Syntax
`meas.ChangeParameter(param,src)`

Variable (Type) - Description
- `meas` A Measurement (object)
- `param` (string) - New parameter. Case insensitive.

For S-parameters and Applications parameters:
Single-digit port numbers can be separated by "_" (underscore). For example: "S21" or "S2_1"
Double-digit port numbers MUST be separated by underscore. For example: "S10_1"

For Ratioed receiver measurements:
Any two receivers in your PNA separated by "/". For example: "A/R1"
See the block diagram showing the receivers in YOUR PNA.

For Unratioed (absolute power) measurements:
Any receiver in the PNA. For example: "A"
See the block diagram showing the receivers in YOUR PNA

With PNA Rev 6.2, Ratioed and Unratioed measurements can also use logical receiver notation to refer to receivers. This notation makes it easy to refer to receivers with an external test set connected to the PNA. You do not need to know which physical receiver is used for each test port. Learn more.

For ADC measurements
Any ADC receiver in the PNA.
For example: "AI1" indicates the Analog Input1.
Learn more about ADC receiver measurements.

For Balanced S-parameter measurements:
"topology: Sabxy"

- `topology` - Choose from:
  - `sbal` - single-ended to balanced
  - `ssb` - single-ended / single-ended to balanced
  - `bbal` - balanced to balanced
*Sabxy*

Where

- **a** - device output (receive) mode
- **b** - device input (source) mode

(choose from the following for both a and b:)

- **d** - differential
- **c** - common
- **s** - single ended

- **x** - device output (receive) logical port number
- **y** - device input (source) logical port number

For example: "**sbal:sdd42**"

See an example program

For **Imbalance** and **Common Mode Rejection** measurements:

"**topology:parameter**" Choose from:

<table>
<thead>
<tr>
<th>Choose this:</th>
<th>To get this:</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;SBAL:IMBSB&quot;</td>
<td>single-ended to balanced</td>
<td>imbalance</td>
</tr>
<tr>
<td>&quot;SBAL:CMRRSB1&quot;</td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds21/Scs21)</td>
</tr>
<tr>
<td>&quot;SBAL:CMRRSB2&quot;</td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Ssd12/Ssc12)</td>
</tr>
<tr>
<td>&quot;SSB:IMB1SSB&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 1</td>
</tr>
<tr>
<td>&quot;SSB:IMB2SSB&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 2</td>
</tr>
<tr>
<td>&quot;SSB:CMRRSSB1&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds31/Scs31)</td>
</tr>
<tr>
<td>&quot;SSB:CMRRSSB2&quot;</td>
<td>single-ended / single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds32/Scs32)</td>
</tr>
<tr>
<td>&quot;BBAL:IMB1BB&quot;</td>
<td>balanced to balanced</td>
<td>imbalance 1</td>
</tr>
<tr>
<td>&quot;BBAL:IMB2BB&quot;</td>
<td>balanced to balanced</td>
<td>imbalance 2</td>
</tr>
<tr>
<td>&quot;BBAL:CMMRBB&quot;</td>
<td>balanced to balanced</td>
<td>common mode rejection (Sdd21/Scc21)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>src</th>
<th>(long integer)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>● Ignored if param is an S-Parameter</td>
</tr>
<tr>
<td></td>
<td>● Source port if param is a ratioed or unratioed receiver measurement (including ADC measurements).</td>
</tr>
</tbody>
</table>

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use chan.getPortNumber to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```cpp
meas.ChangeParameter "S11",2 '2 is ignored
meas.ChangeParameter "VC21",1 '1 is ignored
meas.ChangeParameter "A/R1",2 '2 is the source port
meas.ChangeParameter "a1/b1",1 '1 is the source port
meas.ChangeParameter "R1",2 '2 is the source port

'to change to a parameter with a string name
dim app
set app = CreateObject("Agilentpna835x.application")
dim capabilities
set capabilities = app.Capabilities
dim portnum
portnum = Capabilities.GetPortNumber("Src2 Out1")
app.activemasurement.ChangeParameter "A",portnum
```

**C++ Syntax** HRESULT ChangeParameter(BSTR parameter, long lPort)

**Interface** IMeasurement

---

Last Modified:

- **24-Apr-2008** Added example for string names
- **30-Apr-2007** Modified for ADC and src strings
CheckPower Method

Description
Measures power at a specified frequency. Use this method to test power level before and/or after applying a source power calibration.

VB Syntax
```vbnet
pow = pwrCal.CheckPower (device, freq [,unit])
```

Variable (Type) - Description
- `pow` (double) Variable to store power value returned by this method.
- `pwrCal` A `SourcePowerCalibrator` (object)
- `device` (enum `NAPowerAcquisitionDevice`) The specific sensor on the power meter to be used for the acquisition. Choose from:
  - 0 – `naPowerSensor_A`
  - 1 – `naPowerSensor_B`
  To use the sensor that currently corresponds to the frequency of interest, use the value from the `PowerAcquisitionDevice` property.
- `freq` (double) Frequency (Hz) at which the sensor is to read the power.
- `unit` (enum `NAPowerUnit`) Optional argument. Choose from:
  - `naDBM` – Returns the power in dBm. (default)
  - `naWATT` – Returns the power in Watts.

Return Type
Double

Default
Not Applicable

Examples
```csharp
watt = powerCalibrator.CheckPower(naPowerSensor_A, 1E9, naWATT)
```

C++ Syntax
```cpp
HRESULT put_CheckPower(tagNAPowerAcquisitionDevice enumAcqDevice, double dFreq, tagNAPowerUnit enumPowerUnit, double *pdPower);
```

Interface
`ISourcePowerCalibrator2`
# Clear Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Clears the current diagnostic information.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>embedLODiag.Clear</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>embedLODiag</code></td>
<td>An <code>EmbeddedLODiagnostic</code> <em>(object)</em></td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>embedLO.Clear 'write</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT Clear();</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IEembeddedLODiagnostic</code></td>
</tr>
</tbody>
</table>

Last Modified:

12-Apr-2007   MX New topic
## Clear Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Clears the FIFO data buffer.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>fifo.Clear</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><code>(Type)</code> - Description</td>
</tr>
<tr>
<td><code>fifo</code></td>
<td>A FIFO (object)</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>fifo.Clear 'write</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT Clear();</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IFIFO</td>
</tr>
</tbody>
</table>

Last Modified: 3-Nov-2008  MX New topic
Write-only

CloseCalSet Method  Superseded

Description
This command is no longer necessary. The CalSet.get... and put... commands that
required this command have been replaced,
Closes read/write access to the Cal Set.
See OpenCalSet for an explanation of gaining access to the Cal Set.
When you are finished reading and writing data from or to the Cal Set, close the Cal
Set. Subsequent read/writes will require a new OpenCal Set call.
Reading and writing Cal Set data is performed with the PutStandard, GetStandard,
PutErrorTerm, GetErrorTerm method calls. These methods are provided by the ICal Set
and ICalData2 interfaces.

VB Syntax

CalSet.CloseCalSet

Variable  (Type) - Description
CalSet  (object) - A Cal Set object

Return Type  Not Applicable

Default  Not Applicable

Examples

CSet.CloseCalSet

C++ Syntax  HRESULT CloseCalSet

Interface  ICalSet
ComputeErrorTerms Method

**Description**  Computes error terms for the caltype specified by a preceding OpenCal Set call.

The Cal Set must first be opened using OpenCalSet. If this call has not been made, the following error is issued:

E_NA_Cal Set_ACCESS_DENIED

The standards data required for the CalType must be available in the Cal Set or this error will be returned: E_NA_STANDARD_NOT_FOUND.

**Note:** Error term computation requires data for the actual calibration kit standards from the current kit definition. ComputeErrorTerms assumes that the standards were acquired using only one standard per class.

**VB Syntax**  
`CalSet.ComputeErrorTerms`

**Variable**  
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CalSet</code> (object)</td>
<td>A <code>Cal Set</code> object</td>
</tr>
</tbody>
</table>

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  
`CalSet.ComputeErrorTerms`

**C++ Syntax**  
`HRESULT ComputeErrorTerms()`

**Interface**  ICalSet
ConfigEnhancedNB2 Method

Description  
**Note:** This command replaces ConfigEnhancedNB Method.

This subroutine determines, then returns, the proper configuration for pulsed measurements on the **PNA-X ONLY** using the spectral nulling technique. The configuration returned needs to be sent to the PNA and any other related external equipment.

The routine will take a desired Pulse Repetition Frequency (PRF) and measurement IFBW and return a possibly modified PRF and IFBW for proper pulsed operation on the PNA.

**VB Syntax**

```
Pulsed.ConfigEnhancedNB2 (PRF, BW, PhysicalIF, NCO, ClockFreq, Stage1TapArray, Stage2TapArray, Stage3TapArray, FixedPRF, GateDelay, GateWidth, SWGateDelay, SWGateWidth, SWGateRamp)
```

**Variable (Type) - Description**

**Pulsed** (interface) An interface to the agilentpnapulsed.dll application interface.

**PRF** (Double) The Pulse Repetition Frequency.

[out] The pulse repetition frequency that has been optimized for use with the PNA. NOTE: This value may be different from the value requested.

[in] The desired pulse repetition frequency.

**BW** (Long) The PNA IF Bandwidth.

[out] The PNA IF bandwidth that has been optimized for use with the PNA. NOTE: This value may be different from the value requested. Zero (0) is returned if no solution is found for the specified **PRF** and **BW**.

[in] The desired PNA IF bandwidth.

**PhysicalIF** (Double)

[out] Returns physical intermediate frequency.

**NCO** (Double)

[out] Returns numeric controlled oscillator frequency.

**ClockFreq** (Double)

[out] Returns the clock frequency (in Hz) of the PNA-X.

**Stage1TapArray** (Long array)

[out] Returns the stage 1 filter coefficients

**Stage2TapArray** (Long array)

[out] Returns the stage 2 filter coefficients
Stage3TapArray (Long array)
[ out ] Returns the stage 3 filter coefficients

FixedPRF (Boolean)
[in]

- 1 (True) Signals the .DLL routine to NOT adjust the PRF value; rather adjust ONLY the IF Bandwidth. This is the default setting.
- 0 (False) Adjust both the PRF and IF Bandwidth values as necessary.

GateDelay (Double)
[in] Highest delay value in seconds used in any of the receiver gates.

GateWidth (Double)
[in] Widest pulse width value in seconds used in any of the receiver gates.

SWGateDelay (Double)
[ out ] Returns the SW gate delay in seconds.

SWGateWidth (Double)
[ out ] Returns the SW Gate width in seconds.

SWGateRamp (Long)
[ out ] Returns the SW Gate ramp

Return Type Not Applicable
Default Not Applicable
Example See an example using this command.

C++ Syntax HRESULT ConfigEnhancedNB2(double *pPRF, long *pBW, double *pIF, double *pNCO, double *clock, double *pStg1, double *pStg2, double *pStg3, VARIANT_BOOL fixPRF, double gateDelay, double gateWidth, double *SWGateDelay, double *SWGateWidth, long *SWGateRamp)

Interface AgilentPNAPulsed.Application
ConfigEnhancedNBIFAtten Method

Description
Sets PNA-X receivers to auto gain setting.

VB Syntax
Pulsed.ConfigEnhancedNBIFAtten (PRF, RxWidth, IFAtten)

Variable (Type) - Description

Pulsed (interface) An interface to the agilentpnapulsed.dll application interface.

PRF (Double)

RxWidth (Double)

IFAtten (Long Integer)
[out] IF attenuation value.

Return Type Not Applicable

Default Not Applicable

Example See an example using this command.

C++ Syntax
HRESULT ConfigEnhancedNBIFAtten(double *pPRF, double *pWidth, long *pIF)

Interface AgilentPNAPulsed.Application
**Description**

Note: This method replaces ConfigNarrowBand2 Method. The BW argument now returns 0 if no solution is found for the specified PRF and BW. In addition, adjustments were made to the filter finder algorithm.

This subroutine determines, then returns, the proper configuration for pulsed measurements on the PNA using the spectral nulling technique. The configuration returned needs to be sent to the PNA and any other related external equipment such as pulse generators. The routine will take a desired Pulse Repetition Frequency (PRF) and measurement IFBW and return a possibly modified PRF and IFBW for proper pulsed operation on the PNA. The routine will also return the Sample Rate, Number of Taps, and Offset that must be sent to the PNA to configure it in pulsed mode using the spectral nulling technique.

Although the example below uses COM programming to communicate with the PNA, these commands can be replaced with SCPI equivalents.

Note: The pulsed application may set the offset frequency (option 080) of the PNA to some value other than zero (the default value). If the stop frequency is set to the maximum of the PNA model, then an error message may appear on the PNA stating that the response frequency has exceeded the maximum allowed frequency. To fix this, set the stop frequency to a value that is at least 2 KHz less than the maximum allowed. For example, if you have a 20 GHz PNA, and the stop frequency is set to 20 GHz, and the error message appears, then set the stop frequency to 19.999998 GHz.

**VB Syntax**

```
Pulsed.ConfigNarrowBand (PRF, NumTaps, BW, OffSet, SampleRate, Precision, FixedPRF, PG81110)
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pulsed</td>
<td>(interface) An interface to the agilentpnanpuled.dll application interface.</td>
</tr>
<tr>
<td>PRF</td>
<td>(Double) The Pulse Repetition Frequency.</td>
</tr>
<tr>
<td>[out]</td>
<td>The pulse repetition frequency that has been optimized for use with the PNA. NOTE: This value may be different from the value requested.</td>
</tr>
<tr>
<td>[in]</td>
<td>The desired pulse repetition frequency.</td>
</tr>
<tr>
<td>NumTaps</td>
<td>(Long) The number of taps to send to the PNA for pulsed operation.</td>
</tr>
<tr>
<td>BW</td>
<td>(Long) The PNA IF Bandwidth.</td>
</tr>
<tr>
<td>[out]</td>
<td>The PNA IF bandwidth that has been optimized for use with the PNA. NOTE: This value may be different from the value requested. Zero (0) is returned if no solution is found for the specified PRF and BW.</td>
</tr>
<tr>
<td>[in]</td>
<td>The desired PNA IF bandwidth.</td>
</tr>
<tr>
<td>OffSet</td>
<td>(Double) The offset value to send to the PNA for pulsed operation. The offset value is used to adjust the PNA for the two different possible sample rates that may be returned.</td>
</tr>
</tbody>
</table>
SampleRate (Double)
[**out**] The sample rate to send to the PNA for pulsed operation.

[**in**] Passing a value of 6.2 us will make sure that the offset frequency is not shifted and therefore could be used with converter measurements. Otherwise enter 0.

Precision (Double)  The precision variables sets the precision that will be used to decrement the PRF when running the configuration routines. This variable can be set to the precision required by the external pulse generators so that the configuration routine will not return a PRF that is not within the precision limits of the pulse generators.

FixedPRF (Boolean)
1 (True) Signals the .DLL routine to NOT adjust the PRF value; rather adjust ONLY the IF Bandwidth. This is the default setting.

0 (False) Adjust both the PRF and IF Bandwidth values as necessary.

PG81110 (Boolean)
1 (True) You are using an Agilent 81110 as the pulse generator. This allows increased accuracy in adjustments for offset and PRF.

0 (False) Not using an Agilent 8110.

Return Type Not Applicable

Default Not Applicable

Example See an example using this command.

C++ Syntax HRESULT ConfigNarrowBand(double *pPRF, long *pNumTaps, long *pBW, double *pOffset, double *pSampleRate, int Precision)

Interface AgilentPNAPulsed.Application

Last Modified:

21-Sep-2007  Modified sample rate and
### ConfigurationFile Method

**Description**
Recalls an Interface Control file from the hard drive into the analyzer.

**VB Syntax**

```vbnet
IntControl.ConfigurationFile (filename)
```

**Variable**

- **Type**
  - Description

  - **IntControl**
    - An `InterfaceControl` (object)

  - **filename**
    - Full path, file name, and extension (.xml) of the file to recall.
    - Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vbnet
IntControl.ConfigurationFile ("C:/Program Files/Agilent/Network Analyzer/Documents/MySettings.xml")
```

**C++ Syntax**

```cpp
HRESULT ConfigurationFile(BSTR bstrFile)
```

**Interface**
`IInterfaceControl`
### Configurations Property

**Description**
Returns an array of stored configuration names that can be used with `DeleteConfiguration Method` and `LoadConfiguration Method`.

**VB Syntax**
```vb
names = pathMgr.Configurations
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>names</code></td>
<td>(Variant array) Variable to store the returned configuration names.</td>
</tr>
</tbody>
</table>

**pathMgr**

`PathConfigurationManager` (object)

**Return Type**
Variant array

**Default**
Not Applicable

**Examples**
```vb
names = path.Configurations
```

**C++ Syntax**
`HRESULT get_Configurations (VARIANT* configurations );`

**Interface**
`IPathConfigurationManager`

---

Last Modified:
14-Dec-2006 MX New topic
Configure Method

**Description**
Restarts as an "N-port" PNA using the specified multiport test set.

*See other commands to configure multiport test sets.*

**VB Syntax**
`app.Configure(model, address)`

**Variable (Type) - Description**

- `app` An Application (object)
- `model` String - Model of the test set with which to restart.
  Use "Native" to restart without a test set.
  To see a list of supported test sets, use
- `address` Integer - GPIB Address of the test set. Use 0 for native restart.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
*See an example using this command.*

```plaintext
app.Configure("N44xx",18)
```

**C++ Syntax**
`HRESULT Configure(BSTR model, long address);`

**Interface**
IAppliation9
Continuous Method

Description  The channel continuously responds to trigger signals.

Note: This command does NOT change TriggerSignal to Continuous.

VB Syntax  chan.Continuous

Variable  (Type) - Description

chan  A Channel (object)

Return Type  Not Applicable

Default  Not Applicable

Examples  chan.Continuous

C++ Syntax  HRESULT Continuous()

Interface  IChannel
**Description**  
Creates a new Cal Set and copies the current Cal Set data into it. Therefore, you now have a clone Cal Set with a different ID. Use this command to manipulate data on a Cal Set without corrupting the original cal data.

**VB Syntax**  
`CalSet.Copy`

**Variable**  
`(Type) - Description`

`CalSet (object) - A Cal Set object`

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**

```vbscript
Dim mgr As CalManager  
Dim ocalset As CalSet  
Dim newcalset As CalSet  
Set mgr = pna.GetCalManager  
'Create a new (empty) Cal Set.  
Set ocalset = mgr.CreateCalSet(1)  
ocalset.Description = "original calset"  
pna.Channel(1).SelectCalSet ocalset.GetGUID, True

'Launch the cal wizard and allow the user to perform the calibration.  
If pna.LaunchCalWizard(False) Then  
'If the Launch returns true then the calibration finished.  
ocalset.Save

'Copy the Cal Set to the new one.  
Set newcalset = ocalset.Copy  
nnewcalset.Description = "copy of original calset"

Else  
'If the cal doesn't finish, delete the old Cal Set  
'so it isn't taking up unnecessary memory.  
mgr.DeleteCalSet ocalset.GetGUID
End If
```

As a result, the programmer can manipulate the data in the new Cal Set and always revert back to the old Cal Set as needed.

**C++ Syntax**  
`HRESULT Copy( ICalSet** pCalSet);`

**Interface**  
ICalSet
### CopyToChannel Method

**Description**  
Copies ALL settings from this channel to the specified channel.  
Use **CopyFrom** to copy ONLY the mechanical switch and attenuator settings.

**VB Syntax**  
*chan*.CopyToChannel(*IChanNum*)

**Variable**  
**(Type) - Description**

- **chan**  
  A **Channel** (object)

- **IChanNum**  
  (long integer) – Number of the channel to become a copy of <chan>.

**Return Type**  
None

**Default**  
Not Applicable

**Examples**

```vbnet
Dim chan
Set chan = PNAapp.ActiveChannel
chan.CopyToChannel 2
```

**C++ Syntax**  
HRESULT CopyToChannel(long IChanNum);

**Interface**  
IChannel2

---

Last modified:

3-May-2011  Added copyFrom link
CreateS-Parameter Method

### Description
This method creates a new S-Parameter measurement in an existing or new window.

### VB Syntax
```vbnet
app.CreateSParameter chan, recvr, source, [window]
```

### Variable (Type) - Description
- **app** [Application](#) (object)
- **chan** (long integer) - Channel number of the new measurement
- **recvr** (long integer) - Port number of the receiver (1 or 2)
- **source** (long integer) - Port number of the source (1 or 2)
- **window** (long integer) - Optional argument. Window number of the new measurement. Choose 1 to 4. If unspecified, the S-Parameter will be created in the Active Window.

### Return Type
Not Applicable

### Default
Not Applicable

### Examples
```vbnet
app.CreateSParameter 1, 2, 1, 1 'Creates a new S21 measurement in channel 1 and New window(1) app.CreateSParameter 1, 2, 1 'Creates a new S21 measurement in channel 1 and in the active window
```

### C++ Syntax
```c++
HRESULT CreateSParameter(long ChannelNum, long RcvPort, long SrcPort, long windowNumber)
```

### Interface
IAplication

---

**Last Modified:**

14-Jul-2011  No longer superseded. Use instead of [Create SParameterEX Method](#)
Write-only

CreateCalSet Method

**Description**

Creates a new Cal Set.

The new cal set is initialized with the stimulus settings from the channel whose number is passed as the argument to this method. Stimulus settings include frequency, bandwidth, number of points, and so forth.

Use this method when you want to manually upload data to the Cal Set using the returned ICal Set interface handle.

The channel number does not restrict the usage of this Cal Set on any other channel. It simply provides a link to the originating channel so that the stimulus values can be stored in the Cal Set.

*Note:* Be sure to SAVE the CalSet you are creating. Use `ICalSet::Save`.

**VB Syntax**

```vbnet
calMgr.CreateCalSet (chan)
```

**Variable (Type) - Description**

- `calMgr` *(object)* - A `CalManager` object
- `chan` *(long)* - channel number of the new Cal Set.

**Return Type**

ICal Set Interface

**Default**

Not Applicable

**Example**

```vbnet
calMgr.CreateCalSet 1
```

**C++ Syntax**

```cpp
HRESULT CreateCalSet( long ChannelNumber, ICal Set** pCal Set);
```

**Interface**

ICalManager
### CreateCustomCal Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Creates a custom cal object.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>calmgr.CreateCustomCal(CalType)</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td>calMgr</td>
<td><strong>Cal Manager</strong> (Object)</td>
</tr>
<tr>
<td>CalType</td>
<td><strong>(String)</strong> Name of the calibration. Choose from:</td>
</tr>
<tr>
<td></td>
<td>&quot;VMC&quot; or &quot;VectorMixerCal.VMCType&quot;</td>
</tr>
<tr>
<td></td>
<td>&quot;SMC&quot; or &quot;ScalarMixerCal.SMCType&quot;</td>
</tr>
<tr>
<td>See Also</td>
<td>SMCTYPE Object</td>
</tr>
<tr>
<td></td>
<td>VMCTYPE Object</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td>Dim CalMgr As ICalManager2</td>
</tr>
<tr>
<td></td>
<td>Dim SMC As ISMCTYPE</td>
</tr>
<tr>
<td></td>
<td>Set SMC = CreateCustomCal(&quot;SMC&quot;)</td>
</tr>
<tr>
<td></td>
<td>See SMC and VMC examples using this command.</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT CreateCustomCal( BSTR CustomCal)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ICalManager2</td>
</tr>
</tbody>
</table>
CreateCustomCalEx Method

**Description**

Returns IGuidedCalibration for the specified channel. With a handle to the IGuidedCalibration interface, you can query it for the following objects for properties appropriate to the calibration setup for the particular meastype (channel).

IGuidedCalibration interface is used to configure a calibration (specify connectors, cal kits, and so forth). It is also used to access any custom calibration properties required for unique application channels like Noise Figure or Gain Compression. To access these special properties, make this call on the IGuidedCalibration interface:

```plaintext
CustomInterface = IGuidedCalibration.CustomCalConfiguration();
```

The interface returned by this call can be used to set and get the custom properties on the following application cal objects:

- NoiseCal Object
- GainCompressionCal Object
- SweptIMDCal Object

**Note:** Use CreateCustomCal_Method to create FCA calibration objects.

**VB Syntax**

```plaintext
calmgr.CreateCustomCalEx (chan)
```

**Variable (Type) - Description**

- **calMgr** (Cal Manager (Object))
- **chan** (long integer) Channel number in which to create the Cal object.

**Return Type**

IGuidedCalibration

**Default**

Not Applicable

**Examples**

```plaintext
Dim guidedcal
Set guidedcal = CalManager.CreateCustomCalEx(1)
```

**See Also**

- Noise Figure example
- Gain Compression example

**C++ Syntax**

```plaintext
HRESULT CreateCustomCalEx(long channel, IDispatch** ppObject);
```

**Interface**

ICalManager5
Last Modified:

22-Jul-2009  Added description text per SW
29-May-2007  MN New topic
**CreateCustomMeasurementEx Method**

**Description**
Creates a new custom measurement or a new 'standard' S-Parameter measurement.

**VB Syntax**
```vbnet
app.CreateCustomMeasurementEx chanNum, MeasClass, MeasName [, window]
```

**Variable (Type) - Description**
- `app` *(object)* - An Application object
- `chanNum` *(long)* - Channel number used by the new measurement; can exist or be a new channel.
- `MeasClass` *(string)* - Measurement class of the new custom measurement object. The new custom measurement must be installed and registered on the PNA.
  
  Choose from the following:
  
  - "Standard"
  - "Vector Mixer/Converter"
  - "Scalar Mixer/Converter"
  - "Gain Compression"
  - "Gain Compression Converters"
  - "Noise Figure Cold Source"
  - "Noise Figure Converters"
  - "Swept IMD"
  - "IM Spectrum"
  - "Swept IMD Converters"
  - "IM Spectrum Converters"

- `MeasName` *(variant)* - Measurement names to create:

<table>
<thead>
<tr>
<th>Meas Class</th>
<th>Measurement Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vector Mixer/Converter</td>
<td>&quot;S11&quot;</td>
<td>Learn about VMC parameters</td>
</tr>
<tr>
<td></td>
<td>&quot;VC21&quot;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>&quot;S22&quot;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>&quot;S11&quot;</td>
<td>Learn about SMC parameters</td>
</tr>
<tr>
<td></td>
<td>&quot;SC21&quot;</td>
<td></td>
</tr>
</tbody>
</table>
Scalar Mixer/Converter

| "SC12" | "S22" | "Ipwr" | "RevIPwr" | "Opwr" | "RevOPwr" |

Gain Compression

| GCA and GCX: |
| "CompIn21" | Input power at the compression point. |
| "CompOut21" | Output power at the compression point. |
| "CompGain21" | Gain at the compression point. |
| "CompS11" | Input Match at the compression point. |
| "RefS21" | Linear Gain. |
| "DeltaGain21" | CompGain21 - Linear Gain. |

Gain Compression Converters

| GCX - All Gain Compression parameters (except S21 and S12) plus the following: |
| "S11" | "SC21" |
| "SC21" | "SC12" |
| "S22" | "Ipwr" |
| "RevIPwr" | "Opwr" |
| "RevOPwr" | Mixer parameters |

Noise Figure AND NFX:

<p>| &quot;NF&quot; | Noise figure |</p>
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;ENR&quot;</td>
<td>Validate noise source measurements.</td>
</tr>
<tr>
<td>&quot;T-Eff&quot;</td>
<td>Effective noise temperature.</td>
</tr>
<tr>
<td>&quot;DUTRN&quot;</td>
<td>DUT noise power ratio. (Noise power expressed in Kelvin divided by 290).</td>
</tr>
<tr>
<td>&quot;DUTRN&quot;</td>
<td>DUT noise power ratio. (Noise power expressed in Kelvin divided by 290).</td>
</tr>
<tr>
<td>&quot;SYSRN&quot;</td>
<td>System noise power ratio.</td>
</tr>
<tr>
<td>&quot;SYSRN&quot;</td>
<td>System noise power ratio.</td>
</tr>
<tr>
<td>&quot;DUTNP&quot;</td>
<td>DUT noise power density. (Noise power expressed in dBm/Hz).</td>
</tr>
<tr>
<td>&quot;DUTNP&quot;</td>
<td>DUT noise power density. (Noise power expressed in dBm/Hz).</td>
</tr>
<tr>
<td>&quot;SYSNP&quot;</td>
<td>System noise power density.</td>
</tr>
<tr>
<td>&quot;SYSNP&quot;</td>
<td>System noise power density.</td>
</tr>
<tr>
<td>&quot;OvrRng&quot;</td>
<td>Indication that the noise receiver is being overpowered.</td>
</tr>
<tr>
<td>&quot;T-Rcvr&quot;</td>
<td>Temperature reading (in Kelvin) of the noise receiver board.</td>
</tr>
</tbody>
</table>

**Noise Figure ONLY - NOT NFX:**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;S11&quot;,</td>
<td>Standard S-parameters; measured with the port1 and port2 noise switches</td>
</tr>
<tr>
<td>&quot;S21&quot;,</td>
<td>set for noise mode.</td>
</tr>
<tr>
<td>&quot;S12&quot;,</td>
<td>Unratioed parameters; with notation:</td>
</tr>
<tr>
<td>&quot;S22&quot;</td>
<td>&quot;receiver, source port&quot;</td>
</tr>
</tbody>
</table>

**NFX ONLY:**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;S11&quot;</td>
<td>Mixer parameters</td>
</tr>
<tr>
<td>&quot;SC21&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;SC12&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;S22&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;Ipwr&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;RevIPwr&quot;</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td>&quot;Opwr&quot;</td>
<td>&quot;RevOPwr&quot;</td>
</tr>
<tr>
<td>&quot;ALO1&quot;, &quot;BLO1&quot;</td>
<td>Test port receiver at LO1 frequency</td>
</tr>
<tr>
<td>...and so forth.</td>
<td></td>
</tr>
<tr>
<td>&quot;R1_1&quot;, &quot;B_2&quot;</td>
<td>Unratioed parameters with notation: &quot;receiver_source port&quot;</td>
</tr>
<tr>
<td>...and so forth.</td>
<td></td>
</tr>
<tr>
<td>Swept IMD</td>
<td>There are over 150 possible Swept IMD parameters, too many to list here.</td>
</tr>
<tr>
<td>Swept IMD Converters</td>
<td>Build the parameters with the Swept IMD Parameter dialog, then copy the parameter name to the remote command.</td>
</tr>
<tr>
<td>Learn more</td>
<td>The following are a few example parameters:</td>
</tr>
<tr>
<td>&quot;PwrMainLo&quot;</td>
<td>Absolute power of the Low tone at the DUT output.</td>
</tr>
<tr>
<td>&quot;IM3&quot;</td>
<td>Power of the third product relative to the average power of the f1 and f2 tones measured at the DUT output.</td>
</tr>
<tr>
<td>&quot;OIP3&quot;</td>
<td>Theoretical power level at which the third product will be the same power level as the average of the main tones at the output of the DUT.</td>
</tr>
<tr>
<td>IM Spectrum</td>
<td></td>
</tr>
<tr>
<td>IM Spectrum Converters</td>
<td></td>
</tr>
<tr>
<td>Learn more</td>
<td></td>
</tr>
<tr>
<td>&quot;Output&quot;</td>
<td>View signals OUT of the DUT and into PNA port 2 (B receiver).</td>
</tr>
<tr>
<td>&quot;Input&quot;</td>
<td>View signals IN to the DUT (R1 receiver).</td>
</tr>
<tr>
<td>&quot;Reflection&quot;</td>
<td>View signals reflected off the DUT input and back into PNA port 1 (A receiver).</td>
</tr>
</tbody>
</table>

**window** (long) Optional argument. Number of the window the new custom measurement will be placed in. Choose between 1 and the [maximum number of windows allowed on the PNA](#). If unspecified, the measurement is placed in the active window.
**Return Type**  
IMeasurement

**Default**  
Not Applicable

**Examples**

'**To create a scalar mixer measurement in channel 2:**
Dim MyMeas as Agilent835x.Measurement
Set MyMeas = app.CreateCustomMeasurementEx (2, "Scalar Mixer/Converter", "SC21")

'**To create a vector mixer measurement in channel 2:**
Dim MyMeas as Agilent835x.Measurement
Set MyMeas = app.CreateCustomMeasurementEx (2, "Vector Mixer/Converter", "VC21")

**C++ Syntax**  
HRESULT put_CreateCustomMeasurementEx (long ChannelNum, BSTR guid, 
VARIANT initData, long windowNumber, IMeasurement** ppMeasurement );

**Interface**  
IApplication3

---

Last Modified:

- **2-Mar-2010**   Fixed IM spectrum converters
- **29-Oct-2009**   Added NFX
- **23-Feb-2009**   Added IMD Converters and Noise Figure I
- **18-Sep-2008**   Added IMD and IM Spectrum
- **8-Nov-2007**    Updated for NF and GCA
## CreateMeasurement Method

### Description
Creates a new measurement.

### VB Syntax
```vbnet
app.CreateMeasurement chanNum, param, lPort[, window]
```

### Variable Information

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>Application (object)</td>
<td></td>
</tr>
<tr>
<td><code>chanNum</code></td>
<td>long</td>
<td>Channel number of the new measurement; can exist or be a new channel</td>
</tr>
<tr>
<td><code>param</code></td>
<td>string</td>
<td>New parameter. Case insensitive.</td>
</tr>
</tbody>
</table>

### For S-parameters:
- Any S-parameter that can be measured by your PNA.
- Single-digit port numbers can be separated by "_" (underscore). For example: "S21" or "S2_1"
- Double-digit port numbers MUST be separated by underscore. For example: "S10_1"

### For Ratioed measurements:
- Any two receivers in your PNA separated by "/". For example: "A/R1"
- See the [block diagram](#) showing the receivers in YOUR PNA.

### For Unratioed (absolute power) measurements:
- Any receiver in the PNA. For example: "A"
- See the [block diagram](#) showing the receivers in YOUR PNA

With PNA Rev 6.2, **Ratioed** and **Unratioed** measurements can also use **logical receiver notation** to refer to receivers. This notation makes it easy to refer to receivers with an external test set connected to the PNA. You do not need to know which physical receiver is used for each test port. Learn more.

### For ADC measurements
- Any ADC receiver in the PNA.
- For example: "AI1" indicates the Analog Input1.
- Learn more about ADC receiver measurements.

### For Balanced S-parameter measurements:
- "topology:Sabxy"

  **topology** - Choose from:

  - **sbal** - single-ended to balanced
- **ssb** - single-ended / single-ended to balanced
- **bbal** - balanced to balanced

**Sabxy** -

Where

- **a** - device output (receive) mode
- **b** - device input (source) mode

(choose from the following for both a and b:)

- **d** - differential
- **c** - common
- **s** - single ended

- **x** - device output (receive) logical port number
- **y** - device input (source) logical port number

For example: "**sbal:sdd42**" 

See an example program

**For Imbalance and Common Mode Rejection measurements:**

"**topology:parameter**" Choose from:

<table>
<thead>
<tr>
<th>Choose this:</th>
<th>To get this:</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>&quot;SBAL:IMBSB&quot;</strong></td>
<td>single-ended to balanced</td>
<td>imbalance</td>
</tr>
<tr>
<td><strong>&quot;SBAL:CMRRSB1&quot;</strong></td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds21/Scs21)</td>
</tr>
<tr>
<td><strong>&quot;SBAL:CMRRSB2&quot;</strong></td>
<td>single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Ssd12/Ssc12)</td>
</tr>
<tr>
<td><strong>&quot;SSB:IMB1SSB&quot;</strong></td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 1</td>
</tr>
<tr>
<td><strong>&quot;SSB:IMB2SSB&quot;</strong></td>
<td>single-ended / single-ended to balanced</td>
<td>imbalance 2</td>
</tr>
<tr>
<td><strong>&quot;SSB:CMRRSSB1&quot;</strong></td>
<td>single-ended / single-ended to balanced</td>
<td>common mode rejection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Sds31/Scs31)</td>
</tr>
<tr>
<td>String</td>
<td>Description</td>
<td>Notes</td>
</tr>
<tr>
<td>-----------------</td>
<td>---------------------------------------------------------</td>
<td>--------------------------------------------</td>
</tr>
<tr>
<td>&quot;SSB:CMRRSSB2&quot;</td>
<td>single-ended / single-ended to balanced common mode rejection (Sds32/Scs32)</td>
<td></td>
</tr>
<tr>
<td>&quot;BBAL:IMB1BB&quot;</td>
<td>balanced to balanced imbalance 1</td>
<td></td>
</tr>
<tr>
<td>&quot;BBAL:IMB2BB&quot;</td>
<td>balanced to balanced imbalance 2</td>
<td></td>
</tr>
<tr>
<td>&quot;BBAL:CMRRBB&quot;</td>
<td>balanced to balanced common mode rejection (Sdd21/Scc21)</td>
<td></td>
</tr>
</tbody>
</table>

**lPort** *(long)*

- **Ignored** if `param` is an S-Parameter, balanced, imbalance, or CMRR parameter.
- **Source port** if `param` is ratioed or unratioed (including ADC) measurements.
- Use 0 for **N5264A**.

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

**window** *(long)* Optional argument. Window number of the new measurement. Choose between 1 and the maximum number of windows allowed on the PNA. If unspecified, the measurement will be created in the Active Window.

See also [Traces, Channels, and Windows on the PNA](#).

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```csharp
app.CreateMeasurement(1, "A/R1", 1, 0)
app.CreateMeasurement(1, "a1/b1", 1, 0)
app.CreateMeasurement(1, "bbal:Sdd21", 1)
app.CreateMeasurement(1, "AI2", 2)
app.CreateMeasurement(1, "R1", 0) ' for N5264A
```

**C++ Syntax**

```csharp
HRESULT CreateMeasurement(long ChannelNum, BSTR strParameter, long lPort, long windowNumber)
```

**Interface** `IApplication`
23-Oct-2008    Added support for N5264A
24-Apr-2008    Clarify ports with string names
23-Jul-2007    Added source port link
25-Apr-2007    Updated for ADC measurements.
12-Sept-2006   MQ Updated for logical receiver notation.
## DataToMemory Method

**Description** Stores the active measurement data into memory creating a memory trace. The memory can then be displayed or used in calculations with the measurement data.

**VB Syntax**

```vbnet
meas.DataToMemory
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>meas</td>
<td>A Measurement (object)</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```vbnet
meas.DataToMemory
```

**C++ Syntax**

```cpp
HRESULT DataToMemory()
```

**Interface** IMeasurement
Deembed Method

Description
De-embeds a fixture from an existing Cal Set based on an S2P file. A new Cal Set is created with the effects of the fixture removed.

When the new Cal Set is applied to a channel, the effects of the fixturing are removed from the measurement data. Do NOT enable fixturing. The effects of the fixture are removed when the new Cal Set is selected and correction is turned ON.

VB Syntax

```vb
calMgr.Deembed (cs1, cs2, s2p, port, compPwr, extrap)
```

Variable (Type) - Description

- **calMgr** (object) - A `CalManager` object
- **cs1** (String) - Name of an existing Cal Set which resides on the PNA.
- **cs2** (String) - Name of new Cal Set which contains updated error terms with fixture de-embedded.
- **s2p** (String) - Name of the S2P file which characterizes the adapter/fixture.
- **port** (Long Integer) - Port number from which fixture will be de-embedded.
- **compPwr** (Boolean)
  - **True** - When the Cal Set contains a power correction array for the fixture port, that array will be compensated for the fixture loss.
  - Warning: enabling power compensation can result in an increase in test port power and consequently, increased power to the DUT. Use with caution.
  - **False** - Do not compensate for loss in source power through the fixture.
- **extrap** (Boolean)
  - **True** - Applies a simple extrapolation when the S2P file has a narrower frequency range than the Cal Set. The values for the first and last data points are extended in either direction to cover the frequency range of the Cal Set.
  - **False** - Extrapolation is NOT performed (default setting).

Return Type
Not Applicable

Default
Not Applicable

Example

```vb
calMgr.Deembed  "MyCalSet", "MyNewCalSet", "Fixture.s2p", 1, True, True
```

C++ Syntax

```cpp
HRESULT Deembed (BSTR srcSet, BSTR destSet, BSTR s2p, long port, BOOL compPwr, BOOL extrap);
```

Interface
ICalManager8
Last Modified:

25-Jan-2011    MX New topic
## Delete Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Deletes the measurement.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>meas.Delete</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><code>(Type) - Description</code></td>
</tr>
<tr>
<td><code>meas</code></td>
<td>The Measurement object to delete <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>meas.Delete</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT Delete()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>
DeleteMarker Method

**Description**
Deletes a marker from the measurement.

**VB Syntax**
`meas.DeleteMarker(Mnum)`

**Variable (Type) - Description**
- **meas** (object)
  - A Measurement
- **Mnum** (long)
  - Any existing marker number in the measurement

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
`meas.DeleteMarker(1)`

**C++ Syntax**
`HRESULT DeleteMarker(long IMarkerNumber)`

**Interface**
IMeasurement
### DeleteAllMarkers Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Deletes all of the markers from the measurement.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>meas.DeleteAllMarkers</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>meas</code></td>
<td><strong>(object)</strong> - The Measurement object from which markers will be deleted.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>meas.DeleteAllMarkers</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT DeleteAllMarkers()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IMeasurement</code></td>
</tr>
</tbody>
</table>
DeleteAllSegments Method

Description
Removes all segments from the scratch mixer.

VB Syntax
```
conv.DeleteAllSegments()
```

Variable
```
conv
```
A Converter Object

Return Type
Not Applicable

Default
Not Applicable

Examples
```
mxr.DeleteAllSegments()
```

See example program

C++ Syntax
```
HRESULT DeleteAllSegments();
```

Interface
IConverter5

Last Modified:
26-Oct-2010 New command (A.09.33)
DeleteCalSet Method

Description
Deletes a Cal Set from the set of available Cal Sets. This method immediately updates the Cal Set file on the hard drive. If the Cal Set is currently being used by a channel or does not exist, this request will be denied and an error is returned.

Using the Cal Sets collection is a convenient way to manage Cal Sets.

VB Syntax
`calMgr.DeleteCalSet (calset)`

Variable
(Type) - Description

`calMgr` (object) - A CalManager object

`calset` (string) - Cal Set to be deleted. Specify the Cal Set by GUID or Name. Use EnumerateCalSets to list the available Cal Sets by name.

Return Type
Not Applicable

Default
Not Applicable

Example
```
Set pna=CreateObject("AgilentPNA835x.Application")
Set cmgr = pna.GetCalManager
cmgr.DeleteCalSet ("MyCalSet")
```

C++ Syntax
`HRESULT DeleteCalSet( BSTR strCalset);`

Interface
ICalManager

Last Modified:
6-Mar-2008   Added Name argument
DeleteConfiguration Method

Description
Deletes the specified configuration name from the PNA. The factory configurations cannot be deleted. This is the only method of programmatically distinguishing a factory configuration from a user-named configuration.

VB Syntax
pathMgr.DeleteConfiguration name

Variable (Type) - Description
pathMgr PathConfigurationManager (object)
name (String) Configuration name to be deleted.

Return Type
Not Applicable

Default
Not Applicable

Examples
path.DeleteConfiguration "myMixer"

C++ Syntax
HRESULT StoreConfiguration (long channelNum, BSTR configName);

Interface
IPathConfigurationManager

Last Modified:
14-Dec-2006   MX New topic
DeleteSegment Method

**Description**
Removes the specified number of segments from the scratch mixer starting at the index position.

**VB Syntax**
```
conv.DeleteSegment index,count
```

**Variable**
(Type) - Description
- `conv` A Converter Object
- `index` (Long integer) Position at which to start removing segments. Valid index range is between 1 and the current segment count. Use SegmentCount Property to read the current count in the Applied Mixer.
- `count` (Long integer) Optional argument. Number of segments to remove. If unspecified, 1 segment is removed.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
mxr.DeleteSegment 1,5 'Removes 5 segments beginning at the first position.
```

**C++ Syntax**
```
HRESULT DeleteSegment(long index, long count);
```

**Interface**
IConverter5

---

Last Modified:

26-Oct-2010 New command (A.09.33)
## DeleteShortCut Method

**Description**  
Removes a macro from the list of macros in the analyzer. Does not remove the file.  

**Note:** There are always 12 macro positions. They do not have to be sequential. For example, you can have number 7 but no numbers 1 to 6.

### VB Syntax
```
app.DeleteShortCut item
```

### Variable *(Type) - Description*

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>item</code></td>
<td>(long integer) number of the macro to be deleted.</td>
</tr>
</tbody>
</table>

### Return Type
Not Applicable

### Default
Not Applicable

### Examples
```
app.DeleteShortCut 2
```

### C++ Syntax
```
HRESULT DeleteShortcut(long Number)
```

### Interface
IApplication
### DisallowAllEvents Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Sets event filtering to monitor NO events.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.DisallowAllEvents</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>app</code></td>
<td>An <code>Application</code> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.DisallowAllEvents</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT DisallowAllEvents()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IApplication</code></td>
</tr>
</tbody>
</table>
**Write only**

**DiscardChanges Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Cancels changes that have been made to the Converter setup and reverts to the previously-saved setup. Same as the <strong>Cancel</strong> button on the <a href="#">mixer setup dialog box</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>conv.DiscardChanges</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - <strong>Description</strong></td>
</tr>
<tr>
<td><code>conv</code></td>
<td>A <a href="#">Converter Object</a></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>conv.DiscardChanges</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT DiscardChanges();</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IConverter</td>
</tr>
</tbody>
</table>

---

Last Modified:

2-Feb-2009  New topic
About Cal Window

DisplayNAWindowDuringCalAcquisition Method

Description
Set the 'show' state of the window to be displayed during a calibration.

When this command is sent, the specified window is 'flagged' to be shown during calibration. The flag is cleared when the window is closed. A Preset or Instrument State Recall also closes the window. If the same window number is reopened, this command must be sent again to show the window during a calibration. The flag is NOT saved with an instrument state.

Send this command for each additional window to show during a calibration.

VB Syntax
```
calMgr.DisplayNAWindowDuringCalAcquisition (winNum, State)
```

Variable (Type) - Description
- **calMgr** (object) - A CalManager object
- **winNum** (long) - Window number to show during a calibration. The calibration window will also be shown with this window.
  The window must already be created.
  Use NaWindows.count or app.WindowNumber to read existing window numbers.
- **state** (Boolean) Window state. Choose from:
  - **True** - Show the specified window during calibration.
  - **False** - Do NOT show the specified window during calibration.

Return Type
Not Applicable

Default
Not Applicable

Example
```
calMgr.DisplayNAWindowDuringCalAcquisition 2, True
```

C++ Syntax
```
HRESULT DisplayNAWindowDuringCalAcquisition( long WinNum, VARIANT_BOOL bVal);
```

Interface
ICalManager5

Last Modified:
- 28-Jan-2009  Removed 'Read'
- 8-Nov-2007  MX New topic
### DisplayOnlyCalWindowDuringCalAcquisition Method

**Description**
Clears the flags for windows to be shown during calibrations other than the Cal Window. To flag a window to be shown see [DisplayNAWindowDuringCalAcquisition](#)

**VB Syntax**
`calMgr.DisplayOnlyCalWindowDuringCalAcquisition`

**Variable**
- **(Type)** - **Description**
  - `calMgr` (object) - A [CalManager](#) object

**Return Type**
Not Applicable

**Default**
Not Applicable

**Example**
`calMgr.DisplayOnlyCalWindowDuringCalAcquisition`

See example using this command

**C++ Syntax**
```cpp
HRESULT DisplayOnlyCalWindowDuringCalAcquisition()
```

**Interface**
ICalManager5

---

**Last Modified:**
8-Nov-2007   MX New topic
## DoPrint Method

**Description**  Prints the screen to the default Printer.

**VB Syntax**  
```vb
app.DoPrint
```

**Variable**  
- **app**  An Application (object)

**Return Type**  Not Applicable

<table>
<thead>
<tr>
<th>Examples</th>
<th>C++ Syntax</th>
<th>Interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>app.DoPrint</td>
<td>HRESULT DoPrint()</td>
<td>IApplication</td>
</tr>
</tbody>
</table>

### Return Type
- **Default**  Not Applicable
DoECAL1PortEx Method

**Description**  This method replaces DoECAL1Port Method. Does a 1-Port calibration using an ECAL module. You must first have a 1-port measurement active to perform the calibration. The characterization within the ECal module that will be used for the calibration is specified by ECALCharacterizationEx. The default value is 0.

**VB Syntax**  `cal.DoECAL1PortEx [port],[module]`

---

**Variable** *(Type) - Description*

- `cal`  A Calibrator *(object)*
- `port` *(long integer)* Optional argument - Port number to calibrate. Choose from:
  1. Calibrate port 1 (default if unspecified)
  2. Calibrate port 2
- `module` *(long integer)* Optional argument. ECal module. Choose from modules 1 through 8
  Use IsECALModuleFoundEx to determine the number of modules connected to the PNA
  Use GetECALModuleInfoEx to returns the model and serial number of each module.

**Return Type**  None

**Default**  Not Applicable

**Examples**  `cal.DoECAL1PortEx,2,2`

**C++ Syntax**  `HRESULT DoECAL1PortEx(long port, long moduleNumber = 1);`

**Interface**  ICalibrator4
DoECAL2PortEx Method

Description
This method replaces DoECAL2Port Method.
Does a 2-port calibration using an ECal module.

2-port refers to the number of ports to calibrate; NOT to the number of ECal module ports.

You must first have a measurement active to perform the calibration.
The characterization within the ECal module that will be used for the calibration is specified by ECalCharacterizationEx. The default value is 0.

VB Syntax
`cal.DoECAL2PortEx [portA],[portB],[module]`

Variable (Type) - Description

**cal**  A Calibrator (object)

**portA** (long integer) Optional argument - Number of the receive port to calibrate. Choose from:
1 - Calibrate port 1 (default, if unspecified)
2 - Calibrate port 2
3 - Calibrate port 3

And so forth for all available PNA / test set ports.

**portB** (long integer) Optional argument - Number of the source port to calibrate. Choose from:
1 - Calibrate port 1
2 - Calibrate port 2 (default, if unspecified)
3 - Calibrate port 3

And so forth for all available PNA / test set ports.

**module** (long integer) Optional argument. ECal module.
Choose from modules 1 through 8
Use IsECALModuleFoundEx to determine the number of modules connected to the PNA
Use GetECALModuleInfoEx to return the model and serial number of each module.

Return Type
None

Default
Not Applicable

Examples
`cal.DoECAL2PortEx,1,2,3`

C++ Syntax
`HRESULT DoECAL2PortEx( long portA = 1, long portB =2, long moduleNumber = 1);`

Interface ICalibrator4
1-Jan-2007   Corrected Port B default
## DoneCalConfidenceCheckECAL Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Concludes the Confidence Check and sets the ECal module back into the idle state.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>cal.DoneCalConfidenceCheckECAL</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>cal</code></td>
<td>A Calibrator <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>None</td>
</tr>
<tr>
<td>Default</td>
<td>None</td>
</tr>
<tr>
<td>Examples</td>
<td><code>cal.DoneCalConfidenceCheckECAL</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT DoneCalConfidenceCheckECAL();</code></td>
</tr>
<tr>
<td>Interface</td>
<td>ICalibrator</td>
</tr>
</tbody>
</table>
### DoReceiverPowerCal Method

#### Description

**Note:** This command replaces DataToDivisor, LogMagnitudeOffset, Normalization, InterpolateNormalization.

Immediately performs a receiver power calibration. The connection to the receiver must be in place when this command is sent.

A Receiver Power Cal requires that the active measurement be an Unratioed power measurement.

#### VB Syntax

```
cal.DoReceiverPowerCal(param, srcPort [,pwrOffset])
```

#### Variable (Type) - Description

**cal**  
A **Calibrator** (object)

**param**  
(string) – Receiver to be calibrated. Choose any receiver in your PNA. See a block diagram of your PNA.

With PNA Rev 6.2, receivers can also be referred to using **logical receiver notation**. This notation makes it easy to refer to receivers with an **external test set** connected to the PNA. You do not need to know which physical receiver is used for each test port. Learn more.

**srcPort**  
(long integer) – Number of the port which will supply source power to the receiver during this cal.

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

**pwrOffset**  
(double) – Optional argument. Offset value in dB. Adjusts a receiver power cal to account for components or adapters that are added between the source port and receiver while performing this cal. Specify loss as a negative number; and gain as a positive number.

#### Return Type

None

#### Default

Not Applicable

#### Examples

```
cal.DoReceiverPowerCal "B", 1, -10
```

#### C++ Syntax

```
HRESULT DoReceiverPowerCal(BSTR parameter, long ISrcPort, double dPowerOffset);
```

#### Interface

ICalibrator5

---

1758
24-Apr-2008    Added note for string names
30-Apr-2007    Edited for src strings
## DoResponseCal Method

### Description
Performa and immediately applies a Response cal. Same as selecting **Normalize** from the Unguided Cal - Measure Standards page. [Learn more](#).

### VB Syntax
```
cal.DoResponseCal (measParam),(SourcePort)
```

### Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>cal</td>
<td>Calibrator (object)</td>
<td>A Calibrator (object)</td>
</tr>
<tr>
<td>measParam</td>
<td>String</td>
<td>Measurement parameter to correct. It is NOT necessary for this measurement to be present.</td>
</tr>
<tr>
<td>SourcePort</td>
<td>long integer</td>
<td>Source port number to calibrate. Optional for S-parameter measurements. Choose from: 0 - N5264A Measurement Receiver (no source ports). 1 - Calibrate port 1 2 - Calibrate port 2 (default, if unspecified) 3 - Calibrate port 3 And so forth for all available PNA / test set ports.</td>
</tr>
</tbody>
</table>

### Return Type
None

### Default
Not Applicable

### Examples
```
cal.DoResponseCal "A/R",1
```

### C++ Syntax
```
HRESULT DoResponseCal(BSTR param, long SourcePort);
```

### Interface
ICalibrator9

---

**Last Modified:**

26-Oct-2009  MX New topic
### Embed Method

**Description**
Embeds a fixture (usually a matching network) into an existing Cal Set based on an S2P file. A new Cal Set is created with the effects of the matching network included in the correction data.

When the new Cal Set is applied to a channel, the effects of the fixture are included in the measurement data. Do NOT enable fixturing. The effects of the matching network are included when the new Cal Set is selected and correction is turned ON.

**VB Syntax**
```vbnet
calMgr.Embed (cs1, cs2, s2p, port, compPwr, extrap)
```

**Variable (Type) - Description**
- **calMgr** *(object)* - A `CalManager` object
- **cs1** *(String)* - Name of an existing Cal Set which resides on the PNA.
- **cs2** *(String)* - Name of new Cal Set which contains updated error terms with fixture embedded.
- **s2p** *(String)* - Name of the S2P file which characterizes the adapter/fixture.
- **port** *(Long Integer)* - Port number from which fixture will be embedded.
- **compPwr** *(Boolean)*
  - **True** - Increase the source power to compensate for the loss through the fixture. The result is that the specified power level will be correct at the DUT input.
  - Warning: enabling power compensation can result in an increase in test port power and consequently, increased power to the DUT. Use with caution.
  - **False** - Do not compensate for loss in source power through the matching network.
- **extrap** *(Boolean)*
  - **True** - Applies a simple extrapolation when the S2P file has a narrower frequency range than the Cal Set. The values for the first and last data points are extended in either direction to cover the frequency range of the Cal Set.
  - **False** - Extrapolation is NOT performed (default setting).

**Return Type**
Not Applicable

**Default**
Not Applicable

**Example**
```vbnet
calMgr.Embed "MyCalSet", "MyNewCalSet", "Fixture.s2p", 1, True, True
```

**C++ Syntax**
```cpp
HRESULT Embed (BSTR srcSet, BSTR destSet, BSTR s2p, long port, BOOL compPwr, BOOL extrap);
```

**Interface**
ICalManager8
Last Modified:

25-Jan-2011  MX New topic
**EnumerateCalSets Method**

**Description**
Returns an array of Cal Set names being stored on the PNA.

**VB Syntax**
```vbnet
value = calMgr.EnumerateCalSets
```

**Variable (Type) - Description**
- `value` *(variant)* - Variable to store the returned Cal Set names
- `calMgr` *(object)* - A `CalManager` object

**Return Type**
VARIANT array

**Default**
Not Applicable

**Example**
```vbnet
Dim pnaSet
pna=CreateObject("AgilentPNA835x.Application")
Dim catalog
catalog=pna.getcalmanager.EnumerateCalSets
For i=lbound(catalog) to Ubound(catalog)
    wscript.echo catalog(i)
Next
```

**C++ Syntax**
```c++
HRESULT EnumerateCalSets(VARIANT* names);
```

**Interface**
ICalManager4
**EnumerateItems Method**

**Description**
Returns a list of all name-value pairs (items) in the Cal Set.

**See Also**
- [Item Property](#) (Learn about Name-Value pairs.)
- [RemoveItem Method](#)

**VB Syntax**
```vbnet
names = CalSet.EnumerateItems
```

**Variable** *(Type)* | Description
---|---
`names` | (Variant array) List of string names.
`CalSet` | *(object)* - A [CalSet](#) object

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
See example

**C++ Syntax**
```cpp
HRESULT EnumerateItems (VARIANT* itemNames);
```

**Interface**
ICalSet6

**Last Modified:**
24-Sep-2010 | MX New topic
**Execute Method**

**Description**
Allows the use of COM to send a SCPI command.
This method can be used with :SYST:ERR? to convert scpi errors into text.

*See an example* of how to return error information when using the [Parse method](#).

**Note:** The SCPIStringParser Methods can NOT be used with SCPI Status Reporting. However, the *OPC? will work.*

**VB Syntax**
```
scpi.Execute(SCPI_Command)
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>scpi</code></td>
<td>A <a href="#">ScpiStringParser</a> (Object)</td>
</tr>
</tbody>
</table>

**SCPI_Command**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>String</code></td>
<td>Any valid SCPI command</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Not Applicable

**Examples**
```
Dim scpi As ScpiStringParser
Set scpi = app.ScpiStringParser
scpi.Execute("SYST:PRES");
ErrorString = scpi.Execute("SYST:ERROR?");
```

**C++ Syntax**
```
Execute(BSTR SCPI_Command, BSTR * pQueryResponse);
```

**Interface**
IScpiStringParser2

---

**Last Modified:**
27-Apr-2009    Added note
ExecuteShortcut Method

**Description**
Executes a Macro (shortcut) stored in the analyzer. Use `app.getShortcut` to list existing macros. Use `app.putShortcut` to associate the macro number with the file.

**VB Syntax**
```vbnet
app.ExecuteShortcut index
```

**Variable**
- **app** (Type) - An Application (object)
- **index** (long integer) - Number of the macro stored in the analyzer.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
app.ExecuteShortcut 1
```

**C++ Syntax**
```cpp
HRESULT ExecuteShortcut(long index)
```

**Interface**
IApplication
GenerateGlobalDeltaMatchSequence Method

Description
Initiates a global delta match calibration.

Learn more about Delta match calibration.
See example of a complete Delta Match calibration.

VB Syntax
`numSteps = guided.GenerateGlobalDeltaMatchSequence conn, cKit`

Variable
(Type) - Description

`numSteps` Long Integer - Variable to store the returned number of connection steps required by the Global Delta Match Cal.

`guided` GuidedCalibration (object)

`conn` String Connector Type for port 1.

`cKit` String Cal Kit for all ports.

Return Type
Not Applicable

Default
Not Applicable

Examples
`guided.GenerateGlobalDeltaMatchSequence "APC 3.5 female", "85052B"

C++ Syntax
`HRESULT GenerateGlobalDeltaMatchSequence(BSTR port_1_conn, BSTR cal_kit, long *num_steps);`

Interface
IGuidedCalibration2
GenerateErrorTerms Method

Description
Generates the error terms for the specified calibration type, stores the error terms in a Cal Set, saves the Cal Set, and returns the Cal Set GUID.

If ALL the data for the cal type has NOT been acquired an error message is returned.

Note: The manner in which the calibration is assigned to a Cal Set (Cal Register or User Cal Set) is determined by the setting of RemoteCalStoragePreference.

VB Syntax
value = obj.GenerateErrorTerms

Variable (Type) - Description
value (String) - Variable to store the returned GUID or error message.

obj Any of the following:
GuidedCalibration (object)
SMCType (object)
VMCType (object)

Return Type String
Default Not Applicable

Examples string = SMC.GenerateErrorTerms

C++ Syntax
HRESULT GenerateErrorTerms(BSTR* calsetGUID);

Interface IGuidedCalibration
SMCType
VMCType

Last Modified:
28-Jan-2009 Fixed C++ Syntax
**GenerateSteps Method**

**Description**
Returns the number of steps required to complete the calibration.

For an ECal User Characterization this generate steps for the ECal User Characterization process. The channel must already be calibrated using the same, or greater number of PNA ports as the ECal module. Also, the PNA ports must begin with Port 1 and use sequential port numbers.

After this command is executed, subsequent commands can be used to query the number of measurement steps, issue the acquisition commands, query the connection description strings, and subsequently complete a User Characterization calibration.

**VB Syntax**

```vbnet
value = obj.GenerateSteps
```

**Variable (Type) - Description**

- `value` (long) - Variable to store the returned number of steps
- `obj` Any of the following:
  - `GuidedCalibration` (object)
  - `SMCType` (object)
  - `VMCType` (object)
  - `ECalUserCharacterizer` (object)

**Return Type**
Long

**Default**
Not Applicable

**Examples**

```vbnet
value = SMC.GenerateSteps
```

**C++ Syntax**

```cpp
HRESULT put_GenerateSteps(long* steps);
```

**Interface**
IGuidedCalibration
SMCType
VMCType
IECalUserCharacterizer

---

**Last Modified:**
4-Nov-2008 Added ECal object
**GetAllSegments Method**

**Description**
Downloads a segment table from the PNA.

**VB Syntax**

```vbnet
segdata = Segs.GetAllSegments
```

**Variable**

- **segs**
  A `Segments (Collection)`

- **segdata**
  (Variant) A 2-dimensional array of Segment data:
  - Dimension 0 is the number of elements in each segment.
  - Dimension 1 is the number of segments that will be used.

All elements in the returned array are Variant. The type inside each Variant will be as is listed below.

The returned array will contain values for all elements regardless of the settings of `IFBandwidthOption`, `SweepTimeOption`, `SourcePowerOption` and `CouplePorts` properties. Ignore the values for the properties that are set to false.

The following is a list of dimension 0 elements for each segment:

- 0 = Segment state (Boolean True or False)
- 1 = Number of Points in this segment (Integer)
- 2 = Start Freq (Double)
- 3 = Stop Freq (Double)
- 4 = IFBW (Double)
- 5 = Dwell Time (Double)
- 6 + N = Power (Double) where N is the number of source ports of the PNA. For example, with a 4-port, 1-source PNA, indices 6 through 9 correspond to the per-segment power levels for Ports 1 to 4. Use `SourcePortCount Property` and `SourcePortNames Property` to see the available source ports for the PNA.

**Return Type**
Variant, containing an array.

**Default**
Not Applicable

**Examples**
See a VB example using this command
C++ Syntax

HRESULT GetAllSegments (VARIANT *pSegments);

Interface

ISegments5

Last Modified:

28-Apr-2009  MX New topic
**GetCalKitTypeString Method**

**Description**
Returns ECal module model number and serial number based on the index number of the attached ECal modules.

**VB Syntax**

\[ ECalID = cal.GetCalKitTypeString (module) \]

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECalID (string)</td>
<td>variable to store the returned ECal module ID information.</td>
</tr>
<tr>
<td>cal (Calibrator (object))</td>
<td>A Calibrator (object)</td>
</tr>
<tr>
<td>module (long integer)</td>
<td>ECal module. Choose from modules 1 through 8 Use IsECALModuleFoundEx to determine the number of modules connected to the PNA</td>
</tr>
</tbody>
</table>

**Return Type**
String

**Default**
Not Applicable

**Examples**

```
info = cal.GetCalKitTypeString(2)
```

Example return string:

"N4691-60003 ECal 01234"

**C++ Syntax**

```
HRESULT GetCalKitTypeString(long moduleNumber, BSTR* info);
```

**Interface**
ICalibrator8

---

Last Modified:

30-Oct-2009   MX New topic
### GetCompatibleCalKits Method

**Description**

**Note:** This command replaces `CompatibleCalKits Property`.

Returns a comma-separated list of valid kits that use the specified connector type. This includes mechanical cal kits, applicable characterizations found within ECal modules currently connected to the PNA, and all user characterizations stored in PNA disk memory.

For ECal modules, the returned list includes the serial numbers.

See the [ECalUserCharacterizer Object](#).

Use items in the list to select the kit to be used with the `CalKitType Property`.

**VB Syntax**

```
value = guidCal.GetCompatibleCalKits (connectorType)
```

**Variable (Type) - Description**

- **value** (Variant) - Variable to store the returned list of cal kits. One-dimensional array of string values.
- **guidCal** A [GuidedCalibration](#) (object)
- **connectorType** (String) Connector type for which compatible cal kits will be returned.

Use [ValidConnectorType](#) to return a list of connector type strings.

Use [ConnectorType](#) to set the connector type for each port to be calibrated.

**Return Type**

Variant – Containing one-dimensional array of strings.

**Default**

Not Applicable

**Examples**

```vbnet
Dim kits As Variant
kits = guidedCal.GetCompatibleCalKits "Type N (50) male"
```

**C++ Syntax**

```
HRESULT GetCompatibleCalKits(BSTR connector, VARIANT* Kits);
```

**Interface**

IGuidedCalibration5

---

**Last Modified:**

- 17-Feb-2011  Edited connector type argument
- 25-Aug-2009  MX New topic
## GetAuxIO Method

**Description**  
This method returns the [AuxIO](#) interface.

**VB Syntax**  
`app.GetAuxIO`

**Variable**  
**Type** - Description  
- `app`  
  An [Application](#) (object)

**Return Type**  
IHWAuxIO

**Default**  
Not Applicable

**Example**  
```vbnet  
Dim app As AgilentPNA835x.Application  
Dim aux As IHWAuxIO  
Set aux = app.GetAuxIO  
```

**C++ Syntax**  
`HRESULT GetAuxIO (IHWAuxIO **pAux);`

**Interface**  
IApplication
Write-only

About Modifying Cal Kits

GetCalStandard Method

**Description**

Returns a handle to a calibration standard for modifying its definitions. To select a standard for performing a calibration (use Calibrator.AquireCalStandard).

**VB Syntax**

calKit.GetCalStandard(index)

**Variable**

*(Type)* - Description

- **calKit**
  A calKit *(object)*

- **index** 
  *(long)* - Number of calibration standard. Choose 1 to 30; (there are 30 cal standards in every kit).

**Return Type**

calStandard

**Default**

Not Applicable

**Examples**

Dim short As CalStandard
Set short = calKit.getCalStandard(1)
short.label = "myShort"

**C++ Syntax**

HRESULT GetCalStandard(long standardNumber, ICalStandard **pCalStd)

**Interface**

ICalKit
GetCalManager Method

Description
This method returns the ICalManager interface.

VB Syntax
app.GetCalManager()

Variable (Type) - Description

app An Application (object)

Return Type
ICalManager*

Default
Not Applicable

Example
dim app as AgilentPNA835x.Application
dim mgr as CalManager
set mgr = app.GetCalManager()

C++ Syntax
HRESULT GetCalManager( ICalManager **mgr);

Interface
IApplication

Last Modified:
30-Jan-2008 Added parenthesis
**Get CalSetByGUID Method**

**Description**
Requests a Cal Set by GUID. Returns an ICal Set interface.

**VB Syntax**
```vbnet
calMgr.GetCalSetByGUID (GUID)
```

**Variable**
- **(Type)** - **Description**
  - `calMgr` (object) - A CalManager object
  - `GUID` (string) - GUID of the Cal Set being requested.

**Return Type**
Interface object

**Default**
Not Applicable

**Example**
```vbnet
calMgr.GetCalSetByGUID (2B893E7A-971A-11d5-8D6C-00108334AE96)
```

**C++ Syntax**
```cpp
HRESULT GetCalSetByGUID( BSTR* strGUID, ICalSet* pCalSet);
```

**Interface**
ICalManager
GetCalSetCatalog Method - Superseded

Description
This method is replaced with EnumerateCalSets.

Returns a string containing a list of comma-separated GUIDs in the following format:
{FD6F863E-9719-11d5-8D6C-00108334AE96},
{1B03B2CE-971A-11d5-8D6C-00108334AE96},
{2B893E7A-971A-11d5-8D6C-00108334AE96}

VB Syntax
value = calMgr.GetCalSetCatalog

Variable

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(string) - Variable to store the returned GUID list</td>
</tr>
<tr>
<td>calMgr</td>
<td>(object) - A CalManager object</td>
</tr>
</tbody>
</table>

Return Type
String

Default
Not Applicable

Example
value = calMgr.GetCalSetCatalog

C++ Syntax
HRESULT GetCalSetCatalog(BSTR);

Interface
ICalManager

Last Modified:
6-Mar-2008    Superseded
GetCalSetUsageInfo Method

**Description**
Returns a string identifying the Cal Set currently in use by the specified channel. This method identifies the Cal Set being used by returning its GUID. This method also identifies the "Error Term set" within the Cal Set.

Error term sets are identified by integers, with set 0 belonging to the original (non-interpolated) terms. As stimulus values for a channel are changed causing interpolation to be required, a new Error Term set is constructed within the Cal Set to hold the interpolated Error Terms. The sets are sequentially numbered 1, 2, 3, and so forth. These Error Term sets are destroyed when they are no longer being used.

If there is no Cal Set in use for the given channel, the <GUID> argument is set to the empty string.

**VB Syntax**
```
calMgr.GetCalSetUsageInfo chan, GUID, setNumber
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>calMgr</td>
<td>(object) - A CalManager object</td>
</tr>
<tr>
<td>chan</td>
<td>(long) - channel of the Cal Set being requested</td>
</tr>
<tr>
<td>GUID</td>
<td>(string) - variable to store the GUID of the Cal Set being requested. If there is no Cal Set in use for the given channel, the &lt;GUID&gt; argument is set to the empty string.</td>
</tr>
<tr>
<td>setNumber</td>
<td>(long) - variable to store the error term ID being requested. If the returned argument is greater than 0, the set is being interpolated.</td>
</tr>
</tbody>
</table>

**Return Type**
String, Long Integer

**Default**
Not Applicable

**Example**
```
calMgr.GetCalSetUsageInfo 1, GUID, EtermID
```

**C++ Syntax**
```
HRESULT GetCalSetUsageInfo (long lChannel, BSTR* CalSetGUID, long* etermSetID);
```

**Interface**
ICalManager

---

Last Modified:
13-May-2011 Removed parens
GetCalTypes Method

**Description**
Returns a list of available calibration types known to the PNA. The Standard CalTypes are the same on all PNA's, but the Custom CalTypes are not necessarily the same. They are dependent on the custom measurement in the PNA. Learn more about applying Cal Types.

See also CalibrationTypeID to apply a Cal Type containing in a Cal Set.

**VB Syntax**

```vbnet
v = mgr.GetCalTypes
```

**Variable**

- **mgr**  
  A CalManager (Object)

- **v**  
  Name/GuidPair that contains the calibration type name and associated GUID for each cal type known to the PNA.

**Return Type**

(variant) Two dimensional array.

**Examples**

```vbnet
v = CalManager.GetCalTypes
```

**C++ Syntax**

HRESULT GetCalTypes( VARIANT * NameGuidPair )

**Interface**

ICalManager2
GetComplex Method

Description
Retrieves complex data from the specified location. See also getNAComplex, getData, and getPairedData Methods.

VB Syntax
measData.getComplex location, numPts, real(), imag()

Variable (Type) - Description

measData
An IArrayTransfer interface which supports the Measurement object

location (enum NADataStore - IArrayTransfer) - Where the data you want is residing. Choose from:
0 - naRawData
1 - naCorrectedData
2 - naMeasResult
3 - naRawMemory
4 - naMemoryResult
5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using DataToDivisor Method.

numPts (long integer) - Number of data points requested
[out] - specifies number of data elements returned
[in] - specifies the data being requested or the capacity of the arrays

real (single) - Array to store the real values

imag (single) - Array to store the imaginary values

Return Type
Single

Default
Not Applicable

Examples
Dim real(201) AS Single
Dim imag(201) AS Single
Dim pts as Integer
Dim measData As IArrayTransfer
Set measData = app.ActiveMeasurement
measData.getComplex naCorrectedData, pts, real(0), imag(0)

C++ Syntax
IArrayTransfer - HRESULT getComplex(tagNADataStore DataStore, long* pNumValues, float* pReal, float* pImag)

Interface IArrayTransfer
GetConverter Method

**Description**  
This method returns a handle to a Converter object.

**VB Syntax**  
`chan.GetConverter()`

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>chan</code></td>
<td>A Channel (object)</td>
</tr>
</tbody>
</table>

**Return Type**  
IConverter

**Default**  
Not Applicable

**Example**

```vba
Dim app as AgilentPNA835x.Application
Set app = CreateObject("AgilentPNA835x.Application")
Dim chan As IChannel
Set chan = app.ActiveChannel
Dim convert
Set convert = chan.GetConverter()
```

**C++ Syntax**  
`HRESULT GetConverter(IConverter **obj);`

**Interface**  
IChannel

Last Modified:

2-Feb-2009  New topic
GetDataByString Method

**Description**
Retrieves variant data from the specified location in your choice of formats.

The PNA returns complex trace data which is ratioed if required by the measurement parameter, such as S11 or A/B. Otherwise it is raw receiver data, such as A or B.

**Equation Editor Notes:**

- When equation editor is active on a trace in a standard S-parameter channel, GetData returns the data from the parameter on the trace that was measured last. For example, for the equation “S22 + S33 + S11”, then S33 is the last measured parameter because it uses source port 3.

- In applications, if equation editor is active and the original parameter for the trace is not requested anywhere in the channel, then zeros are returned. If the original parameter is being measured within the channel, then data for the original parameter is returned.

- In general, if an equation contains no measurement parameters, then data for the original parameter is returned.

**VB Syntax**
```
data = meas.getDataByString location, format
```

**Variable**
- **data** *(variant)* - Array to store the data.
- **meas** *(object)* - A Measurement object
- **location** *(string)* – Name of the buffer to be read. Choose from:
  - "naRawData"
  - "naCorrectedData"
  - "naMeasResult"
  - "naRawMemory"
  - "naMemoryResult"
  - "naDivisor" - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using DataToDivisor Method.

See Data Access Map

- **format** *(enum NADATAFormat)* - Format in which you would like the data. It does not have to be the displayed format. Choose from:
  - 0 - NADATAFormat_LinMag
  - 1 - NADATAFormat_LogMag
  - 2 - NADATAFormat_Phase
3 - naDataFormat_Polar
4 - naDataFormat_Smith
5 - naDataFormat_Delay
6 - naDataFormat_Real
7 - naDataFormat_Imaginary
8 - naDataFormat_SWR
9 - naDataFormat_PhaseUnwrapped
10 - naDataFormat_InverseSmith
11 - naDataFormat_Kelvin
12 - naDataFormat_Fahrenheit
13 - naDataFormat_Centigrade

Learn more about Data Format.

* Specify Smith or Polar formats to obtain complex data pairs, which require a two-dimensional array \( \text{varData} \) (numpts, 2) to accommodate both real and imaginary data.

All scalar formats return a single dimension \( \text{varData} \) (numpts).

<table>
<thead>
<tr>
<th>Return Type</th>
<th>Variant array</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td>\text{meas.getDataByString} ( \text{&quot;naMeasResult&quot;}, \text{naDataFormat_Phase} )</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>\text{HRESULT getDataByString( BSTR location, tagDataFormat dataFormat, VARIANT * pData );}</td>
</tr>
<tr>
<td>Interface</td>
<td>IMeasurement</td>
</tr>
</tbody>
</table>

Last Modified:

11-Jun-2009 Added EE notes
1-Oct-2007 Added temperature formats
19-Jul-2007 Corrected example
GetData Method

**Description**
Retrieves variant data from the specified location in your choice of formats. To get smoothed data from any of the specified locations, the format must be the same as the displayed format.

The PNA returns complex trace data which is ratioed if required by the measurement parameter, such as S11 or A/B. Otherwise it is raw receiver data, such as A or B.

This method returns a variant which is less efficient than methods available on the `IArrayTransfer` interface.

If you plan to put this data back into analyzer, `putDataComplex` (variant data) method requires complex, two-dimensional data. Therefore, request the data in Polar format.

**Equation Editor Notes:**
- When equation editor is active on a trace in a standard S-parameter channel, `GetData` returns the data from the parameter on the trace that was measured last. For example, for the equation “S22 + S33 + S11”, then S33 is the last measured parameter because it uses source port 3.
- In applications, if equation editor is active and the original parameter for the trace is not requested anywhere in the channel, then zeros are returned. If the original parameter is being measured within the channel, then data for the original parameter is returned.
- In general, if an equation contains no measurement parameters, then data for the original parameter is returned.

**VB Syntax**

```vbnet
data = meas.GetData location, format
```

**Variable (Type) - Description**

- **data** Variant array to store the data.
- **meas** A Measurement (object)
- **location** (enum NADataStore) - Where the data you want is residing. See Data Access Map. Choose from:
  - 0 - naRawData
  - 1 - naCorrectedData
  - 2 - naMeasResult
  - 3 - naRawMemory
  - 4 - naMemoryResult
  - 5 - naDivisor  When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using `DataToDivisor` Method.
format  (enum NADDataFormat) - Format in which you would like the data. It does not have to be the displayed format. Choose from:

0 - naDataFormat_LinMag
1 - naDataFormat_LogMag
2 - naDataFormat_Phase
3 - naDataFormat_Polar*
4 - naDataFormat_Smith*
5 - naDataFormat_Delay
6 - naDataFormat_Real
7 - naDataFormat_Imaginary
8 - naDataFormat_SWR
9 - naDataFormat_PhaseUnwrapped
10 - naDataFormat_InverseSmith
11 - naDataFormat_Kelvin
12 - naDataFormat_Fahrenheit
13 - naDataFormat_Centigrade

Learn more about Data Format.

* Specify Smith or Polar formats to obtain complex data pairs, which require a two-dimensional array varData (numpts, 2) to accommodate both real and imaginary data.

All scalar formats return a single dimension varData(numpts).

naDataFormat_Phase and naDataFormat_PhaseUnwrapped returns degrees. However, putDataScalar method accepts data in radians (not degrees) and displays in degrees.

Return Type  Variant array - automatically dimensioned to the size of the data

Default  Not Applicable

Examples

```vba
Dim varData As Variant
varData = meas.GetData(naMeasResult, naDataFormat_Phase)
'Print Data
For i = 0 to chan.NumberOfPoints-1
    Print varData(i)
Next i
```

See a C# example.

C++ Syntax

```c++
HRESULT getData(tagNADDataStore DataStore, tagDataFormat DataFormat, VARIANT *pData)
```

Interface  IMeasurement

Last Modified: 1787
11-Jun-2009  Added EE notes
14-Apr-2009  Added link to C#
1-Oct-2007  Added temperature formats
**GetECALModuleInfoEx Method**

**Description**
This property replaces Get ECALModuleInfo Method. Returns the following information about the connected ECAL module: model number, serial number, connector type, calibration date, min and max frequency.

The characterization within the ECAL module that this information will be read from is specified by ECALCharacterizationEx. The default value is 0.

**VB Syntax**
```vb
moduleInfo = cal.GetECALModuleInfoEx(module)
```

**Variable (Type) - Description**
- `moduleInfo` (string) - variable to store the module information
- `cal` A Calibrator (object)
- `module` (long integer) - ECAL module.
  Choose from modules 1 through 8
  Use IsECALModuleFoundEx to determine the number of modules connected to the PNA

**Return Type** String

**Default** Not Applicable

**Examples**
```vb
ingo = cal.GetECALModuleInfoEx(2)
```

Example return string:
```
ModelNumber: 85092-60007, SerialNumber: 01386, ConnectorType: N5FN5F, PortAConnector: Type N (50) female, PortBConnector: Type N (50) female, MinFreq: 30000, MaxFreq: 9100000000, NumberOfPoints: 250, Calibrated: July 4 2002
```

**C++ Syntax**
```cpp
HRESULT GetECALModuleInfoEx(long moduleNumber, BSTR* info);
```

**Interface** ICalibrator4
### GetEcalUserCharacterizer Method

**Description**  
This method returns a handle to an `ECalUserCharacterizer` object.

**VB Syntax**  
`calMgr.GetEcalUserCharacterizer()`

**Variable**  
**Type** - Description  
- `calMgr`  
  A `CalManager Object (object)`

**Return Type**  
`IEcalUserCharacterizer`

**Default**  
Not Applicable

**Example**  
```vbnet
Dim mgr as ICalManager
Set mgr = app.GetCalManager
Dim ecalCharacterizer
Set ecalCharacterizer = mgr.GetECalUserCharacterizer()
```

**C++ Syntax**  
```cpp
HRESULT GetEcalUserCharacterizer( IECalUserCharacterizer **obj);
```

**Interface**  
ICalManager6

---

**Last Modified:**  
6-Mar-2009  
MX New topic
GetENRData Method

**Description**  
Read the ENR calibration data from PNA memory.

**VB Syntax**  
`vData = enr.GetENRData()`

**Variable**  
(Type) - Description

- `vData`  
Variable to store the returned ENR data. Frequency value in Hz, followed by corresponding ENR value in dB.

- `enr`  
An ENRFile (object)

**Return Type**  
Variant Array

**Default**  
Not Applicable

**Examples**  
See example program

**C++ Syntax**  
`HRESULT GetENRData (VARIANT vdata);`

**Interface**  
IENRFile

Last Modified:

2-Aug-2007   MX New topic
GetErrorCorrection Method

Description
Reads the error correction state for the channel.
Use ErrorCorrection Property to set this value.
When this command returns true, some measurements on the channel MAY not have error correction ON. This is because the Cal Set currently in place may not contain the appropriate calibration data. To read the error correction state for a measurement, use Error Correction Property.

VB Syntax
chan.GetErrorCorrection (boolean)

Variable (Type) - Description
chan A Channel (object)
boolean (boolean) Variable to store the returned value.
False - Error correction has been set OFF
True - Error correction has been set ON

Return Type
Boolean

Default
About Error Correction

Examples
chan.GetErrorCorrection(value)

C++ Syntax
HRESULT GetErrorCorrection (VARIANT_BOOL *bState)

Interface
IChannel8
GetErrorTerm Method - Superseded

**Description**

*Note:* This command is replaced by **Get ErrorTermByString Method**

Retrieves error term data that is used for error correction. The data is complex pairs. Memory for the returned Variant is allocated by the server. The server returns a variant containing a two-dimensional safe Array.

This method returns a variant which is less efficient than `getErrorTermComplex` on the ICalData interface.

[Learn about reading and writing Calibration data.](#)

**VB Syntax**

```
data = cal.getErrorTerm term, rcv, src
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>data</code></td>
<td>Variant array to store the data.</td>
</tr>
<tr>
<td><code>cal</code></td>
<td>A Calibrator <em>(object)</em></td>
</tr>
<tr>
<td><code>term</code></td>
<td><em>(enum As NaErrorTerm)</em>. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naErrorTerm_Directivity_Isolation</td>
</tr>
<tr>
<td></td>
<td>naErrorTerm_Match</td>
</tr>
<tr>
<td></td>
<td>naErrorTerm_Tracking</td>
</tr>
<tr>
<td><code>rcv</code></td>
<td><em>(long integer)</em> - Receiver Port</td>
</tr>
<tr>
<td><code>src</code></td>
<td><em>(long integer)</em> - Source Port</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>To get this</th>
<th>Specify these parameters:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Error Term</strong></td>
<td><strong>term</strong></td>
</tr>
<tr>
<td>Fwd Directivity</td>
<td>naET_Directivity_Isolation</td>
</tr>
<tr>
<td>Rev Directivity</td>
<td>naET_Directivity_Isolation</td>
</tr>
<tr>
<td>Fwd Isolation</td>
<td>naET_Directivity_Isolation</td>
</tr>
<tr>
<td>Rev Isolation</td>
<td>naET_Directivity_Isolation</td>
</tr>
<tr>
<td>Fwd Source Match</td>
<td>naErrorTerm_Match</td>
</tr>
<tr>
<td>Rev Source Match</td>
<td>naErrorTerm_Match</td>
</tr>
<tr>
<td>Fwd Load Match</td>
<td>naErrorTerm_Match</td>
</tr>
<tr>
<td>Rev Load Match</td>
<td>naErrorTerm_Match</td>
</tr>
<tr>
<td>Fwd Reflection Tracking</td>
<td>naErrorTerm_Tracking</td>
</tr>
<tr>
<td>Rev Reflection Tracking</td>
<td>naErrorTerm_Tracking</td>
</tr>
<tr>
<td></td>
<td>Fwd Trans Tracking</td>
</tr>
<tr>
<td>----------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td></td>
<td>Rev Trans Tracking</td>
</tr>
</tbody>
</table>

**Return Type**  
Variant

**Default**  
Not Applicable

**Examples**  
Dim varError As Variant  
varError = cal.getErrorTerm(naErrorTerm_Tracking, 2, 1)

**C++ Syntax**  
HRESULT getErrorTerm(tagNAErrorTerm ETerm, long ReceivePort, long SourcePort, VARIANT* pData)

**Interface**  
ICalibrator
GetErrorTerm Method  Superseded

Description
This command has been replaced with Get ErrorTermByString.
Returns error term data from the Cal Set. The returned data is complex pairs.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

VB Syntax

data = calSet.getErrorTerm(setNumber, term, rcv, src)

Variable (Type) - Description

data (Variant) Two-dimensional safe array to store the returned data. Memory for the
returned Variant is allocated by the PNA and must be released by client.
Note: See also getErrorTermComplex on the ICalData2 interface to avoid using the
variant data type.

calSet A Cal Set (object)

setNumber (Long) There can be more than one set of error terms in a Cal Set.

- SetNumber 0 contains the original set of error terms for a Cal Set.
- SetNumbers > 0 contain Interpolated error terms. Interpolated error terms are
generated when interpolation is required and destroyed when no longer used.
Learn about Interpolation.
- To determine the SetNumber in use by a channel, see GetCalSetUsagInfo

term (enum As NaErrorTerm2). Choose from:
0 - naET_Directivity (rcv = src)
1 - naET_SourceMatch (rcv = src)
2 - naET_ReflectionTracking (rcv = src)
3 - naET_TransmissionTracking (rcv ≠ src)
4 - naET_LoadMatch (rcv ≠ src)
5 - naET_Isolation (rcv ≠ src)

rcv (Long) - Receiver Port

src (Long) - Source Port

Return Type Variant

Default Not Applicable

Examples
Dim varError As Variant
varError = CalSet.getErrorTerm(0, naET_TransmissionTracking, 2, 1)
C++ Syntax

HRESULT getErrorTerm(long setID, tagNAErrorTerm2 ETerm, long ReceivePort, long SourcePort, VARIANT* pData)

Interface

ICalSet
### GetErrorTermByString Method

**Description**
Returns error term data from the Cal Set by specifying the string name of the error term.

- Learn more about [Reading and Writing Cal Data](#).
- See examples of [Reading](#) and [Writing](#) Cal Set Data.
- See [GetCalSetUsageInfo](#) to determine the setNumber.

### VB Syntax

```
pdata = calset.GetErrorTermByString(setNumber, errorTerm)
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>pdata</code></td>
<td>Variant</td>
<td>Two-dimensional safe array to store the returned data. Memory for the returned Variant is allocated by the PNA and must be released by client.</td>
</tr>
<tr>
<td><code>calset</code></td>
<td>A Cal Set (object)</td>
<td></td>
</tr>
<tr>
<td><code>setNumber</code></td>
<td>(Long)</td>
<td>Set number of the required Cal Set data.</td>
</tr>
</tbody>
</table>

- SetNumber 0 contains the original "master" set of error terms for a Cal Set.
- SetNumbers > 0 refers to the PNA channel number that contains the error terms. When retrieving channel error terms, Correction must be ON.

New beginning PNA Rev 7.2: The channel error term data contains interpolation, fixturing, and port extension data if each is ON.

- For Balanced Measurements, interpolation, fixturing, and port extensions can be ON independently.
- For Standard S-parameters, to get port extension data, both fixturing and port extensions must be ON.

| `errorTerm` | (String) | The string name used to identify a particular error term in the Cal Set. An example string for port 3 directivity in a full 2 port cal might be "Directivity(3,3)". To determine the string names of error terms, see [GetErrorTermList2](#). |

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
[See an Example](#)
C++ Syntax  HRESULT GetErrorTermByString (long SetNumber, BSTR bufferName, VARIANT* pdata);

Interface  ICalSet2
GetErrorTermComplex Method  Superseded

Description
This command has been replaced by GetErrorTermComplexByString.
Retrieves error term data from the error correction buffer. The data is in complex pairs.
Learn more about reading and writing Cal Data using COM.
This method exists on a non-default interface. If you cannot access this method, use the GetErrorTerm Method on ICalibrator.

VB Syntax
```
eData.GetErrorTermComplex term, rcv, src, numPts, real(), imag()
```

Variable (Type) - Description
- **eData**
  An ICalData pointer to the Calibrator object
- **term** (enum NAEErrorTerm) - The error term to be retrieved. Choose from:
  - naErrorTerm_Directivity_Isolation
  - naErrorTerm_Match
  - naErrorTerm_Tracking
- **rcv** (long integer) - Receiver Port
- **src** (long integer) - Source Port
- **numPts** (long integer) - on input, max number of data points to return; on output: indicates the actual number of data points returned.
- **real()** (single) - array to accept the real part of the error-term. One-dimensional for the number of data points.
- **imag()** (single) - array to accept the imaginary part of the error-term. One-dimensional for the number of data points.
<table>
<thead>
<tr>
<th>Error Term</th>
<th>Specify these parameters:</th>
<th>rcv</th>
<th>src</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fwd Directivity</td>
<td>naET_Directivity Isolation</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Directivity</td>
<td>naET_Directivity Isolation</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Isolation</td>
<td>naET_Directivity Isolation</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Isolation</td>
<td>naET_Directivity Isolation</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Source Match</td>
<td>naErrorTerm_Match</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Source Match</td>
<td>naErrorTerm_Match</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Load Match</td>
<td>naErrorTerm_Match</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Load Match</td>
<td>naErrorTerm_Match</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Reflection Tracking</td>
<td>naErrorTerm_Tracking</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Reflection Tracking</td>
<td>naErrorTerm_Tracking</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Trans Tracking</td>
<td>naErrorTerm_Tracking</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Trans Tracking</td>
<td>naErrorTerm_Tracking</td>
<td>1</td>
<td>2</td>
</tr>
</tbody>
</table>

**Return Type** Single

**Default** Not Applicable

**Examples**

```plaintext
ReDim rel(numpts)
ReDim img(numpts)
Dim eData As ICalData
Set eData = chan.Calibrator
eData.getErrorTermComplex naErrorTerm_Directivity_Isolation, 1, 1, 201, rel(0), img(0)
```

**C++ Syntax**

```c
HRESULT raw_getErrorTermComplex(tagNAErrorTerm ETerm, long ReceivePort, long SourcePort, long* pNumValues, float* pReal, float* plmag)
```

**Interface** ICalData
GetErrorTermComplex Method  Superseded

Description
This command is replaced with Get ErrorTermComplexByString.
Returns error term data from the Cal Set. The data is in complex pairs.

Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

Note: This method exists on a non-default interface. If you cannot access this method, use the GetErrorTerm Method on ICal Set.

VB Syntax
```
iCalData2.GetErrorTermComplex setNumber, term, rcv, src, numPts, real(), imag()
```

Variable (Type) - Description

*iCalData2* An *ICalData2* pointer to the Cal Set object

*setNumber* (Long) There can be more than one set of error terms in a Cal Set.

- setNumber 0 contains the original set of error terms for a Cal Set.
- setNumbers > 0 contain Interpolated error terms. Interpolated error terms are generated when interpolation is required and destroyed when no longer used. Learn about Interpolation.
- To determine the setNumber in use by a channel, see GetCalSetUsageInfo

*term* (enum NAErrorTerm2) - The error term to be retrieved. Choose from:

- 0 - naET_Directivity
- 1 - naET_SourceMatch
- 2 - naET_ReflectionTracking
- 3 - naET_TransmissionTracking
- 4 - naET_LoadMatch
- 5 - naET_Isolation

*rcv* (Long) - Receiver Port

*src* (Long) - Source Port

*numPts* (Long) An In/Out parameter.

On the way in, you specify the max number of values being requested.
On the way out, the PNA returns number of values actually returned.

*real()* (single) - array to accept the real part of the error-term. One-dimensional for the number of data points.
imag() (single) - array to accept the imaginary part of the error-term. One-dimensional for the number of data points.

**Return Type** Single

**Default** Not Applicable

**Examples**

```vba
dim numpts as long
umpts = ActiveChannel.NumberOfPoints
ReDim r(numpts) ' real part
ReDim i(numpts) ' imaginary part
Dim CalSet as CalSet
set CalSet = pna.GetCalManager.GetCalSetByGUID( txtGUID )
Dim eData As ICalData2
Set eData = CalSet
eData.getErrorTermComplex 0, naET_LoadMatch, 1, 2, numpts,
r(0),i(0)
```

**C++ Syntax**

```c++
HRESULT getErrorTermComplex(long setID, tagNAErrorTerm2 ETerm, long ReceivePort, long SourcePort, long* pNumValues, float* pReal, float* pImag)
```

**Interface** ICalData2
GetErrorTermComplexByString Method

**Description**
Returns error term data from the Cal Set by specifying the string name.
Learn more about **Reading and Writing Cal Data**
See examples of **Reading** and **Writing** Cal Set Data

**Note:** This method exists on a non-default interface. If you cannot access this method, use **GetErrorTermByString**

**VB Syntax**

```vb
ICalData3.GetErrorTermComplexByString setNumber, errorTerm, numPoints, real(0), imag(0)
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICalData3</td>
<td>An ICalData3 pointer to a CalSet (Object)</td>
</tr>
<tr>
<td>setNumber</td>
<td>(Long) There can be more than one set of error terms in a Cal Set.</td>
</tr>
<tr>
<td></td>
<td>• setNumber 0 contains the original set of error terms for a Cal Set.</td>
</tr>
<tr>
<td></td>
<td>• setNumbers &gt; 0 contain Interpolated error terms. Interpolated error terms are generated when interpolation is required and destroyed when no longer used. Learn about Interpolation.</td>
</tr>
<tr>
<td></td>
<td>• To determine the setNumber in use by a channel, see <strong>GetCalSetUsageInfo</strong></td>
</tr>
<tr>
<td>errorTerm</td>
<td>(String) The string name of error term in the Cal Set. An example string for port 3 directivity in a full 2 port cal might be &quot;Directivity(3,3)&quot;. For a list error term string names, use <strong>Get ErrorTermList2</strong></td>
</tr>
<tr>
<td>numPoints</td>
<td>(Long) An In/Out parameter. On the way in, you specify the max number of values being requested. On the way out, the PNA returns number of values actually returned.</td>
</tr>
<tr>
<td>real</td>
<td>(Single) The real component of the complex data.</td>
</tr>
<tr>
<td>imag</td>
<td>(Single) The imaginary component of the complex data.</td>
</tr>
</tbody>
</table>

**Return Type**
Single

**Default**
Not Applicable

**Examples**
See example

**C++ Syntax**

```c++
HRESULT GetErrorTermComplexByString(long etermSetID, BSTR bufferName, long* inumPoints, single* real, single* imag);
```
Interface ICaIData3
**GetErrortermList Method Superseded**

**Description**

*Note:* This command is replaced by `CalSet.getErrorTermList2`.

Returns the list of Error Terms contained in this Cal Set for the CalType specified in the `OpenCal Set` method. Learn more about reading and writing Cal Data using COM.

The list is a comma separated, textual representation of the error terms with the term name followed by the port path in parentheses:

- `Term (n, n)`
- `Term (m, n)`

Before calling this method you must open the Cal Set with `OpenCal Set`. If the Cal set is not open, this method returns `E_NA_Cal Set_ACCESS_DENIED`.

Use `StringToNAErrorTerm2` to convert the list entries to values that can be used with `GetErrorTerm` and `PutErrorTerm`.

*Note:* The port path designation `(m n)` indicates the ports that contribute to the error being compensated. Directivity, source match and reflection tracking are single port characteristics, designated in this list by `(n n)` where `n` equals the port being characterized.

Other terms characterize the interaction between ports. For example, the load match term is describing the match at port `(m)` while looking into port `(n)`. Thus the notation `(m n)` indicates the two ports that contribute to the loadmatch error.

**VB Syntax**

```vb
CalSet.GetErrorTermList (SetID, count, strList)
```

**Variable**

<table>
<thead>
<tr>
<th><strong>(Type)</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CalSet</code></td>
<td>(object) - A Cal Set object</td>
</tr>
<tr>
<td><code>SetID</code></td>
<td>(long) - specifies the error term set to query. Use 0 for the master set.</td>
</tr>
<tr>
<td><code>count</code></td>
<td>(long) - the number of error terms in the returned list</td>
</tr>
<tr>
<td><code>strList</code></td>
<td>(string) - comma separated list of error terms found in Cal Set</td>
</tr>
</tbody>
</table>

**Return Type**

Not Applicable

**Default**

Not Applicable

**Examples**

```vb
Dim count As Integer
Dim list As String
OpenCalSet (naCalType_TwoPortSOLT 1, 2)
GetErrorTermList (0, count, list)
CloseCalSet ()
```

Assuming the cal set contained the full set of error terms for this two-port Cal, the returned list would be:

"Directivity(1 1), SourceMatch(1 1), ReflectionTracking(1 1), TransmissionTracking(2 1), LoadMatch(2 1), Isolation(2 1)"
C++ Syntax  HRESULT GetErrorTermList (long etermSetID, long* count, BSTR* strList);

Interface  ICalSet
## GetErrorTermList2 Method

**Description**
Returns a list of error terms names found in the Cal Set containing the specified prefix.

Learn more about [Reading and Writing Cal Data](#).

See examples of [Reading](#) and [Writing](#) Cal Set Data.

**VB Syntax**

```vbnet
list = CalSet.GetErrorTermList2(setNumber, calTypePrefix)
```

**Variable**

**list** *(Variant)* Variant containing a string array of error term names.

**CalSet** *(object)* - A [CalSet](#) object

**setNumber** *(Long)* There can be more than one set of error terms in a Cal Set.

- setNumber 0 contains the original set of error terms for a Cal Set.
- setNumbers > 0 contain Interpolated error terms. Interpolated error terms are generated when interpolation is required and destroyed when no longer used. [Learn about Interpolation](#).
- To determine the setNumber in use by a channel, see [GetCalSetUsageInfo](#)

**caltypePrefix** *(String)* The string used to identify Cal Set data as belonging to a specific Cal Type. This string is used as a filter so that only the error term names of interest are returned. If the prefix is empty, all terms are returned.

An example prefix for a two port cal on ports 2 and 3 might be: "Full 2 Port Cal (2,3)".

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
See an [Example](#)

**C++ Syntax**

```cpp
HRESULT GetErrorTermList2 (long SetNumber, BSTR caltypePrefix, VARIANT* list)
```

**Interface**
ICalSet2
### GetExtendedCalInterface Method

**Description**
Returns an interface that exposes the properties of Noise Calibration.

**VB Syntax**
```vbnet
Cal2.GetExtendedCalInterface (interface)
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>Cal2</code></td>
<td>An ICalibrate2 (object)</td>
</tr>
<tr>
<td><code>interface</code></td>
<td>(object) Returns a handle to the specified interface. Choose from ' &quot;NoiseCal&quot;</td>
</tr>
</tbody>
</table>

**Return Type**

**Default**

**Example**
```vbnet
dim noiseCal
dim noiseCalExtensions
set noiseCal= Get Calmanager?.CreateCustomCalEx("NoiseCal")
set noiseCalExtensions = noiseCal.GetExtendedCalInterface("INoiseCal")
```

**C++ Syntax**
```c
HRESULT GetExtendedCalInterface();
```

**Interface**
ICalibrate2

---

Last Modified:
29-May-2007    MN New topic
# Get ExternalTestSetIO Method

**Description**  This method returns the `IExternalTestSetIO` interface.

**VB Syntax**  `app.GetExternalTestSetIO`

**Variable**  
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
</tbody>
</table>

**Return Type**  `IHWExternalTestSetIO`

**Default**  Not Applicable

**Example**  
```vbnet
dim app as AgilentPNA835x.Application
dim ets as HWExternalTestSetIO
set ets = app.GetExternalTestSetIO
```

**C++ Syntax**  
```cpp
HRESULT GetExternalTestSetIO (IHWExternalTestSetIO **ptestset);
```

**Interface**  `IApplication`
GetFilterStatistics Method

Description
Returns all four Filter Statistics resulting from a SearchFilterBandwidth.
To retrieve individual filter statistics, use meas.FilterCF, meas.FilterBW,
meas.FilterLoss, meas.FilterQ properties.

VB Syntax
meas.GetFilterStatistics cf,bw,loss,q

Variable
 meas (Type) - Description
   meas A Measurement (object)
   cf,bw,loss,q Dimensioned variables to store the returned values

Return Type
(double) cf
(singe) bw,loss,q

Default
Not Applicable

Examples
'Dimension variables
Dim cf as Double
Dim bw as Single
Dim loss as Single
Dim q as Single
meas.GetFilterStatistics cf,bw,loss,q

C++ Syntax
HRESULT GetFilterStatistics(double* centerFreq, float* bw, float* loss, float* quality)

Interface
IMeasurement
Read-only

GetGuid Method

| Description | Returns a string containing the GUID identifying this Cal Set. Each Cal Set is assigned a GUID (global unique ID). GUIDs are used to retrieve and select Cal Sets on the PNA. Learn more about reading and writing Cal Data using COM. |

**VB Syntax**

\[ value = CalSet.GetGuid \]

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(string) - Variable to store the returned GUID</td>
</tr>
<tr>
<td>CalSet</td>
<td>(object) - A Cal Set object</td>
</tr>
</tbody>
</table>

**Return**

| Type | String |

**Default**

| Not Applicable |

**Examples**

\[ guid = CalSet.GetGuid 'Read \]

**C++ Syntax**

`HRESULT GetGUID( BSTR* pGUIDString);`

**Interface**

ICalSet
## InputVoltageEX Property

<table>
<thead>
<tr>
<th>Description</th>
<th>This command replaces the <code>get InputVoltage Method</code>. Reads the ADC voltage from the specified location.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td>( \text{volts} = \text{AuxIO}\text{.InputVoltageEX loc} )</td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>( \text{volts} ) (double) - variable to store the return value</td>
</tr>
<tr>
<td><strong>AuxIO</strong></td>
<td>(object) - A Hardware Auxiliary Input / Output object</td>
</tr>
<tr>
<td><strong>loc</strong></td>
<td>(Long) Location from which to read data.</td>
</tr>
</tbody>
</table>

- **For PNA-X models**; reads ADC voltages from the Power I/O connector. Choose from:
  1. Reads voltage on Analog In 1 port (pin 7).
  2. Reads voltage on Analog In 2 port (pin 8).
  3. Reads voltage on GndSens (pin 6).
  4. Reads voltage on Analog Out 1 port (pin 3).
  5. Reads voltage on Analog Out 2 port (pin 4).

- **For all other PNA models**:
  1. Reads voltage on the Analog IN (pin 14) of the AUX IO connector.
  4. Reads voltage on Analog Out 1 port (pin 3).
  5. Reads voltage on Analog Out 2 port (pin 2).

| **Return Type** | Double |
| **Default** | Not Applicable |

### Examples

```vbnet
Dim aux as HWAuxIO
Set aux = PNA.getAuxIO
volts = aux.InputVoltageEX 1
'for PNA-X, read voltage on PowerI/O pin 7
'for all other models, reads voltage on Aux I/O Analog In (pin 14)
```

### C++ Syntax

```c++
HRESULT get_InputVoltageEX (long muxLoc, double* vtVoltage);
```

### Interface

HWAuxIO2

Last Modified: 1812
5-Aug-2008   Removed 'get'
10-Jul-2007   MX New topic
### get_Input1 Method

**Description**
Reads a hardware latch that captures high to low transitions on Input1 of the Material Handler IO. Reading the latch causes it to reset and is ready for the next transition. The hardware latch is only capable of capturing one transition per query. Additional transitions are ignored until after the next query.

Momentarily driving Input1 high, then low, causes a transition to be detected and latched.

**VB Syntax**

```vbnet
inp1 = handlerIo.get_Input1
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>inp1</code> (variant)</td>
<td>A variable to store the return value</td>
</tr>
<tr>
<td><code>handlerIo</code> (object)</td>
<td>A HandlerIO object</td>
</tr>
</tbody>
</table>

**Return Type**

Variant -

- `0` - a high to low transition occurred at Input1 since the last time it was queried.
- `1` - no high to low transition occurred.

**Default**
Not Applicable

**Examples**

```vbnet
input1 = handlerIo.get_Input1  'Read
```

**C++ Syntax**

```cpp
HRESULT get_Input1 (VARIANT* Data);
```

**Interface**

IHWMaterialHandlerIO
**GetIPConfigurationStruct Method**

**Description**
Returns an NA_IPConfiguration data structure which contains information about the current status of the PNA’s computer networking configuration. This is the same set of information that is returned in a single string by the **LANConfiguration** property.

**VB Syntax**

```
value = app.GetIPConfigurationStruct
```

**Variable**

- **(Type) - Description**
  - **value (NA_IPConfiguration)** Variable to receive the PNA IP (LAN) configuration information.
  - **app** An Application (object)

**Return Type**
NA_IPConfiguration

**Default**
Not Applicable

**Examples**

```vbnet
Dim networkConfigInfo As NA_IPConfiguration
networkConfigInfo = app.GetIPConfigurationStruct()

MsgBox “Host name = “ & networkConfigInfo.HostName
MsgBox “Domain name = “ & networkConfigInfo.DomainName
MsgBox “IP address = “ & networkConfigInfo.IPAddress

If Not networkConfigInfo.DHCPEnabled Then
    MsgBox “IP address is static”
Else
    MsgBox “IP address is dynamic”
End If

MsgBox “Subnet mask = “ & networkConfigInfo.SubNet
MsgBox “Gateway = “ & networkConfigInfo.DefaultGateway
MsgBox “Primary DNS server = “ & networkConfigInfo.DNSServer1
MsgBox “Secondary DNS server = “ & networkConfigInfo.DNSServer2
MsgBox “First suffix in DNS suffix search order = “ & networkConfigInfo.DNSSuffix1
MsgBox “Second suffix in DNS suffix search order = “ & networkConfigInfo.DNSSuffix2
MsgBox “Primary WINS server = “ & networkConfigInfo.PrimaryWINSServer
```
<table>
<thead>
<tr>
<th>C++ Syntax</th>
<th>HRESULT GetIPConfigurationStruct (tagNA_IPConfiguration * pIPConfig);</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interface</td>
<td>IApplication14</td>
</tr>
</tbody>
</table>

Last Modified:

2-Jun-2008    MX New topic
GetIsolationPaths Method

Description
Gets the list of paths (port pairings) for which isolation standards will be measured during calibration.

VB Syntax
value = obj.GetIsolationPaths

Variable
(Type) - Description
value (Variant) - Variable to store the returned port paths in pairs. One-dimensional array of Long Integers.

obj Any of the following:
GuidedCalibration (object)

Return Type
Variant – Containing one-dimensional array of Long Integers.

Default
No port pairs (empty Variant variable)

Examples
pathList = guidedCal.GetIsolationPaths
'displaying the paths separated by commas, with a dash (-) between the pair of port numbers comprising each path

For i = LBound(portList) To UBound(portList) Step 2
    msg = msg + CStr(portList(i)) + "-" + CStr(portList(i+1))
If i+1 < UBound(portList) Then msg = msg + ","
Next
MsgBox msg, 0, "List of isolation paths"

C++ Syntax
HRESULT GetIsolationPaths(VARIANT* pathList);

Interface
IGuidedCalibration

Last Modified:
16-Apr-2007    MX New topic
GetLibraryFunctions Method

Description
Returns the functions in an imported (loaded) DLL.

VB Syntax
functions = equation.GetLibraryFunctions location

Variable (Type) - Description
functions (variant) - Array to store the returned functions.

equation A MeasurementEquation object

location (string) – Full path and filename of the *.dll to be read.

Return Type
Variant array

Default
Not Applicable

Examples
functions=equation.GetLibraryFunctions "C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll"

C++ Syntax
HRESULT GetLibraryFunctions( BSTR filename, BSTR* functionList);

Interface
IMeasurementEquation2
Get MaterialHandlerIO Method

**Description**
This method returns the MaterialHandlerIO interface.

**VB Syntax**
```vbnet
app.GetMaterialHandlerIO
```

**Variable**
- **app** *(Type)*
  - **Description**
  - An Application *(object)*

**Return Type**
IHWMaterialHandlerIO

**Default**
Not Applicable

**Example**
```vbnet
Dim app As AgilentPNA835x.Application
Dim hand As IHWMaterialHandlerIO
Set hand = app.GetMaterialHandlerIO
```

**C++ Syntax**
```cpp
HRESULT GetMaterialHandlerIO (IHWMaterialHandlerIO **phand);
```

**Interface**
IApplication
GetNAComplex Method

**Description**  
Retrieves complex data from the specified location.  
See also `getComplex` and `getData` Method.

**VB Syntax**  
`measData.getNAComplex location, numPts, data`

**Variable**  
**Type** - Description  

- **measData**  
  An IArrayTransfer interface which supports the Measurement object

- **location**  
  (enum NADataStore) - Where the data you want is residing. Choose from:
  0 - naRawData
  1 - naCorrectedData
  2 - naMeasResult
  3 - naRawMemory
  4 - naMemoryResult
  5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using `DataToDivisor Method`.

- **numPts**  
  (long integer) - Number of data points requested
  [out] - specifies number of data elements returned
  [in] - specifies the data being requested or the capacity of the `dComplex` array

- **data**  
  (NAComplex) - A one-dimensional array of NaComplex to store the data.

**Return Type**  
NAComplex

**Default**  
Not Applicable

**Examples**  
```vbnet
Dim dComplex(201) As NaComplex
Dim measData As IArrayTransfer
Dim pts as Long
Set measData = app.ActiveMeasurement
measData.getNAComplex naCorrectedData, pts, dComplex(0)
```

**Notes**  
The data is stored as Real and Imaginary (Re and Im) members of the NaComplex user defined type. You can access each number individually by iterating through the array.

```vbnet
For i = 0 to NumPts-1
    dReal (i) = dcomplex (i).Re
    dImag (i) = dcomplex (i).Im
Next i
```

**C++ Syntax**  
```c
HRESULT getNAComplex(tagNADataStore DataStore, long* pNumValues, TsComplex* pComplex)
```
**Interface**  IArrayTransfer
### GetNumberOfGroups Method

**Description**
Returns the number of groups a channel has yet to acquire. To set the number of groups for a channel, use **Number Of Groups Method**.

**VB Syntax**

```
value = chan.GetNumberOfGroups
```

**Variable**

- **value** *(Type)*: Long Integer
  - Description: Number of groups

- **chan** *(Type)*: Channel
  - Description: (object)

**Return Type**

- **(Type)**: Long Integer

**Default**
Not Applicable

**Examples**

```vbnet
groups = chan.GetNumberOfGroups 'Read
```

**C++ Syntax**

```c++
HRESULT GetNumberOfGroups(long* numberOfGroups);
```

**Interface**

IChannel3
get_Output Method

Description
Type 1 and Type2 configurations: Returns the last value written to the selected output pin.
Type3 configuration: Returns the current state of the selected output pin. If an Input1 trigger occurs, the state may not be the same value as was written.

All configurations: Data is written using put_Output Method.

VB Syntax

```vbnet
data = handlerIo.get_Output (pin)
```

Variable

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>data</td>
<td>(variant) - A variable to store the return value. The returned value will be one of the following:</td>
</tr>
<tr>
<td></td>
<td>0 - TTL Low</td>
</tr>
<tr>
<td></td>
<td>1 - TTL High</td>
</tr>
<tr>
<td>handlerIo</td>
<td>(object) - A HandlerIO object</td>
</tr>
<tr>
<td>pin</td>
<td>(enum as NAMatHandlerOutput) - output to read. Choose from:</td>
</tr>
<tr>
<td></td>
<td>naOutput1 (0)</td>
</tr>
<tr>
<td></td>
<td>naOutput1User (1)</td>
</tr>
<tr>
<td></td>
<td>naOutput2 (2)</td>
</tr>
<tr>
<td></td>
<td>naOutput2User (3)</td>
</tr>
</tbody>
</table>

Learn about User Output

Return Type

Variant

Default

Not Applicable

Examples

```vbnet
data = handlerIo.get_Output (naOutput1)
```

C++ Syntax

```
HRESULT get_Output ( tagNAMatHandlerOutput Output, VARIANT* Data );
```

Interface

IHWMaterialHandlerIO
### get_OutputVoltage Method

**Description**  
E836x and PNA-L: Reads voltages on the DAC/Analog Output 1|2 of the Auxiliary IO connector.  
PNA-X: Reads voltage on the Power I/O connector AnalogOut1|2.

**VB Syntax**  
```
volts = AuxIO.get_OutputVoltage (output)
```

**Variable (Type) - Description**

- **volts** *(double)* - variable to store the return value
- **AuxIO** *(object)* - A Hardware Auxiliary Input / Output object
- **output** *(variant)* Number of the output DAC from which to read voltage. Choose from:
  1. Output 1 (Aux I/O pin 3) and (Power I/O pin 3)
  2. Output 2 (Aux I/O pin 2) and (Power I/O pin 4)

**Return Type**  
Double

**Default**  
Not Applicable

**Examples**
```
Dim aux as HWAuxIO  
Set aux = PNA.getAuxIO  
volts = aux.get_OutputVoltage(1)  
' read voltage from Analog Out 1 (Aux I/O pin3) or (Power I/O pin3)
```

**C++ Syntax**  
```
HRESULT get_OutputVoltage( VARIANT Output, double* Voltage );
```

**Interface**  
IHWAuxIO

---

**Last Modified:**  
10-Jul-2007  
Added PNA-X capability
Read-only
get OutputVoltageMode Method

Description
This command returns the mode of the selected "Analog Out" line on the Auxiliary IO connector and Power I/O connector. The modes give the user the option to have the requested voltage applied immediately or not until the sweep is done. To set the mode, use put_OutputVoltageMode Method.

VB Syntax
\[
\text{mode} = \text{auxlo.get\_OutputVoltageMode(output)}
\]

Variable (Type) - Description

- \text{mode} (enum NAOutputVoltageMode) - variable to store the returned mode.
  - naWaitEOS - While in this mode any voltage changes sent to the selected analog out will only get applied to the output between sweeps.
  - naNoWait - While in this mode any voltage changes sent to the selected analog out will occur right away without waiting until the end of a sweep, the voltage gets applied immediately.

- auxlo (object) - A Hardware Auxiliary Input / Output object

- output (double) Analog Output. Choose from 1 or 2

Return Type
enum as NAOutputVoltageMode

Default
naWaitEOS

Examples
\[
vOutMode = \text{auxIo.get\_OutputVoltageMode (1)}
\]

C++ Syntax
\[
\text{HRESULT get\_OutputVoltageMode(VARIANT Output, tagNAOutputVoltageMode* pMode)};
\]

Interface
IHWAuxIO

Last Modified:
10-Jul-2007 Added PNA-X capability
GetPairedData Method

**Description**
Retrieves pairs of data from the specified location.

**Note:** This method exists on a non-default interface. If you cannot access this method, use the **Get Data** Method on IMeasurement.

**VB Syntax**
`measData.getPairedData location, format, numPts, d1, d2`

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>measData</code></td>
<td>An IArrayTransfer interface which supports the Measurement object</td>
<td></td>
</tr>
<tr>
<td><code>location</code></td>
<td>(enum NADataStore)</td>
<td>Where the data you want is residing. Choose from: 0 - naRawData 1 - naCorrectedData 2 - naMeasResult 3 - naRawMemory 4 - naMemoryResult 5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using <strong>DataToDivisor Method</strong>. See <a href="#">Data Access Map</a></td>
</tr>
<tr>
<td><code>format</code></td>
<td>(enum NAPairedDataFormat)</td>
<td>Format in which you would like the Paired data. Choose from: 0 - naLogMagPhase - Log magnitude and phase 1 - naLinMagPhase - Linear magnitude and phase 2 - naRealImaginary - Real and Imaginary <strong>Note:</strong> Selecting naRealImaginary format is the same as using the <strong>getComplex</strong> method</td>
</tr>
<tr>
<td><code>numPts</code></td>
<td>(long integer)</td>
<td>Number of data points requested [out] - specifies number of data elements returned [in] - specifies the data being requested or the capacity of the dPaired array</td>
</tr>
<tr>
<td><code>d1</code></td>
<td>(single)</td>
<td>Array to store the magnitude / real values</td>
</tr>
<tr>
<td><code>d2</code></td>
<td>(single)</td>
<td>Array to store the phase / imaginary values</td>
</tr>
</tbody>
</table>

**Return Type**
Two Single arrays

**Default**
Not Applicable

**Examples**
```vbnet
Dim logm() As Single
Dim phase() As Single
Public measData As IArrayTransfer
```
Set measData = app.ActiveMeasurement
Dim numpts As Long
numPoints = app.ActiveChannel.NumberOfPoints
ReDim logm(numPoints)
ReDim phase(numPoints)

measData.getPairedData naCorrectedData, naLogMagPhase,
numPoints, logm(0), phase(0)

Print values(0), values(1)

C++ Syntax
HRESULT getPairedData(tagNADataStore DataStore, tagNAPairedDataFormat PairFormat, long* pNumValues, float* pReal, float* pImag)

Interface IArrayTransfer
get_Port Method

Description
Returns the value from the specified "readable" port.

VB Syntax
\[ \text{data} = \text{handlerIo}.\text{get}_\text{Port}(\text{port}) \]

Variable (Type) - Description

\( \text{data} \) (variant) - A variable to store the return value. The following table shows what the returned data represents:

<table>
<thead>
<tr>
<th>Port</th>
<th>MSB............</th>
<th>LSB............</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>C3...C0</td>
<td></td>
</tr>
<tr>
<td>D</td>
<td>D3...D0</td>
<td></td>
</tr>
<tr>
<td>E</td>
<td>D3...D0 + C3...C0</td>
<td></td>
</tr>
</tbody>
</table>

\( \text{handlerIo} \) (object) - A HandlerIO object

\( \text{port} \) (enum as NAMatHandlerPort) - port to get data from. Choose from:
- \( \text{naPortC} \) - (2)
- \( \text{naPortD} \) - (3)
- \( \text{naPortE} \) - (4)

Note: Reading data from the Write-only ports (A,B,F,G,H) will return an error. Ports C and D must be put in Read mode before reading from C, D, or E using PortMode Property.

Return Type Variant

Default 0

Examples
\[ \text{data} = \text{handlerIo}.\text{get}_\text{Port}(\text{naPortC}) \]

C++ Syntax
\[ \text{HRESULT get}_\text{Port}(\text{tagNAMatHandlerPort Port, VARIANT* Data}); \]

Interface IHWMaterialHandlerIO
### Description
Reads a 4-bit value from Port C of the Aux I/O connector (pins 22-25) and the Material Handler IO (pins 21-24 Anritsu) - (pins 22-25 Avantest).

**Note:** These lines are connected to both the Handler IO and Aux IO in the PNA.

### VB Syntax

```vbnet
value = AuxIO.get_PortCData
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value</td>
<td>(variant)</td>
<td>Variable to store the returned data</td>
</tr>
<tr>
<td>AuxIO</td>
<td>(object)</td>
<td>A Hardware Auxiliary Input / Output object</td>
</tr>
</tbody>
</table>

### Return Type
Integer

### Default
None

### Examples

```vbnet
value = auxIo.get_PortCData 'Reading a value of 15 when in Positive Logic indicates Port C lines C0, C1, C2, C3 are High. If in Negative Logic they are Low.
```

### C++ Syntax

```c++
HRESULT get_PortData( VARIANT* Data );
```

### Interface
IHWAuxIO
GetPortNumber Method

**Description**
Returns the port number that is associated with the specified port name. These numbers are used with several commands to specify a PNA port.

To learn more, see [Remotely Specifying a Source Port](#).

**VB Syntax**
```
value = object.GetPortNumber(portName)
```

**Variable (Type) - Description**
- **value** *(Long)* - Variable to store the returned Port Number integer value.
- **object** *(Channel object)* - always more complete than capabilities object.
- A **Capabilities (object)** - use when a channel is not available.

- **portName** *(String)* - Name of the PNA port.
  - Use [SourcePortNames Property](#) to return a list of PNA port (string) names.
  - If an external source is selected, specify the external source name that is used in the [Select an External Source dialog](#).

**Return Type**
Long Integer

**Default**
Not Applicable

**Examples**
```
value = chan.GetPortNumber("Src2 Out1")  'Read
```

**C++ Syntax**
```
HRESULT GetPortNumber(BSTR name, long *number);
```

**Interface**
IChanne13
ICapabilities4

---

**Last Modified:**
- 12-Oct-2009 Added parens
- 23-May-2008 Added channel object
- 24-Apr-2008 Fixed example
- 30-Apr-2007 Edited for Ext Source Control
GetRaw2DData Method

**Description**
Returns raw data at all frequency and power data points for any GCA sweep. Previously 2D sweep only.

- When using SMART sweep, ALL data is returned including ALL background iteration sweeps.
- When using 2D sweeps, ALL data is returned.

Use the standard "get data" commands to return just the displayed data results (not the background sweeps).

A compression parameter must be present. Learn more.

**VB Syntax**
```vbnet
data = gca.GetRaw2DData location, format, param
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>data</td>
<td>Variant array in which to store returned measurement data.</td>
</tr>
<tr>
<td>gca</td>
<td>A GainCompression (object)</td>
</tr>
<tr>
<td>location</td>
<td>(enum NADataStore) - Where the data you want is residing. Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - naRawData</td>
</tr>
<tr>
<td></td>
<td>1 - naCorrectedData</td>
</tr>
<tr>
<td>format</td>
<td>(enum NADataFormat) - Format in which you would like the data. It does not have to be the displayed format. Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - naDataFormat_LinMag</td>
</tr>
<tr>
<td></td>
<td>1 - naDataFormat_LogMag</td>
</tr>
<tr>
<td></td>
<td>2 - naDataFormat_Phase</td>
</tr>
<tr>
<td></td>
<td>3 - naDataFormat_Polar*</td>
</tr>
<tr>
<td></td>
<td>4 - naDataFormat_Smith*</td>
</tr>
<tr>
<td></td>
<td>5 - naDataFormat_Delay -- <strong>Not valid for this command.</strong></td>
</tr>
<tr>
<td></td>
<td>6 - naDataFormat_Real</td>
</tr>
<tr>
<td></td>
<td>7 - naDataFormat_Imaginary</td>
</tr>
<tr>
<td></td>
<td>8 - naDataFormat_SWR</td>
</tr>
<tr>
<td></td>
<td>9 - naDataFormat_PhaseUnwrapped</td>
</tr>
<tr>
<td></td>
<td>10 - naDataFormat_InverseSmith</td>
</tr>
</tbody>
</table>

Learn more about Data Format.

* Specify Smith or Polar formats to obtain complex data pairs, which require a two-
dimensional array varData (numpts, 2) to accommodate both real and imaginary data. All scalar formats return a single dimension varData(numpts).

naDataFormat_Phase and naDataFormat_PhaseUnwrapped returns degrees.

**param** (String) Parameter of data to return. Not case-sensitive. The specified parameter need NOT be displayed. However, a compression parameter must be present. Learn more.

Choose from:

- "pin"  - (CompIn21) Input power at the compression point.
- "pout" - (CompOut21) Output power at the compression point.
- "gain"  - (CompGain21) Device gain (S21) at the compression point.
- "inputmatch"  - (CompS11) Input match at the compression point.
- "DeltaGain"  - (DeltaGain21) Measured Gain (watts) / Ref Gain (watts). Learn more.
- "AI1" and "AI2" - ADC measurements at the specified compression level. Learn more.

**Return Type** Variant Array

**Default** Not Applicable

**Examples**

data = gca.GetRaw2DData naRawData, naDataFormat_Real, "pin"

**C++ Syntax**

HRESULT GetRaw2DData(tagNADataStore location, tagNADataFormat format, BSTR data_name, VARIANT* pData);

**Interface** IGainCompression

Last Modified:

- 13-May-2011  Added returns background sweeps
- 31-Mar-2009  Added new params
- 12-May-2008  Edited for any GCA sweep
- 22-Oct-2007  MX New topic
**GetDataIm Method**

**Description**

Reads the Imaginary part of the data acquired from any GCA sweep. Previously 2D sweep only.

**VB Syntax**

```vb
data = gca.GetDataIm stim, dPoint, param
```

**Variable (Type) - Description**

- **data** Variant array in which to store returned measurement data.
- **gca** A GainCompression (object)
- **stim** (NAGCAIndexSelect)
  - **naFrequencySelect** - for the specified frequency data point, returns all of the measured data for each power stimulus.
  - **naPowerSelect** - for the specified power data point, returns all of the measured data for each frequency stimulus.
- **dPoint** Data point (Frequency or Power) for which data is returned.
- **param** Parameter of data to return. Not case-sensitive. Choose from:
  - "pin" - input power at each data point.
  - "pout" - output power at each data point.
  - "gain" - device gain (S21) at each data point.
  - "inputmatch" - input match (S11) at each data point.
  - "DeltaGain" - Measured Gain (watts) / Ref Gain (watts). Learn more.
  - "AI1" and "AI2" - ADC measurements at the specified compression level. Learn more.

**Return Type** Variant Array

**Default** Not Applicable

**Examples**

For the fifth frequency data point, returns ‘Power Output’ imaginary (phase) data from all power stimulus values. If there are 30 power sweep points, 30 values are returned.

```vb
data = gca.GetDataIm naFrequencySelect, 5, "pout"
```

For the 30th stimulus power data point, returns ‘Power Output’ imaginary (phase) data from all frequency stimulus values. If there are 201 frequency sweep points, 201 values
are returned.

```c
data = gca.GetDataIm naPowerSelect, 30, "pout"
```

**C++ Syntax**

```c
HRESULT GetDataIm(tagNAGCAIndexSelect index_select, int index,BSTR data_name, VARIANT* pData);
```

**Interface**

IGainCompression

---

Last Modified:

- 31-Mar-2009  Added new params
- 12-May-2008  Edited for any GCA sweep
- 22-Oct-2007  MX New topic
### GetDataRe Method

**Description**
Reads the REAL part of the data acquired from any GCA sweep. Previously 2D sweep only.

**VB Syntax**
```
data = gca.GetDataRe stim, dPoint, param
```

**Variable**

<table>
<thead>
<tr>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>data</strong> VARIANT array in which to store returned measurement data.</td>
</tr>
<tr>
<td><strong>gca</strong> A GainCompression (object)</td>
</tr>
<tr>
<td><strong>stim</strong> (NAGCAIndexSelect)</td>
</tr>
<tr>
<td>- <strong>naFrequencySelect</strong> - for the specified frequency data point, returns all of the measured data for each power stimulus.</td>
</tr>
<tr>
<td>- <strong>naPowerSelect</strong> - for the specified power data point, returns all of the measured data for each frequency stimulus.</td>
</tr>
<tr>
<td><strong>dPoint</strong> Data point (Frequency or Power) for which data is returned.</td>
</tr>
<tr>
<td><strong>param</strong> Parameter of data to return. Not case-sensitive. Choose from:</td>
</tr>
<tr>
<td>- &quot;<strong>pin</strong>&quot; - input power at each data point.</td>
</tr>
<tr>
<td>- &quot;<strong>pout</strong>&quot; - output power at each data point.</td>
</tr>
<tr>
<td>- &quot;<strong>gain</strong>&quot; - device gain (S21) at each data point.</td>
</tr>
<tr>
<td>- &quot;<strong>inputmatch</strong>&quot; - input match (S11) at each data point.</td>
</tr>
<tr>
<td>- &quot;<strong>DeltaGain</strong>&quot; - Measured Gain (watts) / Ref Gain (watts). Learn more.</td>
</tr>
<tr>
<td>- &quot;<strong>AI1</strong>&quot; and &quot;<strong>AI2</strong>&quot; - ADC measurements at the specified compression level. Learn more.</td>
</tr>
</tbody>
</table>

**Return Type** Variant Array

**Default** Not Applicable

**Examples**
For the fifth frequency data point, returns 'Power Output' REAL data from all power stimulus values. If there are 30 power sweep points, 30 values are returned.
```
data = gca.GetDataRe naFrequencySelect, 5, "pout"
```

For the 30th stimulus power data point, returns 'Power Output' REAL data from all frequency stimulus values. If there are 201 frequency sweep points, 201 values are
data = gca.GetDataRe naPowerSelect, 30, "pout"

**C++ Syntax**

```cpp
HRESULT GetDataRe(tagNAGCAIndexSelect index_select, int index, BSTR data_name, VARIANT* pData);
```

**Interface**

IGainCompression

---

**Last Modified:**

- 31-Mar-2009  Added new params
- 12-May-2008  Edited for any GCA sweep
- 22-Oct-2007  MX New topic
# GetRxLevelingConfiguration Method

**Description**  
This method returns a handle to a `RxLevelingConfiguration` object.

**VB Syntax**  
`chan.GetRxLevelingConfiguration()`

**Variable**  
**(Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chan</td>
<td>Channel</td>
<td>A Channel (object)</td>
</tr>
</tbody>
</table>

**Return Type**  
IRxLevelingConfiguration

**Default**  
Not Applicable

**Example**  
```vbnet
dim app as AgilentPNA835x.Application  
dim mgr as RxLevelingConfiguration  
set mgr = app.GetRxLevelingConfiguration()  
```

**C++ Syntax**  
```cpp
HRESULT GetRxLevelingConfiguration( IRxLevelingConfiguration **mgr);  
```

**Interface**  
IChannel17

---

**Last Modified:**  
3-Mar-2009   MX New topic
GetSourceByRole Method - Superseded

Description
This command is replaced by: chan.RoleDevice

Returns the name of a source that is assigned to the specified role.

VB Syntax
source = conv.GetSourceByRole (role)

Variable (Type) - Description

source (String) Source name, from Source Configuration dialog, that is assigned to the specified role.

conv A Converter Object

role (String) Role for which the source name will be returned. Use GetSourceRoles for a list of valid roles.

Return Type Not Applicable

Default Not Applicable

Examples
"LO" = conv.GetSourceByRole ("RF2")

C++ Syntax
HRESULT GetSourceByRole(BSTR roleID, BSTR deviceName);

Interface IConverter

Last Modified:
3-May-2011 superseded
2-Feb-2009 New topic
GetSourceRoles Method - Superseded

Description
This command is replaced by: chan.DefinedRoles

Returns the defined role names ("RF2", "LO1").

VB Syntax
conv.GetSourceRoles (roles)

Variable (Type) - Description
conv  A Converter Object
roles  (Variant array) Variable to store returned list of valid roles.

Return Type  Variant
Default  Not Applicable

Examples
conv.GetSourceRoles (roles)

C++ Syntax
HRESULT GetSourceRoles(VARIANT* roles);

Interface  IConverter

Last Modified:
3-May-2011  Superseded
3-Feb-2009  New topic
**GetReferenceMarker Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns a handle to the reference marker.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>meas.GetReferenceMarker</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Object</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>meas.GetReferenceMarker</code></td>
</tr>
</tbody>
</table>
| **C++ Syntax**            | `HRESULT GetReferenceMarker(IMarker**
|                           | refMarker)`                                |
| **Interface**             | IMeasurement                               |
GetRequiredEtermNames

Description

Returns an array of strings specifying the error terms required by the caltype's correction algorithm in order to correct the specified parameter.

This function interrogates a specific caltype (caltypeGUID) for the list of error terms it would need in order to correct the specified parameter. All the standard S Parameter calibration types embed port specifiers in the error term name. The specific port information is gleaned from the passed parameter. For example, to query the error term requirements specific to a two port cal on ports 1 and 3, issue this with a parameter of S13 or S31. The buffer names returned will be formatted in this way:

Full 1 Port SOLT(1,3):TransmissionTracking(3,1)

VB Syntax

EtermNames = GetRequiredEtermNames(CalTypeGUID As String, Parameter As String)

Variable (Type) - Description

caltypeGUID: [in] the GUID of the desired calibration type

parameter [in] string specifying the parameter to be corrected

EtermNames [out] array of strings containing the error term names.

Note: In C++ Allocated by server. Must be freed by caller using SysFreeString.

Return Type

Not Applicable

Default

Not Applicable

Examples

enames = GetRequiredEtermNames(ctGUID, Parm)

C++ Syntax

HRESULT GetRequiredEtermNames( BSTR caltypeGUID, BSTR parameter,  VARIANT* EtermNames )

Interface

ICalManager2
GetScalar Method

**Description**
Retrieves scalar data (ONE number per data point) from the specified location.

*Note:* This method exists on a non-default interface. If you cannot access this method, use the Get Data Method on IMeasurement.

**VB Syntax**
```
measData.getScalar location, format, numPts, data
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>measData</strong></td>
<td>An IArrayTransfer interface which supports the Measurement object</td>
</tr>
</tbody>
</table>
| **location**          | (enum NADataStore) - Where the data you want is residing. Choose from:
                         | 0 - naRawData
                         | 1 - naCorrectedData
                         | 2 - naMeasResult
                         | 3 - naRawMemory
                         | 4 - naMemoryResult
                         | 5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using DataToDivisor Method. |
| **format**            | (enum NADataFormat) - Format in which you would like the data. Choose from:
                         | 0 - naDataFormat_LinMag
                         | 1 - naDataFormat_LogMag
                         | 2 - naDataFormat_Phase
                         | 3 - naDataFormat_Polar
                         | 4 - naDataFormat_Smith
                         | 5 - naDataFormat_Delay
                         | 6 - naDataFormat_Real
                         | 7 - naDataFormat_Imaginary
                         | 8 - naDataFormat_SWR
                         | 9 - naDataFormat_PhaseUnwrapped
                         | 10 - naDataFormat_InverseSmith
                         | 11 - naDataFormat_Kelvin
                         | 12 - naDataFormat_Fahrenheit
                         | 13 - naDataFormat_Centigrade
| **numPts**            |                                                |
| **data**              |                                                |

*Note:* Polar, Smith, and Inverse Smith are invalid formats for this command. See Get Complex Method.
Learn more about Data Format.

\( numPts \)  
**long integer** - Number of data points requested  
[out] - specifies number of data elements returned  
in] - specifies the data being requested or the capacity of the \( dScalar \) array

\( data \)  
**single** - Array to store the scalar data.

**Return Type** Single  
**Default** Not Applicable

**Examples**
Dim dScalar() As Single  
Dim measData As IArrayTransfer  
Set measData = app.ActiveMeasurement  
Dim numpts as Long  
numpts = app.ActiveChannel.NumberOfPoints  
ReDim dScalar(numPoints)

measData.getScalar naCorrectedData, naDataFormat_LogMag, numpts, dScalar(0)  
Print dScalar(0), dScalar(1)

**C++ Syntax**
HRESULT getScalar(tagNADataStore DataStore, tagNADataFormat DataFormat, long* pNumValues, float* pVals)

**Interface** IArrayTransfer

---

Last Modified:  
1-Oct-2007  
Added temperature formats
**GetShortcut Method**

**Description**
Returns the Title, Path, and optional argument strings, of the specified Macro (shortcut). Use this method to list the titles and paths of macros in the analyzer.

**VB Syntax**

```vbnet
app.GetShortcut index, title, path, arguments
```

**Variable**

- **app**
  - Type: Application (object)

- **index**
  - Type: long
  - Description: Number of the macro. Use a number between 1 and 12.

- **title**
  - Type: string
  - Description: Title of the specified macro. (Appears in the softkey label)

- **path**
  - Type: string
  - Description: Pathname of the specified macro.

- **arguments**
  - Type: string
  - Description: Arguments for the specified macro

**Return Type**

String

**Default**

Not Applicable

**Example**

```vbnet
Dim t As String
Dim p As String
Dim arg As String
Dim i As Integer
For i = 1 To 12
    app.GetShortcut i, t, p, arg
    Print t, p
Next
```

**C++ Syntax**

```c++
HRESULT GetShortcut(long Number, BSTR* title, BSTR* pathname, BSTR* arguments)
```

**Interface**

IApplication

**Remarks**

Shortcuts can also be defined and accessed using the macro key on the front panel. However, the benefit of this feature is primarily for the interactive user.
**GetSnPData Method Superseded**

**Description**  
Note: this command is replaced by GetSnPDataWithSpecifiedPorts Method.

Reads SnP data from the selected measurement. Learn more about SnP that is returned from the PNA.

**VB Syntax**

```vbnet
data = meas.GetSnPData type
```

**Variable**  
(Variables) - Description

- **data**  
  Variant array to store the data.

- **meas**  
  A Measurement (object)

- **type**  
  (string) - Type of SnP data to return. If unspecified, <n> is set to 2. Choose from:
  - "S1P" returns data for the active measurement.
  - "S2P" returns data for the current 2-port measurement (4 S-parameters).
  - "S3P" returns data for the current 3 port measurement (9 S-parameters). Valid only on instruments with 3 ports or more.
  - "S4P" returns data for the current 4 port measurement (16 S-parameters). Valid only on instruments with 4 ports or more.

SnP data can be output using several data formatting options. See SnPFormat Property

**Return Type**  
Variant - 3 dimensional array.

- First dimension size is number of parameters returned.
- Second dimension size is number of points in the channel
- Third dimension size is 2 (real,imaginary)

For example:

Data(0,5,1) returns the imaginary value of the fifth data point of S11 (if the s2p request includes port #1)

**Default**  
Not Applicable

**Examples**

```vbnet
snp = meas.GetSnPData("s1p")
```

**C++ Syntax**

`HRESULT GetSnPData(BSTR snptype, VARIANT * response)`

**Interface**

IMeasurement3

---

Last Modified:
14-Nov-2008  Added returned data detail
### GetSnPDataWithSpecifiedPorts Method

**Description**

Note: This command replaces Get SnPData. This command is more explicit regarding the data to be returned, and works for PNA with multiport test sets.

Reads SnP data for the measurement by specifying the PNA port numbers. Learn more about SnP that is returned from the PNA.

**VB Syntax**

```vbnet
data = meas.GetSnPDataWithSpecifiedPorts ports
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>data</code></td>
<td>(Variant) array to store the data.</td>
</tr>
<tr>
<td><code>meas</code></td>
<td>A Measurement (object)</td>
</tr>
<tr>
<td><code>ports</code></td>
<td>(Variant Array) One-dimensional array containing the list of port numbers for which data is required.</td>
</tr>
</tbody>
</table>

**Return Type**

Variant - 3 dimensional array.

- First dimension size is number of parameters returned.
- Second dimension size is number of points in the channel
- Third dimension size is 2; format of the data is specified with SnPFormat Property.

For example:

Data(0,5,1) returns the imaginary part of the fifth data point of S11 (if the s2p request includes port #1)

**Default**

Not Applicable

**Example**

'This VBScript example can be pasted into a notepad file and run on the PNA as a macro. Learn how.

```vbnet
Dim pna
Dim meas
Dim param
Dim point
Dim snp
Dim ports

'List the port numbers for required data
ports = Array(3,4)
Set pna = CreateObject("AgilentPnA835x.application")
Set meas = pna.ActiveMeasurement
```
'limit amount of data to display
set chan=pna.ActiveChannel
chan.NumberOfPoints=2
snp = meas.GetSnPDataWithSpecifiedPorts (ports)
' returns a 3 dimensional array
' snp(param,point,data pair)
'-------------------------------------
' show me the data
For param = LBound(snp, 1) To UBound(snp, 1)
  MsgBox ("Parameter: " & (param + 1))
  For point = LBound(snp, 2) To UBound(snp, 2)
    MsgBox "Point:" & (point + 1) & "," & snp(param, point, 0) 
      & "," & snp(param, point, 1)
  Next
Next

C++ Syntax  HRESULT GetSnpDataWithSpecifiedPorts(VARIANT portsToMeasure,VARIANT*
response);

Interface  IMeasurement7

Last modified:

13-Jun-2011  Fixed data format
13-Nov-2008  Added detail to return type
9/18/06  MQ Added for multiport
getSourcePowerCalDataEx Method

Description  **Note:** This method replaces `getSourcePowerCalData Method`  
Retrieves (as variant data type) source power calibration data, if it exists, from the channel.

If the channel is sweeping the source backwards, then the first data point is the highest frequency value; the last data point is the lowest. Use the `Get X-Axis Values` command to return the X-axis values in the displayed order.

**Note:** This method returns a variant which is less efficient than methods available on the `ISourcePowerCalData` interface

VB Syntax  
```vbnet
data = chan.getSourcePowerCalDataEx (buffer, srcPort)
```

Variable  **(Type) - Description**

*data*  *(variant)* – Array to store the data.

*chan*  *(object)* – A `Channel` object

*buffer*  *(enum NAPowerCalBuffer)* - The requested source power cal data buffer.

0  - `naCorrectionValues`  – Last iteration of Cal data

1  - `naPriorIterationCorrectionValues`  – Prior iteration of Cal data. This argument can be used to determine the final power reading at each point of the power cal, for a cal that did not pass tolerance limits.

The following formula can be used to determine the power reading (in dB):

\[
\text{Power reading} = \text{Target power at the source port} + \text{specified power cal offset value} + \text{‘prior’ iteration corr value} – \text{actual power corr value}.
\]

The “actual” value in this equation is returned with `naCorrectionValues`.

*srcPort*  *(long integer)* – The source port for which calibration data is being requested.

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

Return Type  Variant array – automatically dimensioned to the size of the data.

Default  Not Applicable

Examples  
```vbnet
Dim varData As Variant
Const port1 As Long = 1
varData = chan.getSourcePowerCalDataEx (naCorrectionValues, port1)
'Print the data
```
For i = 0 to chan.NumberOfPoints - 1
Print varData(i)
Next i

**C++ Syntax**

```c++
HRESULT getSourcePowerCalDataEx(tagNASourcePowerCalBuffer bufSelect, long sourcePort, VARIANT *pData);
```

**Interface**

IChannel4

---

Last Modified:

- March 17, 2010 Added Prior argument
- 24-Apr-2008 Added note for string names
- 27-Jun-2007 Updated for PNA-X source port names
getSourcePowerCalDataScalarEx Method

**Description**

Note: This method replaces **getSourcePowerCalDataScalar Method**

Retrieves (as scalar values) source power calibration data, if it exists, from this channel. If the channel is sweeping the source backwards, then the first data point is the highest frequency value; the last data point is the lowest. Use the **Get X-Axis Values2** command to return the X-axis values in the displayed order.

Note: This method exists on a non-default interface. If you cannot access this method, use the **getSourcePowerCalDataEx Method** on IChannel4.

**VB Syntax**

```vbnet
chanData.getSourcePowerCalDataScalarEx buffer, srcPort, numValues, data
```

**Variable (Type) - Description**

- **chanData** (interface) – An **ISourcePowerCalData2** interface on the Channel object.

- **buffer** (enum **NASourcePowerCalBuffer**) - The requested source power cal data buffer.
  - 0 - **naCorrectionValues**  Last iteration of Cal data.
  - 1 - **naPriorIterationCorrectionValues**  Prior iteration of Cal data. This argument can be used to determine the final power reading at each point of the power cal, for a cal that did not pass tolerance limits.

  The following formula can be used to determine the power reading (in dB):

  \[ \text{Power reading} = \text{Target power at the source port} + \text{specified power cal offset value} + \text{\'prior\' iteration corr value} - \text{actual power corr value}. \]

  The "actual" value in this equation is returned with **naCorrectionValues**.

- **srcPort** (long integer) – The source port for which calibration data is being requested.

  Note: If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use **chan.getPortNumber** to translate the string into a port number. To learn more see **Remotely Specifying a Source Port**.

- **numValues** (long integer) – Number of data values.

  [out] – specifies number of data values returned.

  [in] – specifies number of values being requested (this must not be larger than the capacity of the data array).

- **data** (single) – Array to store the data.

**Return Type**

Single

**Default**

Not Applicable
Examples

Dim numValues As Long
Dim scalarCalValues() As Single
Dim chanData As ISourcePowerCalData2
Const port1 As Long = 1
numValues = app.ActiveChannel.NumberOfPoints
ReDim scalarCalValues(numValues)
Set chanData = app.ActiveChannel

chanData.getSourcePowerCalDataScalarEx naCorrectionValues,
port1, numValues, scalarCalValues(0)

'Print the data
For i = 0 to numValues - 1
Print scalarCalValues(i)
Next I

C++ Syntax

HRESULT getSourcePowerCalDataScalarEx(tagNASourcePowerCalBuffer bufSelect,
long sourcePort, long *pNumValues, float *pData);

Interface

ISourcePowerCalData2

Last Modified:

17-Mar-2010       Added Prior argument
24-Apr-2008       Added note for string names
27-Jun-2007       Updated for PNA-X source port names
GetStandard Method Superseded

Description
This command has been replaced with Get StandardByString

Returns standard acquisition data from the Cal Set. The returned data is complex pairs.

Learn more about Reading and Writing Cal Data

See examples of Reading and Writing Cal Set Data

VB Syntax
\[
data = \text{CalSet.getStandard}(\text{standard, rcv, src})
\]

Variable (Type) - Description
\[
data \text{ (Variant)} \text{ Two-dimensional safe array to store the returned data. Memory for the returned Variant is allocated by the PNA and must be released by client.}
\]

Note: See also getStandardComplex on the ICalData2 interface to avoid using the variant data type.

CalSet A Cal Set (object)

standard (enum NACalClass) Standard data to be read. Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

SOLT Standards
1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

TRL Standards
1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
5 - naTRL_Isolation

rcv \textbf{(long)} - Receiver Port

src \textbf{(long)} - Source Port

\textbf{Return Type} (\textit{variant})

\textbf{Default} Not Applicable

\textbf{Examples}
\begin{verbatim}
Dim varStd As Variant
Dim varStd2 As Variant

Cal Set.OpenCalSet( naCalType_TwoPortSOLT, 1, 2)
varStd = CalSet.getStandard(naSOLT_Thru,2,1)
varStd2 = Cal Set.getStandard(naSOLT_Thru,1,2)
Cal Set.CloseCalSet( )
\end{verbatim}

\textbf{C++ Syntax} HRESULT getStandard(tagNACalClass stdclass, long ReceivePort, long SourcePort, VARIANT* pData)

\textbf{Interface} ICalSet
GetStandardByString Method

Description
Returns standard acquisition data from the Cal Set. The returned data is complex pairs.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

VB Syntax
`data = calSet.GetStandardByString(stdName)`

Variable (Type) - Description

| data | (Variant) Two-dimensional safe array to store the returned data. Memory for the returned Variant is allocated by the PNA and must be released by client.
| Note: See also Get StandardComplexByString on the ICalData2 interface to avoid using the variant data type.

| calSet | A CalSet (Object) |
| stdName | (String) The string used to identify a particular standard in the Cal Set. An example string requesting the data for the Load standard in a full 2 port cal might be "S11C(3,3)". |

Return Type
Variant

Default
Not Applicable

Examples
See an example

C++ Syntax
`HRESULT GetStandardByString( BSTR bufferName, VARIANT* pdata)`

Interface
ICalSet2
GetStandardComplex Method Superseded

Description
This command is replaced with GetStandardComplexByString.

Returns standard acquisition data from the Cal Set. The returned data is complex pairs.

Learn more about Reading and Writing Cal Data

See examples of Reading and Writing Cal Set Data

**Note:** This method exists on a non-default interface. If you cannot access this method, use the GetStandard Method on ICal Set

**VB Syntax**
```
ICalData2.getStandardComplex class, rcv, src, numPts, real(), imag()
```

**Variable** (Type) - Description

**ICalData2** An ICalData2 pointer to the Cal Set object

**class** (enum NACalClass) Standard data to be read. Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

**SOLT Standards**
1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

**TRL Standards**
1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
rcv (long) - Receiver Port

src (long) - Source Port

numPts (Long) An In/Out parameter.
On the way **in**, you specify the **max** number of values being requested.
On the way **out**, the PNA returns number of values actually returned.

real() (single) - array to accept the real part of the calibration data. One-dimensional for the number of data points.

imag() (single) - array to accept the imaginary part of the calibration data. One-dimensional for the number of data points.

**Return Type** (single)

**Default** Not Applicable

**Examples**

Dim numpts as long
numpts = ActiveChannel.NumberOfPoints
ReDim r(numpts) ' real part
ReDim i(numpts) ' imaginary part
Dim Cal Set as Cal Set
set Cal Set = pna.GetCalManager.GetCalSetByGUID( txtGUID )
Dim sData As ICalData2
Set sData = Cal Set
sdata.getStandardComplex naSOLT_Open, 1, 1, numpts, r(0), i(0)

**C++ Syntax**

HRESULT getStandardComplex(tagNACalClass stdclass, long ReceivePort, long SourcePort, long* pNumValues, float* pReal, float* pImag)

**Interface** ICalData2
GetStandardComplexByString Method

Description
Returns standard acquisition data from the Cal Set.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

VB Syntax
ICalData3.GetStandardComplexByString stdName, lnumPoints, real(0), imag(0)

Variable (Type) - Description
ICalData3 An ICalData3 pointer to a CalSet (Object)

stdName (String) The string used to identify a particular standard in the Cal Set. An example string requesting the data for the Load standard in a full 2 port cal might be "S11C(3,3)".

lnumPoints (Long) An In/Out parameter.
On the way in, you specify the max number of values being requested.
On the way out, the PNA returns number of values actually returned.

real (Single) The real component of the complex data.

imag (Single) The imaginary component of the complex data.

Return Value Single

Default Not Applicable

Examples See example

C++ Syntax HRESULT GetStandardComplexByString( BSTR bufferName, long* lnumPoints, float* real, float* imag);

Interface ICalData3
GetStandardsList Method  Superseded

Description

Note: This command is replaced by CalSet.getStandardList2.

Returns the list of Standards contained in this Cal Set for the CalType specified in the OpenCal Set method. Learn more about reading and writing Cal Data using COM.

The list is a comma separated, textual representation of the error terms with the term name followed by the port path in parentheses.

Standard (n, n),
Standard (m, n)

Before calling this method you must open the Cal Set with OpenCal Set. If the Cal Set is not open, this method returns E_NA_Cal Set_ACCESS_DENIED.

Use StringToNACalClass to convert the list entrees to values that can be used with GetStandard and PutStandard.

Note: The port path designation (m n) indicates the receive and source ports for the measurement. Shorts, opens and loads are single port devices, designated in this list by (n n) where n equals the port to which the device is connected. These devices are all characterized by reflection measurements.
The dual port thru device is characterized by both transmission and reflection measurements in order to compensate for load match and tracking terms.
The notation (n n) indicates the reflection measurement for this device.
The notation (m n) indicates the transmission measurement, where the source and receive ports are different.

VB Syntax

CalSet.GetStandardsList (count, list)

Variable (Type) - Description

CalSet (object) - A Cal Set object

count (long [out]) - indicates the number of items returned in the list

list (string) - Variable to store the returned Comma separated list of items.

Return Type String

Default Not Applicable

Examples

dim count as Integer
dim list as string
OpenCalSet (naCalType_TwoPortSOLT, 1, 2)
GetStandardsList( count, list)
CloseCalSet( )

Assuming the Cal Set contained the full set of standards for this two port cal, the returned list would be:
"Open(1 1),
Short(1 1),

1859
Load(1 1),
Thru(1 1),
Isolation(2 1),
Open(2 2),
Short(2 2),
Load(2 2),
Thru(2 2),
Isolation(1 2)
Thru(2 1),
Thru(1 2)"

**C++ Syntax**

```cpp
HRESULT GetStandardsList( long* count, BSTR* list);
```

**Interface**

ICalSet
GetStandardList2 Method

**Description**
Returns a list of standards contained by this Cal Set for the specified Cal Type.

**VB Syntax**
```
list = calset.GetStandardList2(calType)
```

**Variable (Type) - Description**
- **calset** *(object)* - A *CalSet* object
- **list** *(Variant)* - Variant containing a string array of standards for the specified *calType*.
- **calType** *(String)* - The string used to identify Cal Set data as belonging to a specific Cal Type. This string is used as a filter so that only the standard names of interest are returned. If the prefix is empty, all names are returned.

An example prefix for a two port cal on ports 2 and 3 might be: "Full 2 Port Cal (2,3)".

**Return Type**
Variant

**Default**
Not Applicable

**Examples**
See an example

**C++ Syntax**
```
HRESULT GetStandardList2 (BSTR caltype, VARIANT* list)
```

**Interface**
ICalSet2
**GetStandardsForClass Method**

**Description**
Get the calibration standard numbers for a specified calibration class. To set the calibration number use `SetStandardsForClass Method`.

**VB Syntax**
```vbnet
calKit.GetStandardsForClass (calclassorder, std1, std2, std3, std4, std5, std6, std7)
```

**Variable (Type) - Description**
- **calKit**: A CalKit (object)
- **calclassorder (enum NACalClassOrder)**: Choose from:
  - 0 - naRefI_1_S11
  - 1 - naRefI_2_S11
  - 2 - naRefI_3_S11
  - 3 - naTran_1_S21
  - 4 - naRefI_1_S22
  - 5 - naRefI_2_S22
  - 6 - naRefI_3_S22
  - 7 - naTran_1_S12
  - 8 - naRefI_1_S33
  - 9 - naRefI_2_S33
  - 10 - naRefI_3_S33
  - 11 - naTran_1_S32
  - 12 - naTran_1_S23
  - 13 - naTran_1_S31
  - 14 - naTran_1_S13
  - 15 - naTRL_T
  - 16 - naTRL_R
  - 17 - naTRL_L
- **std1…std7 (long)**: Calibration Standard Number. Nominal values from 1 through 30. 0 indicates that a standard number has not been selected.
<table>
<thead>
<tr>
<th><strong>Return Type</strong></th>
<th>Not applicable</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Default</strong></td>
<td>Not applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>calkit.GetStandardsForClass naRef1_3_S11, std1, std2, std3, std4, std5, std6, std7</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT GetStandardsForClass(NACalClassOrder calclassorder, long std1, long std2, long std3, long std4, long std5, long std6, long std7)</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>ICalKit</td>
</tr>
</tbody>
</table>
**Read-only**

**GetStepDescription Method**

**Description**
Returns the description of the specified step in the calibration process. For an ECal User Characterization this returns the description of the specified step in the ECal User Characterization process.

**VB Syntax**

```vb
value = obj.GetStepDescription(n)
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>value (string)</td>
<td>Variable to store the returned number of steps.</td>
</tr>
<tr>
<td>obj</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>GuidedCalibration (object)</td>
</tr>
<tr>
<td></td>
<td>SMCType (object)</td>
</tr>
<tr>
<td></td>
<td>VMCType (object)</td>
</tr>
<tr>
<td></td>
<td>ECalUserCharacterizer (object)</td>
</tr>
</tbody>
</table>

**n** (Long) Step in the process.
Use GenerateSteps to determine the total number of steps.

**Return Type**
String

**Default**
Not Applicable

**Examples**

```vb
value = SMC.GetStepDescription(5)
```

**C++ Syntax**

```cpp
HRESULT get_GetStepDescription(long step, BSTR* str);
```

**Interface**
IGuidedCalibration
SMCType
VMCType
IECalUserCharacterizer

---

**Last Modified:**

4-Nov-2008 Added ECal object
GetSupportedALCModes Method

**Description**
Returns the valid ALC Modes for the PNA.
See ALCLevelingMode for a list of supported ALC Modes.

**VB Syntax**
```vbnet
value = chan.GetSupportedALCModes(sourcePort)
```

**Variable (Type) - Description**
- **value** Variant Array  variable to store the returned valid ALC Modes.
- **chan** (object) - A Channel object
- **sourcePort** (long integer) - Source port.

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

**Return Type**
Variant array

**Default**
Not Applicable

**Examples**
```vbnet
modes = chan.GetSupportedALCModes(4) 'Read
```

**C++ Syntax**
```cpp
HRESULT GetSupportedALCModes(long port, VARIANT * ALCModes);
```

**Interface**
IChannel9

---

**Last Modified:**
- 22-Apr-2010  Fixed C++syntax and interface (9)
- 24-Apr-2008  Added note for string names
- 30-Apr-2007  Edited for src strings
- 16-Jan-2007  MX New topic
GetTestResult Method

**Description**

Returns the result of limit line testing. There are three ways to use this command:

- If neither optional parameter is specified, limit results for ALL data is returned.
- If one parameter is specified (*start*), the limit result for that data point is returned.
- If both parameters are specified, limit results are returned beginning with *start*, and ending with (*start*+size)-1

**VB Syntax**

```
testRes = limts.GetTestResult [start,size]
```

**Variable**

- **testRes** *(enum NALimitTestResult)* - A dimensioned variable to store test results. If a limit line is not tested, a PASS is returned.
  
  - 0 - naLimitTestResult_None
  - 1 - naLimitTestResult_Fail
  - 2 - naLimitTestResult_Pass

- **limts** A LimitTest *(object)*

- **start** *(long)* - Optional argument. A start data point number to return limit test results.

- **size** *(long)* - Optional argument. Number of data points from *start* to return limit test results.

**Return Type**

Long Integer

**Default**

Not Applicable

**Examples**

```vba
Dim testRes As NALimitTestResult
testRes = limts.GetTestResult
Select Case testRes
    Case 1
        Print "Fails"
    Case 2
        Print "Pass"
End Select
```

**C++ Syntax**

```
HRESULT GetTestResult(long lStart, long lSize, tagNALimitTestResult *pVal)
```

**Interface**

ILimitTest
GetTraceStatistics Method

Description
Returns all four Trace Statistics. To retrieve individual Trace statistics, use Mean, PeakToPeak, StandardDeviation properties. Use ShowStatistics to display the statistics of the screen.

VB Syntax
meas.GetTraceStatistics pp, mean, stdev

Variable (Type) - Description
meas (Object) - A Measurement
pp, mean, stdev (Double) - Dimensioned variables to store the returned values

Return Type
Double

Default
Not Applicable

Examples
'Dimension variables
Dim pp As Double
Dim mean As Double
Dim stdev As Double
meas.GetTraceStatistics pp, mean, stdv

C++ Syntax
HRESULT GetTraceStatistics(double* pp, double* mean, double* stdDeviation)

Interface
IMeasurement
GetXAxisValues2 Method

**Description**

Returns the channel's X-axis values into a dimensioned Typed array. GetXAxisValues2 is a convenient method for determining the frequency of each point when the points are not linearly spaced - as in segment sweep.

**Note:** This method will fail if called using a scripting client such as VBScript or Agilent Vee. (see remarks)

**Note:** In Segment Sweep, chan.**NumberOfPoints** will return the total number of data points for the combined segments.

**VB Syntax**

`chan.GetXAxisValues2 numPts, data`

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chan</td>
<td>object</td>
<td>A Channel object</td>
</tr>
<tr>
<td>numPts</td>
<td>long integer</td>
<td>Number of data points in the channel</td>
</tr>
<tr>
<td>data</td>
<td>double</td>
<td>Single dimensioned array of data matching the number of points in the channel.</td>
</tr>
</tbody>
</table>

**Return Type**

double

**Default**

Not applicable

**Examples**

```vbnet
Dim App As Application
Set App = New Application
Dim numPoints As Long
Dim values() As Double
numPoints = App.ActiveChannel.NumberOfPoints
ReDim values(numPoints)
App.ActiveChannel.GetXAxisValues2 numPoints, values(0)
Print values(0), values(1)
```

**C++ Syntax**

`HRESULT GetXAxisValues2(long* pNumValues, double* stimulus)`

**Interface**

IChannel

**Remarks:**

This method will fail if called using a scripting client such as VBScript or Agilent Vee. Use the [GetXAxisValues](#) method as a replacement for these COM environments.

This method also cannot be called using late-bound typing in Visual Basic. For instance, if, in the example above, the first line were replaced with "Dim App as Object", then this method would fail.
GetXAxisValues Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Returns the stimulus values for the measurement. To understand how this property is useful, see <strong>IMeasurement2 Interface</strong>.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>data = meas.GetXAxisValues</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - <strong>Description</strong></td>
</tr>
<tr>
<td><strong>data</strong></td>
<td><em>(Variant)</em> Array to store the data.</td>
</tr>
<tr>
<td><strong>meas</strong></td>
<td>A Measurement <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Variant</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td>Dim varData As Variant</td>
</tr>
<tr>
<td></td>
<td>Dim i As Integer</td>
</tr>
<tr>
<td></td>
<td>varData = meas.GetXAxisValues</td>
</tr>
<tr>
<td></td>
<td>'Print Data</td>
</tr>
<tr>
<td></td>
<td>For i = 0 To meas.NumberOfPoints - 1</td>
</tr>
<tr>
<td></td>
<td>Print varData(i)</td>
</tr>
<tr>
<td></td>
<td>Next i</td>
</tr>
<tr>
<td></td>
<td>See C++ example</td>
</tr>
</tbody>
</table>

**C++ Syntax**

```
HRESULT GetXAxisValues(VARIANT* xData);
```

**Interface**

**IMeasurement2**

---

**Last Modified:**

15-Oct-2007  Added link to C++ example
**GetXAxisValues Method**

**Description**
Returns the channel's X-axis values. GetXAxisValues is a convenient method for determining the frequency of each point when the points are not linearly spaced - as in segment sweep.

See the [Measurement2 Interface](#) to learn how this method differs from `meas.GetXAxisValues`.

**Note:** This method returns a variant which is less efficient than `GetXAxisValues2`.

**Note:** In Segment Sweep, chan.NumberOfPoints will return the total number of data points for the combined segments.

**VB Syntax**
```vbnet
data = chan.GetXAxisValues
```

**Variable** *(Type) - Description*

- **data** Variant array to store the data.
- **chan** A Channel *(object)*

**Return Type** Variant

**Default** Not Applicable

**Examples**
```vbnet
Dim varData As Variant
Dim i As Integer
varData = chan.GetXAxisValues
'Print Data
For i = 0 To chan.NumberOfPoints - 1
    Print varData(i)
Next i
```

**C++ Syntax**
```cpp
HRESULT GetXAxisValues (VARIANT* xData)
```

**Interface** IChannel
HasCalType Method

**Description**
Verifies that the Cal Set object contains the error terms required to perform the specified correction (CalType) to an appropriate measurement.

The argument list includes specifiers for up to 3 ports. The number of arguments required depends on the CalType specified. The value for each port is set to 0 if not specified.

**VB Syntax**
```vbnet
check = CalSet.HasCalType(calType, p1, p2, p3)
```

**Variable (Type) - Description**

- `check` *(boolean)* - variable to store the returned value
  - `TRUE (1)` - Cal Set has all of the error terms necessary to apply the specified correction CalType.
  - `FALSE(0)` - Cal Set DOES NOT have all of the error terms necessary to apply the specified CalType.

- `CalSet` *(object)* - A Cal Set object

- `calType` *(enum as naCalType)* - type of correction to be applied. Choose from:

<table>
<thead>
<tr>
<th>Caltype</th>
<th>p arguments required</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 - naCalType_Response_Open</td>
<td>p1</td>
</tr>
<tr>
<td>1 - naCalType_Response_Short</td>
<td>p1</td>
</tr>
<tr>
<td>2 - *naCalType_Response_Thru</td>
<td>p1 (rcv), p2 (src)</td>
</tr>
<tr>
<td>3 - *naCalType_Response_Thru_And_Isol</td>
<td>p1 (rcv), p2 (src)</td>
</tr>
<tr>
<td>4 - naCalType_OnePort</td>
<td>p1</td>
</tr>
<tr>
<td>5 - naCalType_TwoPort_SOLT</td>
<td>p1, p2</td>
</tr>
<tr>
<td>6 - naCalType_TwoPort_TRL</td>
<td>p1, p2</td>
</tr>
<tr>
<td>7 - naCalType_None</td>
<td>N/A</td>
</tr>
<tr>
<td>8 - naCalType_ThreePort_SOLT</td>
<td>p1, p2, p3</td>
</tr>
<tr>
<td>9 - Custom</td>
<td>N/A</td>
</tr>
<tr>
<td>10 - naCalType_FourPort_SOLT</td>
<td>p1, p2, p3</td>
</tr>
<tr>
<td></td>
<td>(port 4 is assumed)</td>
</tr>
</tbody>
</table>

* order of port arguments is significant for these CalTypes

- `p1` *(long)* - required. This argument must be specified.
This specifies either:
- the one significant port for an open/short response cal or a 1 port cal.
- or one of the ports involved in a 2, 3, or 4 port cal
- or the receive port for a thru response / thru-isolation cal.

\( p2 \) (long) - required for any CalType involving more than one port
This specifies either:
- one of the ports involved in a 2, 3, or 4 port cal (order independent)
- or the source port for a thru response / thru-isolation cal

\( p3 \) (long) - required for 3 and 4-port cal
This specifies one of the ports involved in a 3 or 4 port cal (order independent)

**Return Type**: VARIANT_BOOL  
**Default**: Not Applicable  
**Examples**: \[ \text{value} = \text{CalSet.HasCalType(naCalType_TwoPort_TRL, 1, 2)} \]  
**C++ Syntax**: HRESULT HasCalType( tagNACalType, long port1, long port2, long port3, BOOL *pVal);  
**Interface**: ICalSet
**Hold Method**

**Description**  
Puts the channel in Hold - not sweeping.  
See `chans.Hold` to put ALL channels in hold.

**VB Syntax**  
`chan.Hold [sync]`

**Variable**  
*(Type)* - Description

`chan` A `Channel` *(object)*

`[sync]` The [sync] argument is ignored.  
Program control ALWAYS waits until the channel is in the Hold state.

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`chan.Hold`

**C++ Syntax**  
`HRESULT Hold`

**Interface**  
`IChannel`

---

**Last Modified:**

20-Apr-2010  Updated sync
**Hold (channels) Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Places ALL channels in hold mode. To resume all channels sweeping, use <code>chans.Resume</code>. (Must be the same instance of chans). To place a single channel in hold mode, use <code>channel.Hold</code> Method.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>chans.Hold</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td></td>
<td><code>chans</code> A Channel collection (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>chans.Hold</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT Hold();</td>
</tr>
<tr>
<td>Interface</td>
<td>IChannels2</td>
</tr>
</tbody>
</table>
**Write only**

**ImportDataSet Method**

**Description** Imports existing SMC power cal data into the current SMC calibration. The conditions that were present at the time the imported cal was performed must be present in the current SMC cal:

- The source and receive port numbers must match.
- If a THRU was used for the imported cal, the same is assumed for the current cal. Likewise if the power cals were ‘separate’.
- The Input and Output frequency range of the imported power cal MUST be within the Input and Output frequency range of the current SMC calibration. Interpolation will be applied to both cal data sets when the ranges do not exactly match.

The following error messages appear (they are not written to the PNA Error Log):

- *Interpolation target is out of range. Cannot interpolate* when incompatible frequency ranges occur
- *The necessary calibration standards were not found* when the port numbers and THRU method do NOT exactly match.

**VB Syntax**

```vbnet
smc.ImportDataSet (calset, dataName)
```

**Variable (Type) - Description**

- `smc` *(SMCType (object))*
- `calset` *(String)* Name of existing SMC Cal Set from which power cal data is imported.
- `dataName` *(String)* Name of the data set. Use “POWER_STEP”.

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```vbnet
smc.ImportDataSet("MySMCCal", "POWER_STEP")
```

See example program

**C++ Syntax**

```cpp
HRESULT ImportDataSet(BSTR csName, BSTR dataset);
```

**Interface** SMCType4

---

Last Modified: 1875
### ImportLibrary Method

**Description**
Imports an Equation Editor DLL.

**VB Syntax**
```vbnet
equation.ImportLibrary location
```

**Variable**

- **equation** (Type) - A `MeasurementEquation` object
- **location** (string) – Full path and filename of the *.dll to be imported.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
equation.ImportLibrary "C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll"

**C++ Syntax**
```cpp
HRESULT ImportLibrary( BSTR filename);
```

**Interface**
IMeasurementEquation2

---

Last Modified: 10-Jan-2011

New topic
Write only

Initialize Method

Description

Begins a calibration.

**Note:** `chan` must be the active channel.

For ECal User Characterization, use Initialize (ECal).

VB Syntax

```vbnet
obj.Initialize (chan, useCalStorPref)
```

Variable **(Type) - Description**

- **obj**
  Any of the following:
  - `GuidedCalibration` (object)
  - `SMCType` (object)
  - `VMCType` (object)

- **chan**
  (Long) Channel number to calibrate.

- **useCalStorPref**
  (boolean)
  - **True** or **1** - Assignment of Cal Set will be based on the setting of the `RemoteCalStoragePreference` COM property.
  - **False** or **0** – If the channel currently has a selected Cal Set, the calibration will be stored to that Cal Set. Otherwise, the assignment of Cal Set is based upon the setting of the `RemoteCalStoragePreference` COM property.

Return Type

Not Applicable

Default

Not Applicable

Examples

```vbnet
smc.Initialize(2,True)
```

C++ Syntax

```cpp
HRESULT put_Initialize(long channelnumber, VARIANT_BOOL bCalPref);
```

Interface

- `IGuidedCalibration`
- `SMCType`
- `VMCType`

Last Modified:

4-Nov-2008  Added Note about ECal User Char
About ECal User Characterization

InitializeEx Method

**Description**

Note: This property replaces Initialize (ECal) Method.

Initiates a User Characterization of an ECal module. The specified channel number must be an S-parameter measurement channel. The channel must already be calibrated using the same, or greater number of PNA ports as the ECal module. Also, the calibrated PNA ports must begin with Port 1 and use sequential port numbers.

For characterizations that are to be saved to the ECal module, the User Characterization number must already be set before issuing this command using CharacterizationNumber Property. For characterizations that are to be saved to the PNA disk memory, setting the User Characterization number is not necessary.

After this command is executed, subsequent commands can be used to query the number of measurement steps, issue the acquisition commands, query the connection description strings, and subsequently complete an Ecal User characterization.

**VB Syntax**

```vbscript
userChar.InitializeEx(chan, bool)
```

**Variable**

*userChar* ECalUserCharacterizer (object)

*chan* (Long) Channel number of a calibrated S-parameter channel.

*bool* (Boolean) Choose from:

- **True** Check ECal memory to ensure that a new characterization with the channel’s current number of points will fit in the module memory. Select for User Characterizations to be stored in internal ECal memory.

- **False** Skip the check. Select for User Characterizations is to be stored to PNA disk memory.

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```vbscript
userchar.InitializeEx(2, True)
```

**C++ Syntax**

```c++
HRESULT put.InitializeEx(long chanNum,VARIANT_BOOL bCheck);
```

**Interface** IECalUserCharacterizer2

Last Modified:

25-Aug-2009  MX New topic
About ECal User Characterization

Initialize Method Superseded

Description  
Note: This command is replaced with InitializeEx Method

Initiates a User Characterization of an ECal module. The specified channel number must be an S-parameter measurement channel. The User characterization number must already be set before issuing this command using CharacterizationNumber Property.

VB Syntax  
userChar.Initialize(chan)

Variable  
(Type) - Description

userChar  
ECalUserCharacterizer (object)

chan  
(Long) Channel number of a calibrated S-parameter channel.

Return Type  
Not Applicable

Default  
Not Applicable

Examples  
userChar.Initialize(2)

C++ Syntax  
HRESULT put_Initialize(long chanNum);

Interface  
IECalUserCharacterizer

Last Modified:
4-Nov-2008  New topic (8.33)
IsLibraryImported Method

**Description**
Returns whether a DLL has been imported into the PNA.

**VB Syntax**
`flag = equation.IsLibraryImported location`

**Variable (Type) - Description**
- **flag** (Boolean)
  - **True** - DLL has been imported.
  - **False** - DLL has NOT been imported.
- **equation** A MeasurementEquation object
- **location** (string) – Full path and filename of the *.dll.

**Return Type**
Variant boolean

**Default**
Not Applicable

**Examples**
`flag=equation.IsLibraryImported "C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll"`

**C++ Syntax**
`HRESULT IsLibraryImported(BSTR filename, VARIANT_BOOL* plImported);`

**Interface**
IMeasurementEquation2

---

Last Modified:

10-Jan-2011  New topic
### Write-only

#### Item Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Returns an object from the collection of objects.</th>
</tr>
</thead>
</table>
| Notes       | • The order of objects within a collection cannot be assumed.  
              • Most, but not all, PNA Collections are ‘1-based’ |

**VB Syntax**

```
Object.Item(n)
```

**Variable** *(Type) - Description*

**Object**  Any of the following *(collections)*:

- CalFactorSegments collection
- CalFactorSegmentsPMAR Collection
- Cal Sets collection
- Channels collection
- E5091Testset collection
- ECalModules Collection
- ExternalDevices Collection
- ExternalTestsets collection
- FOM collection
- GuidedCalibrationPowerSensors Collection
- LimitTest collection
- Measurements collection
- NaWindows collection
- PowerLossSegments collection
- PowerLossSegmentsPMAR Collection
- PowerSensors collection
- Segments collection
- Traces collection
- PowerMeterInterfaces Collection

*Learn more about collections in the PNA*
.Item  Optional - Item is the default property of a collections object and therefore can be called implicitly. For example, the following two commands are equivalent:

 Channels.Item(3).Averaging = 1
 Channels(3).Averaging = 1

\(\text{n (variant)}\) - number of the item in the collection.

- The **Measurements, Traces, and FOM** collections allow you to specify the name of the measurement as a string. For example:
  measCollection("CH_S11_1").InterpolateMarkers

- The **Cal Sets collection** allows you to specify the name of the cal set. For example:
  Calsets("MyCalSet").Description = "New Description"

- The **ExternalDevices Collection** allows you to specify the name of the external device. For example:
  Set extDev = externalDevices.Item("NewPMAR")

- The **GuidedCalibrationPowerSensors Collection** allows you to specify the name of the Power Sensor. For example:
  Set PowerSensor = GuidedCalibrationPowerSensors.Item.Name = "26GHzPowerSensor"

**Return Type**  (Object)

**Default**  Not Applicable

**Examples**  For \(i = 1\) to Traces.Count 1
  Traces.Item(i).YScale = .5dB
  Next i

**C++ Syntax**  HRESULT Item(VARIANT index, <interface>** pItem)

**Interfaces**  All listed above.

Last Modified:

8-Feb-2011  Added GuidedCalibrationPowerSensors
10-Sep-2008  Added CalSets name note
## LANConfigurationInitialize Method

**Description**
Performs an initialization (reset) of the PNA’s LAN configuration, as dictated by Section 8.14 of the LAN eXtensions for Instrumentation (LXI) standard (Version 1.1). This performs the same operation as pressing the LAN Reset button on the PNA [LAN Status dialog](#).

**VB Syntax**
```vbnet
app.LANConfigurationInitialize
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
app.LANConfigurationInitialize
```

**C++ Syntax**
```c++
HRESULT LANConfigurationInitialize ();
```

**Interface**
IA.Application13
## LaunchCalWizard Method

### Description
Launches the S-parameter Cal Wizard on the PNA and does not return until the Cal Wizard is dismissed.

To launch the Cal Wizard for a PNA Application, use the [LaunchDialog Method](#).

**Note:** The Cal Wizard operates on the active measurement. Therefore, activate the measurement to be calibrated before launching the Cal Wizard.

### VB Syntax
```
success = app.LaunchCalWizard(newCS)
```

### Variable
- **success** *(boolean)* - variable to store the returned value
  - **True** - The Cal was completed
  - **False** - The Cal was canceled without completing the calibration.

- **app** *(An Application object)*

- **newCS** *(boolean)*
  - **True** - Cal will be performed on a new Cal Set.
  - **False** - Cal will be performed using the existing Cal Set assigned to the channel. If no Cal Set is found, a new Cal Set will be created.

- **Return Type** Boolean

- **Default** Not Applicable

### Example
```
dim bSuccess as boolean
dim bNewCalset as boolean
bNewCalSet = false
bSuccess = app.LaunchCalWizard( bNewCalSet)
```

### C++ Syntax
```
HRESULT LaunchCalWizard(VARIANT_BOOL bCalSuccess)
```

### Interface
- IApplication

---

**Last Modified:**

5-Sep-2008  S-parameter only
LaunchDialog Method

**Description**

Launches the specified dialog box.

The Calibration Wizard dialog that appears depends on the active channel. For example, if a Gain Compression channel is active, then the GCA Cal Wizard appears. Use `meas.Activate` to activate a measurement and channel.

Remote operation returns after the dialog is dismissed.

To invoke the Cal Wizard and have it return immediately, then use `Syst:Corr:Wiz` with the SCPI Parser object.

**VB Syntax**

```
app.LaunchDialog dialog, [data]
```

**Variable**  

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
<tr>
<td>dialog</td>
<td>(String) Dialog box to launch. Choose from:</td>
</tr>
<tr>
<td></td>
<td>&quot;SourcePowerCal&quot; See this dialog.</td>
</tr>
<tr>
<td></td>
<td>&quot;PowerMeterSettings&quot; See this dialog.</td>
</tr>
<tr>
<td></td>
<td>“PathConfiguration” See this dialog.</td>
</tr>
<tr>
<td></td>
<td>“CalibrationWizard” Depends on the channel</td>
</tr>
<tr>
<td></td>
<td>“CalibrationSelection” See this dialog.</td>
</tr>
</tbody>
</table>

| [data]   | (Optional argument) Reserved for future use. |

**Return Type**

Not Applicable

**Default**

Not Applicable

**Examples**

```
app.LaunchDialog "SourcePowerCal"
```

**C++ Syntax**

`LaunchDialog( BSTR dialog, [defaultValue(0)] VARIANT dialogData)`

**Interface**

IAplication11

[See the PNA Object Model](#)
LaunchPowerMeterSettingsDialog Method

Description
Launches the Power Meter Settings dialog on the PNA. Changing certain values from that dialog will change values of the corresponding properties on this COM object.

VB Syntax
pwrCal.LaunchPowerMeterSettingsDialog

Variable (Type) - Description
pwrCal A SourcePowerCalibrator (object)

Return Type
None

Default
Not Applicable

Examples
powerCalibrator.LaunchPowerMeterSettingsDialog

C++ Syntax
HRESULT put_LaunchPowerMeterSettingsDialog();

Interface
ISourcePowerCalibrator2
### LoadConfiguration Method

**Description**
Loads the named configuration onto the specified channel. Use [Configurations Method](#) to return the configuration names that are stored on the PNA.

**VB Syntax**
```vbnet
pathMgr.LoadConfiguration ch, name
```

**Variable**
- **(Type)** - **Description**
  - `pathMgr` PathConfigurationManager *(object)*
  - `ch` *(Long)* Channel number of the configuration to be saved.
  - `name` *(String)* Configuration name. "Default" is the default factory configuration.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
path.LoadConfiguration 2, "myMixer"
```

**C++ Syntax**
```cpp
HRESULT LoadConfiguration (long channelNum, BSTR configName );
```

**Interface**
IPathConfigurationManager

---

Last Modified:

14-Dec-2006   MX New topic
LoadENRFile Method

**Description**
Loads an ENR file from disk into PNA memory. This file is typically provided by the manufacturer of the noise source.

**VB Syntax**
`enr.LoadENRFile (filename)`

**Variables**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>enr</code></td>
<td>An ENRFile (object)</td>
</tr>
<tr>
<td><code>filename</code></td>
<td>(String) - Absolute path and filename of the ENR file.</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
See example program

**C++ Syntax**
`HRESULT LoadENRFile(BSTR filename);`

**Interface**
IENRFile

---

Last Modified:

2-Aug-2007    MX New topic
# LoadFile Method

**Description**
Loads a previously-configured mixer attributes file (.mxr)

**VB Syntax**
```vbnet
obj.LoadFile (filename)
```

**Variable (Type) - Description**
- `obj` A [Mixer Interface](#) pointer to the [Measurement](#) (object)
  - Or
  - A [Converter Object](#)
- `filename` (String) Full path, file name, and .mxr extension of the mixer attributes file.
  - Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents".

**Return Type**
String

**Default**
Not Applicable

**Examples**
```vbnet
mixer.LoadFile ("C:/Program Files/Agilent/Network Analyzer/Documents/myMixer.mxr")
```

**C++ Syntax**
```cpp
HRESULT LoadFile(BSTR newVal)
```

**Interface**
IMixer
IConverter

---

Last Modified:

- 2-Feb-2009  Added converter
LoadTheme Method

**Description**  
Load a color theme from a disc file.

**VB Syntax**  
`colors.LoadTheme (filename)`

**Variable (Type) - Description**
- `colors` A `ComColors` (object)
- `filename` (String) - Path and filename of the theme to load.

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`colors.LoadTheme = "c:/Program Files/Agilent/Network Analyzer/Colors/Theme1.colors"

**C++ Syntax**  
`HRESULT LoadTheme(BSTR filename);`

**Interface**  
`IComColors`

---

Last Modified:  
7-Aug-2009    MX New topic
About Triggering

ManualTrigger Method

Description
Triggers the analyzer when TriggerSetup.Source = naTriggerManual.

Note: An SMC Fixed Output measurement cannot be triggered using this command. For more information, see the example program.

VB Syntax
app.ManualTrigger [sync],[timeout]

Variable (Type) - Description
app An Application (object)
[sync] (boolean) - Optional argument.
A variable set to either True or False.
True - The analyzer waits until the trigger is completed to process subsequent commands.
False - Subsequent commands are processed immediately (the default setting).

timeout (long) - Optional argument.
If sync is true, timeout sets the amount of time the PNA will wait until continuing program execution. Units are milliseconds. A value of -1 (the default setting) causes the PNA to wait indefinitely.
If sync is False, the timeout setting is ignored.

Return Type Not Applicable
Default Not Applicable

Examples
' After Manual trigger is executed, the PNA will wait 1 second to continue program execution
Dim wait as Boolean
wait = True
app.ManualTrigger wait, 1000

C++ Syntax
HRESULT ManualTrigger(VARIANT_BOOL bSynchronize, long timeout)

Interface IApplication

Last Modified:
12-Jul-2007 Modified link
**MessageText Property**

**Description**
Returns text for the specified eventID

**VB Syntax**
`app.MessageText, eventID, message`

**Variable**
*(Type) - Description*

- **app**
  An `Application` (object)

- **eventID**
  (enum `naEventID`) Choose from the list in [Working with the Analyzer's Events](#)

- **message**
  (string) - variable to store the returned message

**Return Type**
String

**Default**
Not Applicable

**Examples**
`RFNA.MessageText naEventID_ARRANGE_WINDOW_EXCEED_CAPACITY, message`

**C++ Syntax**
`HRESULT get_MessageText( tagNAEventID msgID, BSTR* message)`

**Interface**
IAplication
NextIFBandwidth Method

**Description**  
A function that returns the Next higher IF Bandwidth value. Use to retrieve the list of available IFBandwidth settings.

**VB Syntax**  
`chan.Next_IFBandwidth bw`

**Variable**  
- **Type** - `chan`  
- **Description** - A Channel (object)

- **Type** - `bw`  
- **Description** - (double) - The argument that you use to send an IFBandwidth. The function uses this argument to return the Next higher IFbandwidth.

**Return Type**  
Double

**Default**  
Not Applicable

**Examples**  
```
Public pnbw As Double 'declare variable outside of procedure

pnBW = chan.IFBandwidth 'put the current IFBW in pnBW
chan.Next_IFBandwidth pnBW 'function returns the Next higher IFBandwidth.
chan.IFBandwidth = pnBW 'set IFBW to the Next value
```

**C++ Syntax**  
`HRESULT Next_IFBandwidth (double *pVal)`

**Interface**  
IChannel
**NumberofGroups Method**

**Description**
Sets the number of trigger signals the channel will receive. After the channel has received that number of trigger signals, the channel switches to Hold mode.

**VB Syntax**
```
chan.NumberOfGroups num, sync
```

**Variable (Type) - Description**

- **chan** (Channel (object))
- **num** (long integer) Number of trigger signals the channel will receive. Choose any number between 1 and 2 million.
- **sync** (boolean)
  Variable set to either:
  True - subsequent commands are not processed until the groups are complete. Do not use with manual trigger.
  False - subsequent commands are processed immediately.

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**
```
chan.NumberOfGroups 5, False
```

**C++ Syntax**
```
HRESULT NumberofGroups(long count, VARIANT_BOOL bWait)
```

**Interface** IChannel
OpenCalSet Method  Superseded

**Description**
This command is no longer necessary. The CalSet.get... and put... commands that required this command have been replaced.

Open the Cal Set to read/write a particular CalType. Learn more about reading and writing Cal Data using COM.

This method is a prerequisite to several other Cal Set methods.

A Cal Set can contain more than one CalType. This method opens the Cal Set and allows access to a particular set of terms. Subsequent commands like getErrorTerm use this information to access the correct error terms in the Cal Set. For example:

```
cset.OpenCalSet (naCalType_TwoPortSOLT, 3, 2)
cset.PutErrorTerm(naDirectivity, 1, 1, Buffer)
```

The directivity error term for port 1 could belong to any number of caltypes: Full1Port (S11), Full2Port (12), Full2Port (13) or Full3Port (123). The CalType and port specifiers in OpenCalSet directs the uploaded directivity term to the correct set of error terms.

To close the Cal Set, see CloseCalSet.

### VB Syntax

```vb
CalSet.OpenCalSet (CalType, p1, p2, p3)
```

### Variable (Type) - Description

- **CalSet** *(object)* - A Cal Set object

- **CalType** *(enum as naCalType)* - type of correction to be applied. Choose from:

<table>
<thead>
<tr>
<th>CalType</th>
<th>p arguments required</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 - naCalType_Response_Open</td>
<td>p1</td>
</tr>
<tr>
<td>1 - naCalType_Response_Short</td>
<td>p1</td>
</tr>
<tr>
<td>2 - *naCalType_Response_Thru</td>
<td>p1 (rcv), p2 (src)</td>
</tr>
<tr>
<td>3 - *naCalType_Response_Thru_And_Isol</td>
<td>p1 (rcv), p2 (src)</td>
</tr>
<tr>
<td>4 - naCalType_OnePort</td>
<td>p1</td>
</tr>
<tr>
<td>5 - naCalType_TwoPort_SOLT</td>
<td>p1, p2</td>
</tr>
<tr>
<td>6 - naCalType_TwoPort_TRL</td>
<td>p1, p2</td>
</tr>
<tr>
<td>7 - naCalType_None</td>
<td>N/A</td>
</tr>
<tr>
<td>8 - naCalType_ThreePort_SOLT</td>
<td>p1, p2, p3</td>
</tr>
<tr>
<td>9 - Custom</td>
<td>N/A</td>
</tr>
</tbody>
</table>
10 - naCalType_FourPort_SOLT p1, p2, p3 (port 4 is assumed)

* order of port arguments is significant for these CalTypes

\( p1 \) (long) - required. This argument must be specified.
This specifies either:
- the one significant port for an open/short response cal or a 1 port cal.
- or one of the ports involved in a 2 or 3 port cal
- or the receive port for a thru response / thru-isolation cal.

\( p2 \) (long) - required for any caltype involving more than one port
This specifies either:
- one of the ports involved in a 2 or 3 port cal (order independent)
- or the source port for a thru response / thru-isolation cal

\( p3 \) (long) - required only for 3 port cal
This specifies either:
- one of the ports involved in a 3 port cal (order independent)

**Return Type** None

**Default** Not Applicable

**Examples**

```csharp
CalSet.OpenCalSet naCalType_ThreePort_SOLT, 3, 2, 1
```

**C++ Syntax**

```csharp
HRESULT OpenCalSet ( naCalType, port1, [optional] port2, [optional] port3);
```

**Interface** ICalSet
### Parse Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Allows the use of COM to send a SCPI command. See a C++ example of how to return error information when using this command.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Note:</strong></td>
<td>The SCPIStringParser Methods can NOT be used with SCPI Status Reporting. However, the *OPC? will work.</td>
</tr>
</tbody>
</table>

**VB Syntax**

```vbnet
scpi.Parse("SCPI command")
```

<table>
<thead>
<tr>
<th><strong>Variable</strong></th>
<th><strong>(Type)</strong> - <strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><code>scpi</code></td>
<td>A <code>ScpiStringParser</code> (object)</td>
</tr>
<tr>
<td><code>SCPI command</code></td>
<td>(string) - Any valid SCPI command</td>
</tr>
</tbody>
</table>

| **Return Type** | String |
| **Default** | Not Applicable |

**Examples**

```vbnet
Dim scpi As ScpiStringParser
Set scpi = app.ScpiStringParser
Dim startfreq As Double
startfreq = 100e6
' scpi.Parse "Sens:Freq:Start " & startfreq'Write
```

```vbnet
Dim str As String
str = scpi.Parse("Sens:Freq:Start?")'Read
```

**C++ Syntax**

```cpp
HRESULT Parse(BSTR SCPI_Command, BSTR *pQueryResponse)
```

**Interface**

IScpiParamStringParser

---

**Last Modified:**
- **April 27, 2009** - Added note
- **1-Jan-2007** - Corrected example
**Write-only**

**Factory Preset Settings**

**Preset Method**

**Description**  
**Application Object:** Deletes all traces and windows. In addition, resets the analyzer to factory defined default settings and creates an S11 measurement named "CH1_S11_1" in window 1.

**Channel Object:** Resets the channel (object) to factory defined default settings. Does NOT delete the current measurements or add a new measurement.

**VB Syntax**  
app.Preset
chan.Preset

**Variable**  
(Type) - Description

app  An Application (object)
chan A Channel (object)

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
app.Preset

**C++ Syntax**  
HRESULT Preset()

**Interface**  
IApplication
IChannel
# PreviousIFBandwidth Method

**Description**  
A function that returns the previous IF Bandwidth value. Use to retrieve the list of available IFBandwidth settings.

**VB Syntax**  
`chan.Previous_IFBandwidth bw`

**Variable**  
**Type** - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>chan</code></td>
<td>A Channel (object)</td>
</tr>
<tr>
<td><code>bw</code></td>
<td>(double) - The argument that you use to send an IFBandwidth. The function uses this argument to return the previous IFBandwidth.</td>
</tr>
</tbody>
</table>

**Return Type**  
Double

**Default**  
Not Applicable

**Examples**  
```vbnet
Public pnbw As Double 'declare variable outside of procedure
PreBW = chan.IFBandwidth 'put the current IFBW in PreBW
chan.Previous_IFBandwidth PreBW 'function returns the Previous IFBandwidth of the current one.
chan.IFBandwidth = PreBW 'set IFBW to the previous value
```

**C++ Syntax**  
`HRESULT Previous_IFBandwidth (double *pVal)`

**Interface**  
IChannel
### PrintToImage Method

**Description**
Saves the screen image to a bitmap file.

**VB Syntax**
```vbnet
app.PrintToFile filename
```

**Variable**
- **app** (Application (object))
- **filename** (string) Full path, file name, and extension of the screen image file.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents". Use one of the following extensions:

- .bmp - not recommended due to large file size
- .jpg - not recommended due to poor quality
- .png - recommended

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
app.PrintToFile "C:/Program Files/Agilent/Network Analyzer/Documents/myfile.png"
```

**C++ Syntax**
`HRESULT PrintToFile(BSTR bstrFile)`

**Interface**
IApplication
PutComplex Method

Description
Puts real and imaginary data into the specified location. This method forces the channel into Hold mode to prevent the input data from being overwritten. Learn more about reading and writing Cal Data using COM.

Data put in the raw data store will be re-processed whenever a change is made to the measurement attributes such as format or correction.

Data put in the measurement results store will be overwritten by any measurement attribute changes.

See also putNAComplex

VB Syntax
```
measData.putComplex location, numPts, real(), imag(), [format]
```

Variable (Type) - Description

**measData** An IArrayTransfer interface which supports the Measurement object

**location** (enum NADataStore) Where the Data will be put. Choose from:
- 0 - naRawData
- 1 - naCorrectedData
- 2 - naMeasResult
- 3 - naRawMemory
- 4 - naMemoryResult
- 5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using DataToDivisor Method.

**numPts** (long integer) - Number of data points in the channel

**real()** (single) - Array containing real data values

**imag()** (single) - Array containing imaginary data values

**format** (enum NADataFormat) optional argument - display format of the real and imaginary data. Only used if destination is naMeasResult or naMemoryResult buffer. If unspecified, data is assumed to be in naDataFormat_Polar

- 0 - naDataFormat_LinMag
- 1 - naDataFormat_LogMag
- 2 - naDataFormat_Phase
- 3 - naDataFormat_Polar
- 4 - naDataFormat_Smith
- 5 - naDataFormat_Delay
- 6 - naDataFormat_Real

1903
7 - naDataFormat_Imaginary
8 - naDataFormat_SWR
9 - naDataFormat_PhasUnwrapped
10 - naDataFormat_InverseSmith
11 - naDataFormat_Kelvin
12 - naDataFormat_Fahrenheit
13 - naDataFormat_Centigrade

Learn more about Data Format.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vba
Dim measData As IArrayTransfer
Set measData = app.ActiveMeasurement
measData.putComplex naMemoryResult, 201,
real(0), imag(0), naDataFormat_SWR
```

**C++ Syntax**
```cpp
HRESULT putComplex( tagNADataStore DataStore, long lNumValues, float* pReal,
float* pImag, tagDataFormat displayFormat)
```

**Interface**
IArrayTransfer

---

Last Modified:

1-Oct-2007  Added temperature formats
PutDataComplex Method

Description
Puts complex data into the specified location. This method forces the channel into Hold mode to prevent the input data from being overwritten.

VB Syntax
meas.putDataComplex location, data

Variable (Type) - Description
meas A measurement (object)
location (enum NADataStore) Where the Data will be put. Choose from:

0 - naRawData
1 - naCorrectedData
2 - naMeasResult
3 - naRawMemory
4 - naMemoryResult
5 - naDivisor - When reading data from, or writing data to, the normalization divisor, you must first create a divisor trace using DataToDivisor Method.

- Data put in 0 - naRawData will be re-processed whenever a change is made to the measurement attributes such as format or correction.
- Data put in 2 - naMeasResult will be overwritten by any measurement attribute changes.
- When putting data into 3 - naRawMemory:
  1. Put the analyzer in hold mode
  2. Call DataToMemory to initialize a memory buffer
  3. Call putDataComplex(naRawMemory, data)
This ensures that the memory buffer is appropriately initialized before receiving new data.

data (variant) - A two-dimensional variant array.

Note: All buffers except naMeasResult and naMemoryResult require Complex data

Return Type
Not Applicable

Default
Not Applicable

Examples
' Put 201 points worth of raw (complex) data into the measurement
' Note that an array of complex numbers is represented by a 2-D
array where the first rank is the number of points, and the 2nd rank is always size 2 (max index 1) representing the Real and Imag parts of the complex number.

' complex array of data (2nd dimension of size 2 represents Re/Im
Dim data(200,1) 
For i = 0 to 200
' Set Real part of data point i
data(i,0) = i/200;
' Set Imag part of data point i
data(i,1) = i/200;
Next
app.ActiveMeasurement.putDataComplex naRawData, data

C++ Syntax HRESULT putDataComplex(tagNADataStore DataStore, VARIANT complexData)

Interface IMeasurement
PutENRData Method

**Description**  Write ENR calibration data to PNA memory. All of the frequency and ENR data must be sent at the same time.

**VB Syntax**  
```
enr.PutENRData (vData)
```

**Variable**  
**(Type)** - **Description**

- **enr**  An [ENRFile](#) (object)
- **vData**  (Variant array) - ENR data. Frequency value in Hz, followed by corresponding ENR value in dB. Enter as many data pairs as necessary.

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  
See example program

**C++ Syntax**  
```cpp
HRESULT PutENRData(VARIANT vdata);
```

**Interface**  IENRFile

---

Last Modified:

2-Aug-2007   MX New topic
### PutErrorTerm Method - Superseded

**Description**

Note: This command is replaced by `PutErrorTermByString`.

Puts variant error term data into the error-correction buffer.

Learn about reading and writing Calibration data.

**VB Syntax**

```vba
cal.putErrorTerm(term, rcv, src, data)
```

**Variable**

- `cal` (A Calibrator **object**)
- `term` (**enum As NaErrorTerm**)
  - `naErrorTerm_Directivity_Isolation`
  - `naErrorTerm_Match`
  - `naErrorTerm_Tracking`
- `rcv` (**long integer**) - Receiver Port
- `src` (**long integer**) - Source Port
- `data` (**variant**) Error term data in a two-dimensional array (0:1, 0:numpts-1).

<table>
<thead>
<tr>
<th>To get this Error Term</th>
<th>Specify these parameters:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Error Term</strong></td>
<td><strong>term</strong></td>
</tr>
<tr>
<td>Fwd Directivity</td>
<td><code>naET_Directivity_Isolation</code></td>
</tr>
<tr>
<td>Rev Directivity</td>
<td><code>naET_Directivity_Isolation</code></td>
</tr>
<tr>
<td>Fwd Isolation</td>
<td><code>naET_Directivity_Isolation</code></td>
</tr>
<tr>
<td>Rev Isolation</td>
<td><code>naET_Directivity_Isolation</code></td>
</tr>
<tr>
<td>Fwd Source Match</td>
<td><code>naErrorTerm_Match</code></td>
</tr>
<tr>
<td>Rev Source Match</td>
<td><code>naErrorTerm_Match</code></td>
</tr>
<tr>
<td>Fwd Load Match</td>
<td><code>naErrorTerm_Match</code></td>
</tr>
<tr>
<td>Rev Load Match</td>
<td><code>naErrorTerm_Match</code></td>
</tr>
<tr>
<td>Fwd Reflection Tracking</td>
<td><code>naErrorTerm_Tracking</code></td>
</tr>
<tr>
<td>Rev Reflection Tracking</td>
<td><code>naErrorTerm_Tracking</code></td>
</tr>
<tr>
<td>Fwd Trans Tracking</td>
<td><code>naErrorTerm_Tracking</code></td>
</tr>
<tr>
<td>Rev Trans Tracking</td>
<td><code>naErrorTerm_Tracking</code></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th><code>rcv</code></th>
<th><code>src</code></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fwd Directivity</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Directivity</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Isolation</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Isolation</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Source Match</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Source Match</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Load Match</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Load Match</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Reflection Tracking</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Rev Reflection Tracking</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Fwd Trans Tracking</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Rev Trans Tracking</td>
<td>1</td>
<td>2</td>
</tr>
</tbody>
</table>

1908
<table>
<thead>
<tr>
<th><strong>Return Type</strong></th>
<th>Not Applicable</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>
| **Examples**      | Dim varError As Variant  
|                   | varError = cal.putErrorTerm (naErrorTerm_Tracking, 2, 1, VarData) |
| **C++ Syntax**    | HRESULT putErrorTerm(tagNAErrorTerm ETerm, long ReceivePort, long SourcePort, VARIANT varData) |
| **Interface**     | ICalibrator       |
PutErrorTerm Method  Superseded

Description  This command is replaced with PutErrorTermByString

Puts error term data into the Cal Set.

Learn more about Reading and Writing Cal Data

See examples of Reading and Writing Cal Set Data

VB Syntax  

```
CalSet.putErrorTerm (term, rcv, src, data)
```

Variable  (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CalSet</td>
<td>Object</td>
<td>A CalSet Object</td>
</tr>
</tbody>
</table>
| term     | (enum As NaErrorTerm2) | Error Term. Choose from:
| rcv      | (long integer) | Receiver Port |
| src      | (long integer) | Source Port |
| data     | (variant) | Error term data in a two-dimensional array (0:1, 0:numpts-1). The data must be complex pairs. |

Note: See also PutErrorTermComplex on the ICalData2 interface to avoid using the variant data type.

Return Type  Not Applicable

Default  Not Applicable

Examples  See an Example

C++ Syntax  

```
HRESULT putErrorTerm(tagNAErrorTerm2 ETerm, long ReceivePort, long SourcePort, VARIANT varData)
```

Interface  ICalSet
PutErrorTermByString Method

**Description**
Puts error term data into the Cal Set.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

**VB Syntax**
calSet.PutErrorTermByString(errorName, vdata)

**Variable (Type) - Description**

- **calSet** (Object) A CalSet Object
- **errorName** (String) The string name used to identify a particular error term in the Cal Set. An example string for port 3 directivity in a full 2 port cal might be "Directivity(3,3)". To determine the string names of error terms, see GetErrorTermList2.
- **vdata** (Variant) This data array is usually two dimensional. Each element is a type single. The two elements represent the real and imaginary parts of a complex pair.

**Note:** This structure is compatible with scripting clients who can only use variants. For alternative methods that use typed arrays, see ICalData3.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
See an Example

**C++ Syntax**
HRESULT PutStandardByString( BSTR bufferName, VARIANT vdata)

**Interface**
ICalSet2
## PutErrorTermComplex Method

### Description

**Note:** This command is replaced by `PutErrorTermComplexByString`

Puts error term data into the error-correction data buffer. Learn more about reading and writing Cal data using COM

### VB Syntax

```vb
data.putErrorTermComplex term, rcv, src, numPts, real(), imag()
```

### Variable

**Variable** (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>data</code></td>
<td>An ICalData pointer to the Calibrator object</td>
<td></td>
</tr>
<tr>
<td><code>term</code></td>
<td>(enum NAErrorTerm) - The error term to be retrieved. Choose from:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>* naErrorTerm_Directivity_Isolation</td>
</tr>
<tr>
<td></td>
<td></td>
<td>* naErrorTerm_Match</td>
</tr>
<tr>
<td></td>
<td></td>
<td>* naErrorTerm_Tracking</td>
</tr>
<tr>
<td><code>rcv</code></td>
<td>(long integer) - Receiver Port</td>
<td></td>
</tr>
<tr>
<td><code>src</code></td>
<td>(long integer) - Source Port</td>
<td></td>
</tr>
<tr>
<td><code>numPts</code></td>
<td>(long integer) - number of data points in the array</td>
<td></td>
</tr>
<tr>
<td><code>real()</code></td>
<td>(single) - array containing the real part of the calibration data. One-dimensional: the number of data points.</td>
<td></td>
</tr>
<tr>
<td><code>imag()</code></td>
<td>(single) - array containing the imaginary part of the calibration data. One-dimensional: the number of data points.</td>
<td></td>
</tr>
</tbody>
</table>

### To get this Error Term

<table>
<thead>
<tr>
<th>Error Term</th>
<th>Specify these parameters:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fwd Directivity</td>
<td><code>term</code> naET_Directivity_Isolation <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Rev Directivity</td>
<td><code>term</code> naET_Directivity_Isolation <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Fwd Isolation</td>
<td><code>term</code> naET_Directivity_Isolation <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Rev Isolation</td>
<td><code>term</code> naET_Directivity_Isolation <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Fwd Source Match</td>
<td><code>term</code> naErrorTerm_Match <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Rev Source Match</td>
<td><code>term</code> naErrorTerm_Match <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Fwd Load Match</td>
<td><code>term</code> naErrorTerm_Match <code>rcv</code> <code>src</code></td>
</tr>
<tr>
<td>Rev Load Match</td>
<td><code>term</code> naErrorTerm_Match <code>rcv</code> <code>src</code></td>
</tr>
</tbody>
</table>
### Fwd Reflection Tracking
- `naErrorTerm_Tracking`
- Parameters:
  - `1 1`

### Rev Reflection Tracking
- `naErrorTerm_Tracking`
- Parameters:
  - `2 2`

### Fwd Trans Tracking
- `naErrorTerm_Tracking`
- Parameters:
  - `2 1`

### Rev Trans Tracking
- `naErrorTerm_Tracking`
- Parameters:
  - `1 2`

### Fwd Trans Tracking
- `naErrorTerm_Tracking`
- Parameters:
  - `2 1`

**Return Type**
- Not Applicable

**Default**
- Not Applicable

**Examples**

```vbnet
Dim eData As ICalData
Set eData = chan.Calibrator
eData.putErrorTermComplex naErrorTerm_Directivity_Isolation, 1, 1, 201, rel(0), img(0)
```

**C++ Syntax**

```cpp
HRESULT putErrorTermComplex(tagNAErrorTerm ETerm, long ReceivePort, long SourcePort, long* pNumValues, float* pReal, float* pImag)
```

**Interface**
- ICalData
PutErrorTermComplex Method  Superseded

Description
This command is replaced with PutErrorTermComplexByString

Puts error term data into the Cal Set.

Learn more about Reading and Writing Cal Data

See examples of Reading and Writing Cal Set Data

VB Syntax
`data.putErrorTermComplex term, rcv, src, numPts, real(), imag()`

Variable (Type) - Description

- `data` An `ICalData2` pointer to a Cal Set object
- `term` (enum `NAErrorTerm2`) - The error term to be written. Choose from:
  - 0 - naET_Directivity
  - 1 - naET_SourceMatch
  - 2 - naET_ReflectionTracking
  - 3 - naET_TransmissionTracking
  - 4 - naET_LoadMatch
  - 5 - naET_Isolation
- `rcv` (long) - Receiver Port
- `src` (long) - Source Port
- `numPts` (long) - number of data points in the real and imaginary arrays.
- `real()` (single) - array containing the real part of the calibration data. One-dimensional: the number of data points.
- `imag()` (single) - array containing the imaginary part of the calibration data. One-dimensional: the number of data points.

Return Type
Not Applicable

Default
Not Applicable

Examples
```
Dim eData As ICalData2
Set eData = app.GetCalManager.Cal Sets.Item(1)
eData.putErrorTermComplex naET_LoadMatch, 1, 2, numpts, rel(0), img(0)
```

C++ Syntax
```
HRESULT putErrorTermComplex(tagNAErrorTerm2 ETerm, long ReceivePort, long SourcePort, long* pNumValues, float* pReal, float plmag)
```
Interface  ICalData2
### PutErrorTermComplexByString Method

**Description**  
Puts error term data into the Cal Set.  
Learn more about [Reading and Writing Cal Data](#).  
See examples of [Reading](#) and [Writing](#) Cal Set Data.

**VB Syntax**  
`ICalData3.PutErrorTermComplexByString errorName, InumPoints, real(0), imag(0)`

**Variable**  
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>ICalData3</code></td>
<td>An <code>ICalData3</code> pointer to a Cal Set object.</td>
</tr>
<tr>
<td><code>errorName</code></td>
<td>(String) The string name used to identify a particular error term in the Cal Set. An example string for port 3 directivity in a full 2 port cal might be &quot;Directivity(3,3)&quot;. To determine the string names of error terms, see <a href="#">GetErrorTermList2</a>.</td>
</tr>
<tr>
<td><code>InumPoints</code></td>
<td>(Long) The number of data points in the real and imaginary arrays.</td>
</tr>
<tr>
<td><code>real</code></td>
<td>(Single) The real component of the complex data.</td>
</tr>
<tr>
<td><code>imag</code></td>
<td>(Single) The imaginary component of the complex data.</td>
</tr>
</tbody>
</table>

**Note**: The size of the real and imaginary arrays should be the same.

**Return Value**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
[See example](#)

**C++ Syntax**  
`HRESULT PutErrorTermComplexByString( BSTR bufferName, long InumPoints, float* real, float* imag);`

**Interface**  
`ICalData3`
PutScalar Method

**Description**

Puts Scalar data in the Measurement Result buffer. The putScalar array is not processed by the analyzer; it is just displayed. Any change to the measurement state (changing the format, for example) will cause the putScalar data to be overwritten with the data processed from the raw data buffer.

**VB Syntax**

`measData.putScalar, format, numPts, data`

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>measData</td>
<td>An IArrayTransfer interface which supports the Measurement object.</td>
<td></td>
</tr>
<tr>
<td>format</td>
<td>(enum NADataFormat)</td>
<td>Format of the data. Choose from:</td>
</tr>
<tr>
<td></td>
<td>0 - naDataFormat_LinMag</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 - naDataFormat_LogMag</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 - naDataFormat_Phase</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3 - naDataFormat_Polar</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4 - naDataFormat_Smith</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5 - naDataFormat_Delay</td>
<td></td>
</tr>
<tr>
<td></td>
<td>6 - naDataFormat_Real</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7 - naDataFormat_Imaginary</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8 - naDataFormat_SWR</td>
<td></td>
</tr>
<tr>
<td></td>
<td>9 - naDataFormat_PhaseUnwrapped</td>
<td></td>
</tr>
<tr>
<td></td>
<td>10 - naDataFormat_InverseSmith</td>
<td></td>
</tr>
<tr>
<td></td>
<td>11 - naDataFormat_Kelvin</td>
<td></td>
</tr>
<tr>
<td></td>
<td>12 - naDataFormat_Fahrenheit</td>
<td></td>
</tr>
<tr>
<td></td>
<td>13 - naDataFormat_Centigrade</td>
<td></td>
</tr>
</tbody>
</table>

Learn more about Data Format.

**Note:** Smith, InverseSmith, and Polar formats are not allowed.

**numPts**

(integer) - Number of values. Usually the number of points in the trace (chan.NumberOfPoints).

**data**

(scalar) - A one-dimensional array of Scalar data matching the number of points in the current measurement.

**Return Type**

Not Applicable

**Default**

Not Applicable
Examples
Dim measData As IArrayTransfer
Set measData = app.ActiveMeasurement
measData.putScalar naDataFormat_LogMag, 201, dScalar(0)

C++ Syntax
HRESULT putScalar((tagDataFormat eFormat, long lNumValues, float* pArrayOfScalar)

Interface
IArrayTransfer

Last Modified:
1-Oct-2007   Added temperature formats
PutNAComplex Method

**Description**
Puts complex data into the specified location. This method forces the channel into Hold mode to prevent the input data from being overwritten. The data is processed and displayed.

Data put in the naRawData store will be **re-processed** whenever a change is made to the measurement attributes such as format or correction.

Data put in the naMeasResult store will be **overwritten** by any measurement attribute changes (such as moving a marker).

**Note:** This method uses NAComplex which is a user-defined data type. If you cannot or prefer not to use this data type, use the putComplex method.

**VB Syntax**
```vbnet
measData.putNAComplex location, numPts, data, [format]
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>measData</td>
<td>An IArrayTransfer interface which supports the Measurement object</td>
</tr>
<tr>
<td>location</td>
<td>(enum NADataStore) Where the Data will be put. Choose from:</td>
</tr>
<tr>
<td>numPts</td>
<td>(long integer) - Number of data points in the channel</td>
</tr>
<tr>
<td>data</td>
<td>(NAComplex) - A one-dimensional array of Complex data matching the number of points in the current measurement.</td>
</tr>
<tr>
<td>format</td>
<td>(enum NADataFormat) - Optional argument. Format of the data. If unspecified, naDataFormat_Polar is assumed. Only used when the destination store is naMeasResult or naMemoryResult.</td>
</tr>
</tbody>
</table>

- 0 - naDataFormat_LinMag
- 1 - naDataFormat_LogMag
- 2 - naDataFormat_Phase
- 3 - naDataFormat_Polar
- 4 - naDataFormat_Smith
- 5 - naDataFormat_Delay
- 6 - naDataFormat_Real
Learn more about Data Format.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vbnet
Dim measData As IArrayTransfer
Set measData = app.ActiveMeasurement

measData.putNAComplex naMemoryResult, 201, dRawComplex(0)
```

**C++ Syntax**

```cpp
HRESULT putNAComplex(tagNADataStore DataStore, long lNumValues, TsComplex* pArrayOfComplex, tagDataFormat displayFormat)
```

**Interface**
IArrayTransfer

Last Modified:

1-Oct-2007 Added temperature formats
**put_Output Method**

**Description**
Writes a TTL HI or TTL Low to output pins 3 or 4 of the Material Handler IO connector. Each pin also has a latched output which is written to with USER. With the latched (USER) outputs, the value is not applied to the associated pin until a positive edge is detected at INPUT1 (pin 2).

**VB Syntax**
```
handlerIo.put_Output (pin) = value
```

**Variable** *(Type) - Description*
- **handlerIo** *(object)* - A HandlerIO object
- **pin** *(enum as NAMatHandlerOutput)* - pin to write data to. Choose from:
  - **naOutput1** *(0)* - pin3
  - **naOutput1User** *(1)* - pin3 latched (applied to pin 3 on positive edge of Input1-pin2)
  - **naOutput2** *(2)* - pin4
  - **naOutput2User** *(3)* - pin4 latched (applied to pin 4 on positive edge of Input1-pin2)
- **value** *(Variant)* - Value to write to the selected pin. Choose from:
  - **0** - TTL LOW
  - **1** - TTL HIGH

**Return Type**
Not Applicable

**Default**
0

**Examples**
```
handlerIo.put_Output(naOutput1) = 1
```

**C++ Syntax**
```
HRESULT put_Output ( tagNAMatHandlerOutput Output, VARIANT Data );
```

**Interface**
IHWMaterialHandlerIO
**About the Aux I/O Connector**

**put_OutputVoltage Method**

**Description**
- **E836x and PNA-L:** Sets voltages on the DAC/Analog Output 1|2 of the Auxiliary IO connector.

- **PNA-X:** Sets voltage on the Power I/O connector AnalogOut1|2.

Read output voltages using `get_OutputVoltage` Method.

**VB Syntax**

```
AuxIO.put_OutputVoltage output, voltage
```

**Variable**

- **(Type) - Description**
  - **AuxIO** *(object)* - A Hardware Auxiliary Input / Output object
  - **output** *(variant)* - Number of the output DAC to write voltage to. Choose from:
    1. Output 1 (Aux I/O pin 3) and (Power I/O pin 3)
    2. Output 2 (Aux I/O pin 2) and (Power I/O pin 4)
  - **voltage** *(double)* - Voltage to write to the output DAC. Choose a voltage from -10 to 10

**Return Type**

None

**Default**

None

**Examples**

```
HWAuxIO.put_OutputVoltage 1,9 'set Analog Out1 to +9v
```

**C++ Syntax**

```
HRESULT put_OutputVoltage (VARIANT Output, double Voltage);
```

**Interface**

IHWAuxIO

---

**Last Modified:**

10-Jul-2007  Added PNA-X capability
Write-only

**put_OutputVoltageMode Method**

**Description**
This command sets the mode of the selected “Analog Out” line on the Auxiliary IO connector and Power I/O connector. The modes give the user the option to have the requested voltage applied immediately or not until the sweep is done. To read the mode on each output use get_OutputVoltageMode Method.

**VB Syntax**
`auxIo.put_OutputVoltageMode (output, mode)`

**Variable (Type) - Description**
- **auxIo (Object)** An AuxIO object
- **output** Analog Output to receive mode setting. Choose from 1 or 2
- **mode (enum NAOutputVoltageMode )**
  - **naWaitEOS** - While in this mode any voltage changes sent to the selected analog out will only get applied to the output between sweeps.
  - **naNoWait** - While in this mode any voltage changes sent to the selected analog out will occur right away without waiting until the end of a sweep, the voltage gets applied immediately.

**Return Type**
NAOutputVoltageMode

**Default**
naWaitEOS

**Examples**
`auxIo.put_OutputVoltageMode 1, naWaitEOS 'Write`

**C++ Syntax**
`HRESULT put_OutputVoltageMode(VARIANT Output, tagNAOutputVoltageMode dNewMode);`

**Interface**
IHWAuxIO

---

Last Modified:
10-Jul-2007  Added PNA-X capability
**Write-only**  

**About the Handler IO Connector**

### put_Port Method

**Description**
Writes a value to the specified port. Use the `get_Port` Method to read the settings from the "readable" ports (C, D, E).

**VB Syntax**
```
handlerIo.put_Port (port, value)
```

**Variable** *(Type) - Description*

- `handlerIo` *(object)* - A HandlerIO object
- `port` *(enum as NAMatHandlerPort)* - port to put data into. Choose from:
  - `naPortA` - (0)
  - `naPortB` - (1)
  - `naPortC` - (2)
  - `naPortD` - (3)
  - `naPortE` - (4)
  - `naPortF` - (5)
  - `naPortG` - (6)
  - `naPortH` - (7)
- `value` The number of the data bits to set. The following table shows what the `value` represents:

**Note:** When writing to port G, port C must be set to output mode  
When writing to port H, both port C and port D must be set to output mode. Use Port Mode Property

<table>
<thead>
<tr>
<th>Port</th>
<th>Max allowable &lt;num&gt;</th>
<th>MSB..............................LSB</th>
<th>Value Meanings</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>255</td>
<td>A7...A0</td>
<td>Write-only</td>
</tr>
<tr>
<td>B</td>
<td>255</td>
<td>B7...B0</td>
<td>Write-only</td>
</tr>
<tr>
<td>C</td>
<td>15</td>
<td>C3...C0</td>
<td>Read-Write</td>
</tr>
<tr>
<td>D</td>
<td>15</td>
<td>D3...D0</td>
<td>Read-Write</td>
</tr>
<tr>
<td>E</td>
<td>255</td>
<td>D3...D0 + C3...C0</td>
<td>Read-Write</td>
</tr>
<tr>
<td>F</td>
<td>65535</td>
<td>B7...B0 + A7...A0</td>
<td>Write-only</td>
</tr>
</tbody>
</table>
Return Type: Not Applicable
Default: Not Applicable
Examples: `handlerIo.put_Port(naPortB, 1)`

C++ Syntax: `HRESULT put_Port ( tagNAMatHandlerPort Port, VARIANT Data );`

Interface: IHWMaterialHandlerIO

Last Modified: 21-Mar-2008  Fixed syntax
# put_PortCData Method

**Description**  
Writes a 4-bit value to Port C on the Aux I/O connector (pins 22-25) and the Material Handler IO (pins 21-24 Anritsu) - (pins 22-25 Avantest).

**Note:** These lines are connected to both the Handler IO and Aux IO in the PNA. Therefore, this command will affect both of these connectors in the same way.

**VB Syntax**  
```vbnet
AuxIO.put_PortCData num
```

**Variable**  
- **(Type) - Description**
  - `AuxIO` *(object)* - A Hardware Auxiliary Input / Output object
  - `num` *(variant)* - 4 bit binary value. Choose from 0-15

**Return Type**  
None

**Default**  
None

**Examples**  
```vbnet
HWAuxIO.put_PortCData 15  'If Positive Logic, Port C lines C0, C1, C2, C3 go High. If Negative Logic, they go Low.
```

**C++ Syntax**  
```c++
HRESULT put_PortCData( VARIANT Data );
```

**Interface**  
IHWAuxIO
PutDataScalar Method

**Description**

Puts formatted variant scalar data into the measurement result buffer. The data will be immediately processed and displayed. Subsequent changes to the measurement state will be reflected on the display.

Always precede this command by setting the format on the measurement to be consistent with the format of the data being sent to the analyzer. In this way, the display annotation will be correct.

Execution of this command does not change the display format.

**VB Syntax**

```vb
meas.putDataScalar format, data
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>meas</td>
<td>A measurement (object)</td>
</tr>
<tr>
<td>format</td>
<td>(enum NADeformFormat) Format of the data. This value is presently ignored by the PNA. Data is always presented in the current format. Choose from: 0 - naDataFormat_LinMag 1 - naDataFormat_LogMag 2 - naDataFormat_Phase 3 - naDataFormat_Polar 4 - naDataFormat_Smith 5 - naDataFormat_Delay 6 - naDataFormat_Real 7 - naDataFormat_Imaginary 8 - naDataFormat_SWR 9 - naDataFormat_PhaseUnwrapped 10 - naDataFormat_InverseSmith 11 - naDataFormat_Kelvin 12 - naDataFormat_Fahrenheit 13 - naDataFormat_Centigrade</td>
</tr>
</tbody>
</table>

**Notes:**

- The `getData` (variant) method includes a "format" argument, which allows scalar (one-dimensional) data. To put data back into the "raw" data buffer using this (putDataComplex) method, specify **Polar** format when using the `getData` method.

- **Phase** format accepts data in radians (not degrees) and displays in degrees. To convert to degrees: radians \* (57.29577951308233) = degrees. The `getData` method returns degrees if the request is for phase data.
data (variant) - A 1-dimension array of single precision floating point numbers.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
' Put 201 points worth of scalar data into the measurement
' 200 is max index, so 0 to 200 is 201 points
Dim data(200) ' array of 201 (scalar) data points
' Fill the array
For i = 0 to 200
    data(i) = i/200;
Next
app.ActiveMeasurement.putDataScalar 0, data

**C++ Syntax**
HRESULT putDataScalar(tagNADataStore DataStore, VARIANT scalarArray)

**Interface**
IMeasurement

Last Modified:
1-Oct-2007  Added temperature formats
PutShortcut Method

**Description**
Defines a Macro (shortcut) file in the analyzer. This command links a file name and path to the Macro file. The file must be put in the PNA at the location indicated by this command.

**VB Syntax**
```
app.PutShortcut index,title,path, arguments
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>index</code></td>
<td>(long) - Number of the macro to be stored in the analyzer. If the index number already exists, the existing macro is replaced with the new macro.</td>
</tr>
<tr>
<td><code>title</code></td>
<td>(string) - The name to be assigned to the macro</td>
</tr>
<tr>
<td><code>path</code></td>
<td>(string) - Full path, file name, and extension of the existing macro &quot;executable&quot; file.</td>
</tr>
<tr>
<td><code>arguments</code></td>
<td>(string) - Arguments that may be required for the specified macro to run.</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
app.PutShortcut 1, "Test", "C:/Automation/MyTest.vbs", ""
```

**C++ Syntax**
```
HRESULT PutShortcut(long Number, BSTR title, BSTR pathname,BSTR arguments)
```

**Interface**
IApplication

---

Last Modified:
25-Jun-2009   Added arguments
### putSourcePowerCalDataEx Method

<table>
<thead>
<tr>
<th>Description</th>
<th><strong>Note:</strong> This method replaces putSourcePowerCalData Method</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inputs source power calibration data (as variant data type) to this channel for a specific source port.</td>
</tr>
<tr>
<td></td>
<td>The effect from this command on the channel is immediate. Do NOT send ApplyPowerCorrectionValuesEX after this command as it may invalidate the uploaded data.</td>
</tr>
<tr>
<td></td>
<td>If the channel is sweeping the source backwards, then the first data point is the highest frequency value; the last data point is the lowest. Use the Get X-Axis Values command to return the X-axis values in the displayed order.</td>
</tr>
<tr>
<td></td>
<td>The calibration is not valid if the current number of points on the channel is not equal to the number of values that were input.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> This method sends variant data which is less efficient than methods available on the ISourcePowerCalData interface.</td>
</tr>
</tbody>
</table>

### VB Syntax

```vbnet
chan.putSourcePowerCalDataEx buffer, srcPort, data
```

<table>
<thead>
<tr>
<th>Variable</th>
<th><strong>(Type)</strong> - <strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>chan</td>
<td>(object) – A Channel object</td>
</tr>
<tr>
<td>buffer</td>
<td>(enum NASourcePowerCalBuffer) - The source power cal data buffer to write to.</td>
</tr>
<tr>
<td></td>
<td>0 - naCorrectionValues This is the only data buffer currently available.</td>
</tr>
<tr>
<td>srcPort</td>
<td>(long integer) – The source port for which calibration data is being requested.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port.</td>
</tr>
<tr>
<td>data</td>
<td>(variant) – Array of source power cal data being input.</td>
</tr>
</tbody>
</table>

### Return Type
None

### Default
Not Applicable

### Examples

```vbnet
chan.putSourcePowerCalDataEx naCorrectionValues, 1, varData
```

### C++ Syntax

```csharp
HRESULT putSourcePowerCalDataEx(tagNASourcePowerCalBuffer bufSelect, long sourcePort, VARIANT varData);
```

### Interface
IChannel4
putSourcePowerCalDataScalarEx Method

Description

Note: This method replaces putSourcePowerCalDataScalar Method

Inputs source power calibration data (as scalar values) to this channel for a specific source port.

The effect from this command on the channel is immediate. Do NOT send ApplyPowerCorrectionValuesEX after this command as it may invalidate the uploaded data.

If the channel is sweeping the source backwards, then the first data point is the highest frequency value; the last data point is the lowest. Use the Get X-Axis Values2 command to return the X-axis values in the displayed order.

VB Syntax

```vbnet
chanData.putSourcePowerCalDataScalarEx buffer, srcPort, numValues, data
```

Variable (Type) - Description

- **chanData** (interface) – An ISourcePowerCalData2 interface on the Channel (object)
- **buffer** (enum NASourcePowerCalBuffer) - The source power cal data buffer to write to.
  - 0 - naCorrectionValues This is the only buffer currently available.
- **srcPort** (long integer) – The source port for which calibration data is being input.
  
  Note: If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use chan.getPortNumber to translate the string into a port number. To learn more see Remotely Specifying a Source Port.

- **numValues** (long integer) – Number of data values being input.
  
  Note: If this does not equal the current number of points on the channel, the calibration will not be valid.

- **data** (single) – Array of source power cal data being input.

Return Type

None

Default

Not Applicable

Examples

```vbnet
Dim chanData As ISourcePowerCalData2
Set chanData = app.ActiveChannel
chanData.putSourcePowerCalDataScalarEx naCorrectionValues, 1, 201, scalarCalValues(0)
```

C++ Syntax

```c++
HRESULT putSourcePowerCalDataScalarEx(tagNASourcePowerCalBuffer bufSelect, long sourcePort, long numValues, float *pData);
```
Interface ISourcePowerCalData2

Last Modified:

25-Oct-2010  Added immediate note
24-Apr-2008  Added string name note
PutStandard Method Superseded

Description

This command is replaced with PutStandardByString

Puts standard acquisition data into the Cal Set.

Learn more about Reading and Writing Cal Data

See examples of Reading and Writing Cal Set Data.

VB Syntax

CalSet.putStandard class, rcv, src, data

Variable (Type) - Description

CalSet (object) - A Cal Set object

class (enum NACalClass) Standard. Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

SOLT Standards

1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

TRL Standards

1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
5 - naTRL_Isolation
rcv  (long) - Receiver Port

src  (long) - Source Port

data  (variant) Error term data in a two-dimensional array (0:1, 0:numpts-1). The data must be complex pairs.

**Note:** See also Put Standard Complex on the ICalData2 interface to avoid using the variant data type.

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  See an Example

**C++ Syntax**  HRESULT putStandard(tagNACalClass stdclass, long ReceivePort, long SourcePort, VARIANT varData)

**Interface**  ICalSet
PutStandardByString

**Description**  
Puts standard acquisition data into the Cal Set.  
Learn more about [Reading and Writing Cal Data](#).  
See examples of [Reading](#) and [Writing](#) Cal Set Data.

**VB Syntax**  
```vb
PutStandardByString(stdName, vdata)
```

**Variable**  
*(Type) - Description*

- **stdName** *(String)*  
The string used to identify a particular standard in the Cal Set. An example string requesting the data for the Load standard in a full 2 port cal might be “S11C(3,3)”.

- **vdata** *(Variant)*  
The variant containing a safearray of variants. This data is usually two dimensional.

**Note:** The vardata array is a safearray of variants wrapped in a variant. This structure is compatible with scripting clients who can only use variants. For alternative methods that used typed arrays, see ICalData3.

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
[See an Example](#)

**C++ Syntax**  
```cpp
HRESULT PutStandardByString( BSTR bufferName, VARIANT vdata);
```

**Interface**  
ICalSet2
About Cal Sets

PutStandardComplex Method  Superseded

Description
This command is replaced with PutStandardComplexByString
Puts standards acquisition data into the Cal Set.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data

VB Syntax
ICalData2.putStandardComplex class, rcv, src, numPts,real(),imag()

Variable  (Type) - Description
ICalData2  An ICalData2 pointer to the Cal Set object

class  (enum NACalClass) Standard. Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

SOLT Standards
1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

TRL Standards
1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
5 - naTRL_Isolation
$rcv$ (long) - Receiver Port

$src$ (long) - Source Port

$numPts$ (long) - The number of data points in the real and imaginary arrays.

$real()$ (single) - one-dimensional array containing the real part of the acquisition data. (0:points-1)

$imag()$ (single) - one-dimensional array containing the imaginary part of the acquisition data. (0:points-1)

**Return Type**: Not Applicable

**Default**: Not Applicable

**Examples**

Dim sdata As ICalData2
Set sdata = calmanager.CreateCal Set( 1 )
sdata.putStandardComplex naSOLT_Open, 1, 1, numpts, rel(0), img(0)

**C++ Syntax**

HRESULT putStandardComplex(tagNACalClass stdclass, long ReceivePort, long SourcePort, long lNumValues, float* pReal, float* pImag)

**Interface**

ICalData2
PutStandardComplexByString

**Description**
Puts standard acquisition data into the Cal Set.
Learn more about Reading and Writing Cal Data
See examples of Reading and Writing Cal Set Data.

**VB Syntax**

```
ICalData3.PutStandardComplexByString(stdName, lnumPoints, real(o), imag(0))
```

**Variable**

- **ICalData3**
  - An ICalData3 pointer to a Cal Set object.
- **stdName**
  - (String) The string used to identify a particular standard in the Cal Set. An example string requesting the data for the Load standard in a full 2 port cal might be "S11C(3,3)".
- **lnumpoints**
  - (long) - The number of data points in the real and imaginary arrays.
- **real**
  - (Single) The real component of the complex data.
- **imag**
  - (Single) The imaginary component of the complex data.

**Return Value**
Single

**Default**
Not Applicable

**Examples**
See an Example

**C++ Syntax**

```
HRESULT PutStandardComplexByString(BSTR bufferName, long InumPoints, float* real, float* imag);
```

**Interface**
ICalData3
### Quit Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Terminates the Network Analyzer application.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.Quit</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.Quit</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT Quit()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IApplication</code></td>
</tr>
<tr>
<td><strong>Remarks</strong></td>
<td>Under the rules of COM, the server should not exit until all references to it have been released. This method is a brute force way of terminating the application. Be sure to release all references (or terminate the client program) before attempting to restart the Network Analyzer application. An alternate approach to terminating the application is to make the application invisible <em>(app.Visible = False)</em> and release all references. The server will shutdown.</td>
</tr>
</tbody>
</table>
Read-only

ReadData Method

Description
Reads a 13-bit data word from the specified address. Data is read using the AD0 through AD12 lines of the external test set connector. The instrument generates the appropriate timing signals. It automatically controls timing signals LDS, LAS and RLW to strobe the address, and then read the data, from the external test set. See the timing diagram for Address and Data I/O read.

VB Syntax
value = ExtIO.ReadData (address)

Variable (Type) - Description
value (variant) - Variable to store the returned data
ExtIO (object) - An ExternalTestSetIO object
address (variant) - address to read data from.

Return Type
Variant

Default
Not Applicable

Examples
value = ExtIO.ReadData (15)

C++ Syntax
HRESULT ReadData (VARIANT Address, VARIANT* Data);

Interface
IHWExternaTestSetIO
ReadRaw Method

Description
Reads a 16-bit value from the external test set. The 16-bit value is comprised of lines AD0 - AD12, Sweep Holdoff In and Interrupt In (inverted).

When this command is used the analyzer does NOT generate the appropriate timing signals; it simply reads the lines. The user needs to first use the WriteRaw method to do the initial setup. The RLW line (pin25) must be set to the appropriate level in order to read the test set connected.

Below is the format of data that is read with ReadRaw:

<table>
<thead>
<tr>
<th>Pin</th>
<th>Bit</th>
<th>Signal name</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>0</td>
<td>AD0*</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>AD1*</td>
</tr>
<tr>
<td>11</td>
<td>2</td>
<td>AD2*</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>AD3*</td>
</tr>
<tr>
<td>9</td>
<td>4</td>
<td>AD4*</td>
</tr>
<tr>
<td>21</td>
<td>5</td>
<td>AD5*</td>
</tr>
<tr>
<td>20</td>
<td>6</td>
<td>AD6*</td>
</tr>
<tr>
<td>19</td>
<td>7</td>
<td>AD7*</td>
</tr>
<tr>
<td>6</td>
<td>8</td>
<td>AD8*</td>
</tr>
<tr>
<td>5</td>
<td>9</td>
<td>AD9*</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>AD10*</td>
</tr>
<tr>
<td>17</td>
<td>11</td>
<td>AD11*</td>
</tr>
<tr>
<td>3</td>
<td>12</td>
<td>AD12*</td>
</tr>
<tr>
<td>2</td>
<td>13</td>
<td>Sweep Holdoff In</td>
</tr>
<tr>
<td>13</td>
<td>14</td>
<td>Interrupt In (inverted internally)</td>
</tr>
<tr>
<td>na</td>
<td>15</td>
<td>Always Zero, grounded internally</td>
</tr>
</tbody>
</table>
*These lines are dependent on the state of RLW (pin25).
Writing a 0(low) to RLW will set lines AD0-AD12 to write mode.
Writing a 1(high) to RLW will set lines AD0-AD12 to read mode.

**VB Syntax**

```
value = ExtIO.ReadRaw (address)
```

**Variable (Type) - Description**

- **value** (variant) - Variable to store the returned data
- **ExtIO** (object) - An External IO object
- **address** (variant) - Address to read data from

**Return Type**

Real

**Default**

Not Applicable

**Examples**

```
value = ExtIO.ReadRaw (address)
```

**C++ Syntax**

```
HRESULT ReadRaw( VARIANT* Input );
```

**Interface**

IHWExternalTestSetIO
### ReCalculate Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Repeats the last calculation that was performed, including all ON (state) segments in segment table.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>conv.ReCalculate()</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>conv</code></td>
<td>A Converter Object</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mxr.ReCalculate()</code></td>
</tr>
<tr>
<td></td>
<td>See example program</td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT ReCalculate()</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IConverter5</td>
</tr>
</tbody>
</table>

Last Modified:

26-Oct-2010   New command (A.09.33)
Recall Method

**Description**
Recalls a measurement state, calibration state, or both, from the hard drive into the analyzer.

Use `app.Save()` to save files.

**VB Syntax**
`app.Recall(filename.ext)`

**Variable**
- **app**: An `Application` object
- **filename.ext**: Full path, file name, and extension, of the file.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"
Use one of the following extensions:

- `.sta` - Instrument State
- `.cal` - Calibration file
- `.cst` - Both Instrument State and Calibration reference
- `.cti` - Citifile (data will always be formatted. See Recalling Citifiles Using the PNA)
- `.csa` - Instrument state and calibration data (not a reference pointer).

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
`app.Recall("C:/Program Files/Agilent/Network Analyzer/Documents/MyState.cst")` 'Recalls "mystate.cst" from the specified folder

**C++ Syntax**
HRESULT Recall(BSTR bstrFile)

**Interface**
IApplication
### Recall Kits Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Recalls the calibration kits definitions that were stored with the SaveKits command.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.RecallKits</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An <a href="#">Application</a> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.RecallKits</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT RecallKits()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IApplication</td>
</tr>
</tbody>
</table>
**Remove Method**

**Description**
Removes an item from a collection of objects.

**VB Syntax**
`Object.Remove item`

**Variable**
*(Type)* - Description

- **Object**
  Any of the following *(objects)*
  - `CalFactorSegments` collection
  - `CalFactorSegmentsPMAR` Collection
  - `Cal Sets` collection
  - `Channels Collection`
  - `ExternalDevices` Collection
  - `GuidedCalibrationPowerSensors` Collection
  - `Measurements` collection
  - `NAWindows` collection
  - `PowerLossSegments` collection
  - `PowerLossSegmentsPMAR_Collection`
  - `Segments` collection

**Note:** `Segments`, `CalFactorSegments`, and `PowerLossSegments` have an OPTIONAL argument `[size]` referring to the number of segments to remove, starting with the `item` parameter.

**Note:** `Segments` - When ALL segments are deleted, `SweepType` is automatically set to Linear because there are no segments to sweep.

- **item** *(variant)* - Collection Item number to be removed.
  **Note:** The `ExternalDevices` Collection requires that you specify `item` as the string name of the device. For example:
  `extDevices.Remove ('mySource')`

**Return Type**
Not Applicable

**Default**
Not Applicable
Examples

Measurements.Remove 3 'Removes the third measurement in the collection
segments.Remove 2,20 'Removes 20 segments (2 - 21)

C++ Syntax

HRESULT Remove(VARIANT index); //Measurements
HRESULT Remove(VARIANT index); //Cal Sets
HRESULT Remove(long windowNumber); //NAWindows
HRESULT Remove(VARIANT index, long size); //Segments
HRESULT Remove(VARIANT index, long size); //CalFactorSegments(PMAR)
HRESULT Remove(VARIANT index, long size); //PowerLossSegments(PMAR)
HRESULT Remove(BSTR name) //ExternalDevices
HRESULT Remove(VARIANT index) // Channels - specify collections index, not the channel number.
HRESULT Remove(VARIANT index); //GuidedCalibrationPowerSensors

Interface

All listed above

Last Modified:

8-Feb-2011 Added GuidedCalibrationPowerSensors (9.33)
16-Sep-2010 Added channels (9.30)
31-Jul-2009 Added External Devices (9.0)
RemoveChannelNumber Method

**Description**
Deletes a channel by specifying the channel number.
Use [Remove Method](#) to delete a channel by specifying the index in the channels collection.

**VB Syntax**

```vbnet
chans.RemoveChannelNumber(chan)
```

**Variable**
- `(Type) - Description`
  - `chans` A [Channels](#) (collection)
  - `chan` The channel number to delete.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vbnet
chan.RemoveChannelNumber(2)
```

**C++ Syntax**

```cpp
HRESULT RemoveChannelNumber(VARIANT channelNumber)
```

**Interface**
IChannels3

---

Last Modified:

16-Sep-2010  MX New topic
Removal Method

**Description**
Removes a name-value pair from the Cal Set. Send the Save (CalSet) Method to save the edited CalSet to the PNA.

**See Also**
EnumerateItems Method
Item Property (Learn about name-value pairs)

**VB Syntax**
```vbnet
CalSet.RemoveItem (name)
```

**Variable**
- **Type** - Description
- **CalSet** (object) - A CalSet object
- **name** (String) Name of the item.

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
See example

**C++ Syntax**
```c++
HRESULT RemoveItem (VARIANT* itemNames);
```

**Interface**
ICalSet6

Last Modified:
24-Sep-2010  MX New topic
RemoveLibrary Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Removes an imported Equation Editor DLL from the PNA.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>equation.RemoveLibrary location</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>equation</code></td>
<td>A <code>MeasurementEquation</code> object</td>
</tr>
<tr>
<td><code>location</code></td>
<td><em>(string)</em> – Full path and filename of the <em>.dll to be removed.</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>equation.RemoveLibrary &quot;C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll&quot;</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT RemoveLibrary( BSTR filename);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IMeasurementEquation2</code></td>
</tr>
</tbody>
</table>

Last Modified:

10-Jan-2011  New topic
### Reset Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Removes all existing windows and measurements from the application. (Unlike <code>Preset</code>, does not create a new measurement.)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.Reset</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.Reset</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT Reset()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IApplication</td>
</tr>
</tbody>
</table>
ResetLOFrequency Method

**Description**
Resets the LO Delta Frequency to 0 (zero) Hz.

**VB Syntax**
`obj.ResetLOFrequency`

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>obj</code></td>
<td>An EmbeddedLO (object) or</td>
</tr>
<tr>
<td></td>
<td>A ConverterEmbeddedLO (object)</td>
</tr>
</tbody>
</table>

**Default**
Not Applicable

**Examples**
`embedLO.ResetLOFrequency` 'write

**C++ Syntax**
`HRESULT ResetLOFrequency();`

**Interface**
IEmbededLO

---

Last Modified:
- 12-Aug-2009  Added ConvEmbedLO object
- 18-Apr-2007  MX New topic
# ResetTheme Method

**Description**
Resets the current theme to the default PNA colors.

**VB Syntax**
```vbnet
colors.ResetTheme()
```

**Variable**
- **Type** - Description
- **colors** - A `ComColors` object

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
colors.ResetTheme()
```

**C++ Syntax**
```cpp
HRESULT ResetTheme();
```

**Interface**
IComColors

---

Last Modified:
7-Aug-2009    MX New topic
ResetTuningParameters Method

Description
Resets the tuning parameters to their default values.

VB Syntax
    obj.ResetTuningParameters

Variable
(Type) - Description
obj   An EmbeddedLO (object) or
      A ConverterEmbeddedLO (object)

Default
Not Applicable

Examples
    embedLO.ResetTuningParameters 'write

C++ Syntax
    HRESULT ResetTuningParameters();

Interface
    IEmbeddedLO

Last Modified:
12-Aug-2009   Added ConvEmbedLO object
18-Apr-2007   MX New topic
**About Modifying Cal Kits**

### RestoreCalKitDefaults Method

**Description**
Restores the original properties of the specified Cal Kit, overwriting the last definition with the factory defaults.

**NOTE**: ONLY works with PNA releases 1.0 through 1.6.

**VB Syntax**

```vbnet
app.RestoreCalKitDefaults calKit
```

**Variable (Type) - Description**

- **app** An Application (object)
- **calKit** (enum NACalKit) - Calibration Kit to restore. Choose from:
  1. naCalKit_85032F_N50
  2. naCalKit_85033E_3_5
  3. naCalKit_85032B_N50
  4. naCalKit_85033D_3_5
  5. naCalKit_85038A_7_16
  6. naCalKit_85052C_3_5_TRL
  7. naCalKit_User7
  8. naCalKit_User8
  9. naCalKit_User9
  10. naCalKit_User10

**Return Type** Not Applicable

**Default** Not Applicable

**Examples**

```vbnet
app.RestoreCalKitDefaults naCalKit_MechKit10
```

**C++ Syntax**

```
HRESULT RestoreCalKitDefaults(tagNACalKit kit)
```

**Interface** IApplication
## RestoreCalKitDefaultsAll Method

**Description**
Restores the original properties of ALL of the Cal Kits, overwriting the last definitions with the factory defaults.

**NOTE:** ONLY works with PNA releases 1.0 through 1.6.

### VB Syntax
`app.RestoreCalKitDefaultsAll`

### Variables

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
</tbody>
</table>

### Return Type
Not Applicable

### Default
Not Applicable

### Examples
`app.RestoreCalKitDefaultsAll`

### C++ Syntax
`HRESULT RestoreCalKitDefaultsAll()`

### Interface
IAplication
About PNA Preferences

**RestoreDefaults Method**

**Description**
Resets the PNA preferences to their factory default settings.

**VB Syntax**
`pref.RestoreDefaults`

**Variable** *(Type)* - Description
- `pref` A `Preferences (object)`

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
`pref.RestoreDefaults`

**C++ Syntax**
`HRESULT RestoreDefaults()`

**Interface**
IPreferences9

---

Last Modified:

24-Apr-2008 MX New topic
### Resume Method

**Description**  
Resumes the trigger mode of all channels that was in effect before sending the `channels.Hold` method. `Channels.Hold` must be sent before `channels.Resume`, using the same instance of the `Channels` object.

**VB Syntax**  
`chsns.Resume`

**Variable**  
*(Type)*  - Description

- `chsns` - A Channel collection *(object)*

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`chsns.Resume`

**C++ Syntax**  
`HRESULT Resume();`

**Interface**  
IChannels2
Save Method

**Description**
Saves the appropriate content to the hard drive depending on the extension that is provided.

Some saved files can be recalled using `app.Recall`, depending on the content.

**VB Syntax**
```
app.Save(filename.ext)
```

**Variable**

- **`app`** - *Type* - *Description*
  
  `app` - An Application (object)

- **`filename.ext`** - *Type* - *Description*
  
  `filename.ext` - Full path, file name, and extension of the file.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

Use one of the following extensions:

- **.cst** - Saves both Instrument State and Cal Set reference - Recalls a calibrated measurement. (Recallable)
- **.sta** - Saves Instrument State only - recalls the instrument state without calibration. (Recallable)
- **.cal** - Calibration file – saves the active Cal Sets currently in use by any channel. Use this mode for archival purposes only. All Cal Sets are saved to a Cal Set data file. This mode provides a method of safeguarding calibration data. This data can be restored to the list of Cal Sets available in the instrument. (Recallable)
- **.csa** - Saves both instrument state AND actual calibration data, not a reference pointer to the Cal Set.
- **.prn** - Saves active trace in comma-separated format (not recallable)
- **.bmp** - Saves a Bitmap of the screen (not recallable)
- **.s1p** - Saves 1-port measurement data
- **.s2p** - Saves 2-port measurement data
- **.s3p** - Saves 3-port measurement data
- **.s4p** - Saves 4-port measurement data

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
app.Save("C:/Program Files/Agilent/Network Analyzer/Documents/Newfolder/MyState.cst") 'Saves "mystate.cst" to the specified folder
```
**C++ Syntax**  HRESULT Save(BSTR bstrFile)

**Interface**  IApplication

Last Modified:

26-Jun-2007  Corrected example
Save Method

**Description**  Saves the current Cal Set to disk. This is the recommended method for saving a Cal Set.

Learn more about reading and writing Cal data using COM

**VB Syntax**  
`CalSet.Save`

**Variable**  
*CalSet* (object) - A *CalSet* object

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  
`myCalSet.Save`

See [Copy Method](#) for an example application of this command.

**C++ Syntax**  
`HRESULT Save();`

**Interface**  ICalSet
SaveCalSets Method  Superseded

**Description**
This command is replaced by `ICalSet::Save` which saves the data for only the current Cal Set to the disk.

Writes new or changed Cal Sets to disk. All Cal Sets are saved in a single file. This file is updated at the following times:

- When a Cal Set has been deleted.
- When a calibration has been performed through the front panel interface.
- When this method is called.
- When `ICalSet::Save` is called.

Learn more about reading and writing Cal data using COM

**VB Syntax**
```vbnet
object.SaveCalSets
```

**Variable**
- **(Type) - Description**
  - `object` (object) - A CalManager object or a Calibrator object

**Return Type**
None

**Default**
Not Applicable

**Example**
```vbnet
calMgr.SaveCalSets
```

**C++ Syntax**
```cpp
HRESULT SaveCalSets();
```

**Interface**
`ICalManager`
`ICalibrator`
SaveCitiDataData Method - Superseded

<table>
<thead>
<tr>
<th>Description</th>
<th>This command is replaced with <a href="#">SaveData Method</a> Saves UNFORMATTED trace data to .cti file. <a href="#">Learn more about citifiles</a>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>app.SaveCitiDataData(filename.cti)</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>app</code></td>
<td>An <a href="#">Application</a> (object)</td>
</tr>
<tr>
<td><code>filename.cti</code></td>
<td>(string) - Full path, file name, and .cti extension of the file. Files are typically stored in &quot;C:/Program Files/Agilent/Network Analyzer/Documents&quot;.</td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>app.SaveCitiDataData(&quot;C:/Program Files/Agilent/Network Analyzer/Documents/myDDCitifile.cti&quot;)</code> 'Saves &quot;myDDCitifile.cti&quot; to the specified folder</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT SaveCitiDataData (BSTR bstrFile)</td>
</tr>
<tr>
<td>Interface</td>
<td>IApplication5</td>
</tr>
</tbody>
</table>
### Description
This command is replaced with [SaveData Method](#).
Saves FORMATTED trace data to .cti file. [Learn more about citifiles](#).

### VB Syntax
```vbnet
app.SaveCitiFormattedData(filename.cti)
```

### Variable *(Type) - Description*
<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
<td></td>
</tr>
<tr>
<td>filename.cti</td>
<td>(string)</td>
<td>Full path, file name, and .cti extension of the file. Files are typically stored in &quot;C:/Program Files/Agilent/Network Analyzer/Documents&quot;</td>
</tr>
</tbody>
</table>

### Return Type
Not Applicable

### Default
Not Applicable

### Examples
```vbnet
app.SaveCitiFormattedData("C:/Program Files/Agilent/Network Analyzer/Documents/Newfolder/myFDCitifile.cti") 'Saves "myFDCitifile.cti" to the specified folder
```

### C++ Syntax
```cpp
HRESULT SaveCitiFormattedData (BSTR bstrFile)
```

### Interface
IApplication5
### SaveData Method

**Description**
Stores trace data to the following file types: *.prn, *.cti, *.csv, *.mdf

To save snp files, use `WriteSnpFileWithSpecifiedPorts`.

To save instrument state and calibration files, use `Save`.

This command replaces the following:
- `SaveCitiDataData Method`
- `SaveCitiFormattedData Method`
- `CitiContents Property`
- `CitiFormat Property`

Some saved files can be recalled using `app.Recall`, depending on the content.

**VB Syntax**

```vb
app.SaveData filename, type, scope, format, selector
```

**Variable (Type) - Description**

- `app` An `Application` (object)
- `filename` (string) - Full path, file name, and extension of the file.

Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents"

### Parameters

<table>
<thead>
<tr>
<th>Type of file to save</th>
<th>&lt;type&gt; (String)</th>
<th>&lt;scope&gt; (String)</th>
<th>&lt;format&gt; (String)</th>
<th>&lt;selector&gt; (Integer)</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.prn</td>
<td>&quot;PRN Trace Data&quot;</td>
<td>&quot;Trace&quot;</td>
<td>&quot;Displayed&quot;</td>
<td>Measurement number</td>
</tr>
<tr>
<td>*.cti (unformatted)</td>
<td>&quot;Citifile Data Data&quot;</td>
<td>&quot;Trace&quot; or &quot;Auto&quot;</td>
<td>&quot;RI&quot;</td>
<td>Measurement number</td>
</tr>
<tr>
<td>*.cti (unformatted)</td>
<td>&quot;Citifile Data Data&quot;</td>
<td>&quot;Displayed&quot;</td>
<td>&quot;RI&quot;</td>
<td>1</td>
</tr>
</tbody>
</table>

Example: `app.SaveData "myData.prn","PRN Trace Data","Trace","Displayed",2`

Example: `app.SaveData "myData.cti","Citifile Data Data","AUTO","RI",3`

**Example**

```vb
app.SaveData "myData.cti","Citifile Data Data","AUTO","RI",3
```
**Example:** `app.SaveData "myData.cti","Citifile Formatted Data","AUTO","RI",3`  

<table>
<thead>
<tr>
<th><em>cti</em> (formatted)</th>
<th>&quot;Citifile Formatted Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.cti","Citifile Formatted Data","AUTO","MA",3`  

<table>
<thead>
<tr>
<th><em>cti</em> (formatted)</th>
<th>&quot;Citifile Formatted Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot; or &quot;Displayed&quot;</th>
<th>-1</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.cti","Citifile Formatted Data","DISPLAYED","MA",-1`  

<table>
<thead>
<tr>
<th><em>csv</em></th>
<th>&quot;CSV Formatted Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.csv","CSV Formatted Data","Trace","DB",3`  

<table>
<thead>
<tr>
<th><em>csv</em></th>
<th>&quot;CSV Formatted Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>-1</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.csv","CSV Formatted Data","displayed","RI",-1`  

<table>
<thead>
<tr>
<th><em>mdf</em></th>
<th>&quot;MDIF Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;Displayed&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.mdf","MDIF Data","trace","displayed",1`  

<table>
<thead>
<tr>
<th><em>mdf</em></th>
<th>&quot;MDIF Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot;</th>
<th>-1</th>
</tr>
</thead>
</table>

**Example:** `app.SaveData "myData.mdf","MDIF Data","displayed","displayed",-1`  

**Notes (for above file types)**  
Use `meas.Number` to read the measurement number of a trace.

**Scope:**  
"Trace" - specified measurement number only.
"Displayed" - all displayed measurements.
"Auto" - for all Standard Meas Class (S-parameter) channels:

- When correction is OFF, saves the specified trace
- When correction is ON, saves all corrected parameters associated with the
calibrated ports in the Cal Set.

"Auto" - for all other channels:

- When correction is OFF or ON, saves the specified trace

The following parameter combinations save *.csv files in specific formats for GCA and Swept IMD classes:

<table>
<thead>
<tr>
<th>Parameters</th>
<th>&lt;type&gt;</th>
<th>&lt;scope&gt;</th>
<th>&lt;format&gt;</th>
<th>&lt;selector&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(String)</td>
<td>(String)</td>
<td>(String)</td>
<td>(Integer)</td>
</tr>
</tbody>
</table>

**GCA channels ONLY:**

- "GCA Sweep Data"
- "Auto"
- "DB"

Example: app.SaveData "myData.csv","GCA Sweep Data","Auto","db",1

**Swept IMD channels ONLY:**

- "IMD Sweep Data"
- "Auto"
- "DB"

Example: app.SaveData "myData.csv","IMD Sweep Data","Auto","db",1

**Return Type** Not Applicable

**Default** Not Applicable

**C++ Syntax**

```c++
HRESULT SaveData(BSTR File, BSTR Type, BSTR Scope, BSTR Format, Long selector);
```

**Interface** IApplication18

Last Modified: 9-Apr-2010  MX New topic
SaveENRFile Method

Description
Saves an ENR table to disk.

VB Syntax
enr.SaveENRFile (filename)

Variable
- **enr**: An ENRFile (object)
- **filename**: (String) - Absolute path and filename of the ENR file.

Return Type
Not Applicable

Default
Not Applicable

Examples
See example program

C++ Syntax
HRESULT SaveENRFile(BSTR filename);

Interface
IENRFile

Last Modified:
2-Aug-2007  MX New topic
SaveFile Method

Description
Saves the mixer/converter test setup to a mixer attributes (.mxr) file.

VB Syntax
obj.SaveFile (filename)

Variable (Type) - Description

obj
A Mixer Interface pointer to the Measurement (object)
Or
A Converter Object

filename
(String) Full path, file name, and .mxr extension of the file.
Files are typically stored in "C:/Program Files/Agilent/Network Analyzer/Documents".

Return Type
String

Default
Not Applicable

Examples
mixer.SaveFile ("C:/Program Files/Agilent/Network Analyzer/Documents/myMixer.mxr")

C++ Syntax
HRESULT SaveFile(BSTR newVal)

Interface
IMixer
IConverter

Last Modified:
2-Feb-2009 Added Converter
SaveToDiskMemory Method

Description
Saves the User Characterization to PNA disk memory. To save to ECal internal memory, use \texttt{SaveToECal Method}. User Characterization can be saved to both PNA disk memory and ECal module memory.

\textbf{Note:} An ECal confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

VB Syntax
\texttt{userChar.SaveToDiskMemory (name)}

Variable \textbf{(Type)} - Description
\begin{itemize}
\item \texttt{userChar} An \texttt{ECalUserCharacterizer (object)}
\item \texttt{name} (String) User characterization name. Although there is no limit to the number of characters, only about 10 characters appear in the Cal Wizard dialog when selecting a user characterization for use.
\end{itemize}

Return Type
Not Applicable

Default
Not Applicable

Examples
\texttt{userChar.SaveToDiskMemory "DUT1"}

C++ Syntax
\texttt{HRESULT SaveToDiskMemory(BSTR name);}

Interface
\texttt{IECalUserCharacterizer2}

Last Modified:
- 15-Jun-2010 Added Note
- 25-Aug-2009 MX New topic
## SaveToECal Method

**Description**  
Saves the User Characterization to the ECal module. This can take several minutes to complete. To save to PNA disk memory, use [SaveToDiskMemory Method](#). User Characterization can be saved to both PNA disk memory and ECal module memory.

**VB Syntax**  
`userChar.SaveToECal`

**Variable**  
(Variable) - Description

- `userChar` - An [ECalUserCharacterizer](#) (object)

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`userChar.SaveToECal`

**C++ Syntax**  
`HRESULT SaveToECal();`

**Interface**  
IECalUserCharacterizer

---

Last Modified:

2-Nov-2008  New topic (8.33)
SaveKits Method

**Description**
Saves the cal kits, typically after modifying a calibration kit. To load a cal kit into the analyzer from the hard drive, use `app.RecallKits`.

**VB Syntax**
`app.SaveKits`

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
`app.SaveKits`

**C++ Syntax**
`HRESULT SaveKits()`

**Interface**
IApplication
SearchCompressionPoint Method

**Description**  Searches the markers domain for the specified compression level.

**VB Syntax**  \texttt{mkr.SearchCompressionPoint}

**Variable (Type) - Description**

\texttt{mkr}  A Marker (object)

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**

\texttt{mkr.SearchCompressionPoint 'Read}

\texttt{See example program}

**C++ Syntax**  \texttt{HRESULT SearchCompressionPoint()}

**Interface**  IMarker4

Last Modified:

8-Feb-2009  MX New topic
**SearchPowerNormalOperatingPoint Method**

**Description**
Initiates a PNOP marker search.
Turns on and sets markers 1, 2, 3, and 4 to calculate various PNOP parameters.
First set **BackOff** and **PinOffset**.
To turn off these markers, either turn them off individually or **DeleteAllMarkers**.
To search a **UserRange**, first activate marker 1 and set the desired UserRange. Then send the SearchPowerNormalOperatingPoint command. The user range applies only to marker 1 searching for the max value. The other markers may fall outside the user range.

**VB Syntax**
```
pnop.SearchPowerNormalOperatingPoint( )
```

**Variable**
- **Type** - **Description**
  - `pnop` A **PNOP (object)**

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
pnop.SearchPowerNormalOperatingPoint
```
See example program

**C++ Syntax**
```
HRESULT SearchPowerNormalOperatingPoint()
```

**Interface**
IPNOP

---

Last Modified:
19-Feb-2010    MX New topic
SearchPowerSaturation Method

**Description**
Initiates a Power Saturation marker search.
Turns on and sets markers 1, 2, and 3 to calculate various Power Saturation parameters.
First set `PMaxBackOff`.
To turn off the Power Saturation markers, either turn them off individually or use `DeleteAllMarkers Method`.

To search a User Range with the PSAT search, first activate marker 1 and set the desired `User Range`. Then send this command. The user range used with the PSAT search only applies to marker 1 searching for the linear gain value. The other markers may fall outside the user range.

**VB Syntax**
```
pSat.SearchPowerSaturation( )
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>pSat</code></td>
<td>A <code>PSaturation (object)</code></td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
pSat.SearchPowerSaturation
```
See example program

**C++ Syntax**
```
HRESULT SearchPowerSaturation()
```

**Interface**
`IPSaturation`

Last Modified:
19-Feb-2010 MX New topic
SearchFilterBandwidth Method

**Description**
Searches the measurement data with the current BandwidthTarget (default is -3). To continually track the filter bandwidth, use BandwidthTracking.

This feature uses markers 1-4. If not already, they are activated. To turn off these markers, either turn them off individually or DeleteAllMarkers.

The bandwidth statistics are displayed on the analyzer screen. To get the bandwidth statistics, use either GetFilterStatistics or FilterBW, FilterCF, FilterLoss, or FilterQ.

The analyzer screen will show either Bandwidth statistics OR Trace statistics; not both.

To search a UserRange with the bandwidth search, first activate marker 1 and set the desired UserRange. Then send the SearchFilterBandwidth command. The user range used with bandwidth search only applies to marker 1 searching for the max value. The other markers may fall outside the user range.

**VB Syntax**
meas.SearchFilterBandwidth

**Variable**
- **meas** A Measurement (object)

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
meas.SearchFilterBandwidth

**C++ Syntax**
HRESULT SearchFilterBandwidth()

**Interface**
IMeasurement
## SearchMax Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Searches the marker domain for the maximum value.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>mark.SearchMax</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A <strong>Marker</strong> <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>mark.SearchMax</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT SearchMax()</code></td>
</tr>
<tr>
<td>Interface</td>
<td>IMarker</td>
</tr>
</tbody>
</table>
# SearchMin Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Searches the marker domain for the minimum value.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchMin</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td>A Marker (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchMin</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SearchMin()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>

1979
SearchNextPeak Method

**Description**  Searches the marker's domain for the next peak value.

**VB Syntax**  `mark/SearchNextPeak`

**Variable**  *(Type)* - Description

  - `mark`  A Marker (object)

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  `mark/SearchNextPeak`

**C++ Syntax**  `HRESULT SearchNextPeak()`

**Interface**  IMarker
# SearchPeakLeft Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Searches the marker's domain for the next <strong>VALID</strong> peak to the left of the marker.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchPeakLeft</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A <strong>Marker</strong> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchPeakLeft</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SearchPeakLeft()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>

1981
### SearchPeakRight Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Searches the marker's domain for the next <strong>VALID</strong> peak to the right of the marker.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchPeakRight</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A <strong>Marker</strong> (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchPeakRight</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SearchPeakRight()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>
### SearchTarget Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Searches the marker's domain for the target value (specified with <code>mark.TargetValue</code>). Searches to the right; then at the end of the search domain, begins again at the start of the search domain.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchTarget</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><em>(Type)</em> - Description</td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A Marker <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchTarget</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT SearchTarget()</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>
### SearchTargetLeft Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Moving to the left of the marker position, searches the marker's domain for the target value (specified with <code>mark.TargetValue</code>).</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchTargetLeft</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A Marker (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchTargetLeft</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SearchTargetLeft()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>

---

1984
**SearchTargetRight Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Moving to the right of the marker position, searches the marker's domain for the target value (specified with <code>mark.TargetValue</code>).</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SearchTargetRight</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>mark</code></td>
<td>A <strong>Marker</strong> <em>(object)</em></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SearchTargetRight</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SearchTargetRight()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>
About Mixer/Converter Settings

SegmentCalculate Method

**Description**
Calculates the specified parameter for the segment.

**VB Syntax**

```vb
conv.SegmentCalculate index, param
```

**Variable (Type) - Description**

- **conv** (A Converter Object)
- **index** (Long integer) Segment for which calculation is performed. Choose a segment between 1 and the current segment count. Use `SegmentCount Property` to read the current count in the **Applied Mixer**.
- **param** (Enum as ConverterCalculation) Mixer port for which to calculate start and stop frequencies. Choose from:

<table>
<thead>
<tr>
<th>enum</th>
<th>1st or only stage requires:</th>
<th>In addition, 2nd stage requires:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 naCalculateINPUT</td>
<td>• Output Start and Stop frequencies</td>
<td>• IF Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td>• LO frequency</td>
<td>• 2nd LO frequency</td>
</tr>
<tr>
<td></td>
<td>• Output sideband (High or Low)</td>
<td>• IF sideband (High or Low)</td>
</tr>
<tr>
<td>1 naCalculateINPUT AndOUTPUT (2 stage mixers ONLY)</td>
<td>NA</td>
<td>• IF Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Both LO frequencies</td>
</tr>
<tr>
<td>2 naCalculateOUTPUT</td>
<td>• Input Start and Stop frequencies</td>
<td>• IF Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td>• LO frequency</td>
<td>• 2nd LO frequency</td>
</tr>
<tr>
<td></td>
<td>• Output sideband (High or Low)</td>
<td>• IF sideband (High or Low)</td>
</tr>
<tr>
<td>3 naCalculateLO1</td>
<td>• Input Start and Stop frequencies</td>
<td>• IF Start and Stop frequencies</td>
</tr>
<tr>
<td></td>
<td>• Output frequency</td>
<td>• 2nd LO frequency</td>
</tr>
<tr>
<td></td>
<td>• Output sideband (High or Low)</td>
<td>• IF sideband (High or Low)</td>
</tr>
<tr>
<td>4</td>
<td>naCalculateLO2</td>
<td>NA</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>----------------</td>
<td></td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
<td></td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
<td></td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mxr.SegmentCalculate 1,2</code> 'Calculates the output frequencies for segment 1.'</td>
<td></td>
</tr>
<tr>
<td><strong>See example program</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT SegmentCalculate(long index, ConverterCalculation param);</td>
<td></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IConverter5</td>
<td></td>
</tr>
</tbody>
</table>

Last Modified:
26-Oct-2010  New command (A.09.33)
Write-only

SelectCalSet Method

**Description**
Selects and applies a Cal Set to the specified channel.

**Note:** Error Correction is not automatically applied as a result of this command being issued. If there is more than one Cal Type in the Cal Set, you must explicitly choose the Cal Type you want to apply. (See meas.Caltype)

**VB Syntax**
```
channel.SelectCalSet calSet, restore
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>channel</td>
<td>(object) - A Channel object</td>
</tr>
<tr>
<td>calSet</td>
<td>(string) - Cal Set to make active. Specify the Cal Set by GUID or Name. Use EnumerateCalSets to list the available Cal Sets.</td>
</tr>
<tr>
<td>restore</td>
<td>(boolean) -</td>
</tr>
<tr>
<td></td>
<td>True (1) - The stimulus stored with the cal set will be applied to the channel.</td>
</tr>
<tr>
<td></td>
<td>False (0) - If a conflict is detected between the existing channel settings and the Cal Set stimulus settings, then the following will occur:</td>
</tr>
<tr>
<td></td>
<td>If interpolation is ON, then interpolation will be attempted. This may fail if the channel frequency is outside the range of the Cal Set.</td>
</tr>
<tr>
<td></td>
<td>If interpolation is OFF, the selection will be abandoned and an error is returned:</td>
</tr>
<tr>
<td></td>
<td>E_NA_CAL_STIMULUS_VALUES_EXCEEDED</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Example**
```
channel.SelectCalSet GUID, 1
chan.SelectCalSet "MyCalSet", 0
```

**C++ Syntax**
```
HRESULT SelectCalSet (BSTR strCset, bool bRestore);
```

**Interface**
IChannel

---

**Last Modified:**
29-Nov-2007    Modified to accept name
**SetAllSegments Method**

**Description**
Uploads a segment table to the PNA replacing any existing segment table. Segments must be ascending in frequency and non-overlapping. If they are not, the segments are 'adjusted' as they are from the User Interface control. The total number of points for all segments cannot exceed the PNA maximum number of points for a sweep.

**VB Syntax**
```
Segs.SetAllSegments (segdata)
```

**Variable**

- `segs`  A `Segments (Collection)`

- `segdata`  (Variant) A 2-dimensional array of Segment data:

  - dimension 0 is the number of elements in each segment.
  - dimension 1 is the number of segments that will be used.

The following is a list of dimension 0 elements for each segment:

**Note:** All elements must be dimensioned as either ALL Double or ALL Variant.

- 0 = Segment state (Boolean True or False)
- 1 = Number of Points in this segment (Integer)
- 2 = Start Freq (Double)
- 3 = Stop Freq (Double)
- 4 = IFBW (Double) optional
- 5 = Dwell Time (Double) optional
- 6 + = Power (Double) optional; see table below.

The first four data elements must always be supplied. After those values, data must be supplied for successive optional elements. For example, to set dwell time values, you must also supply IFBW values, because IFBW (#4) precedes dwell time (#5) in the array order.

The `IFBandwidthOption`, `SweepTimeOption`, and `SourcePowerOption` settings do NOT affect the order in which elements are interpreted.

The number of elements to supply for Power depends on the following two settings:

1. `SourcePowerOption` = True allows segments to have independent power levels.
2. `CouplePorts` = False allows different power levels for each test port.
<table>
<thead>
<tr>
<th>CouplePorts</th>
<th>SourcePowerOption</th>
<th>Number of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>False</td>
<td>False</td>
<td>Each port has its own channel-wide power setting, which is set using TestPortPower. Provide exactly 7 elements per segment. The last element (power) is ignored.</td>
</tr>
<tr>
<td>False</td>
<td>True</td>
<td>Provide 6 elements + total number of ports. The first 7 elements are still interpreted the same. The remaining elements (in-order) are interpreted as the power levels to set on that segment for Ports 2 through N, where N is the total number of ports currently enabled for the PNA or for a PNA with multiport external test set.</td>
</tr>
<tr>
<td>True</td>
<td>False</td>
<td>Provide exactly 7 elements per segment. The last element (power) is ignored.</td>
</tr>
<tr>
<td>True</td>
<td>True</td>
<td>Provide exactly 7 elements per segment. The last element (power) is honored.</td>
</tr>
</tbody>
</table>

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  See a VB example using this command

See a C++ example using this command

**C++ Syntax** HRESULT SetAllSegments (VARIANT Segments );

**Interface**  ISegments2

Last Modified:

28-Apr-2009  More edits

15-Oct-2007  Major edits and link to C++ example
SetBBPorts Method

**Description**  For a Balanced - Balanced device type, maps the PNA ports to the DUT ports. Set the Balanced device type using the **DUTTopology Property**

**VB Syntax**  
```vbnet
balTopology.SetBBPorts p1Pos, p1Neg, p2Pos, p2Neg
```

**Variable (Type) - Description**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>balTopology</code></td>
<td>A <strong>BalancedTopology (object)</strong></td>
</tr>
<tr>
<td><code>p1Pos</code>, <code>p1Neg</code>, <code>p2Pos</code>, <code>p2Neg</code></td>
<td>(Long Integer) PNA port number that connects to each of the following DUT ports:</td>
</tr>
</tbody>
</table>

**Return Type**  Not applicable - To read port mappings, use the **BalancedTopology** properties.

**Default**  Not Applicable

**Examples**  
```vbnet
balTop.SetBBPorts 1,2,3,4
```

**C++ Syntax**  
```c++
HRESULT SetBBPorts (long p1Pos, long p1Neg, long p2Pos, long p2Neg)
```

**Interface**  IBalancedTopology
SetCalInfo Method

Description
Specifies the type of Unguided calibration. This method should be the first method called on the calibrator object. It prepares the internal state for the rest of the calibration.

Note: You can NOT perform a 3 or 4-port cal using SetCalInfo even though there is enumCalTypes. You must use the GuidedCalibration object.

Learn more about reading and writing Cal data using COM

The analyzer can measure both ports simultaneously, assuming you have two of each standard type. For a 2-port cal, See cal.Simultaneous2PortAcquisition

VB Syntax
```vbnet
cal.SetCalInfo (type,rcvPort,srcPort)
```

Variable (Type) - Description

- **cal**: A Calibrator (object)
- **type**: (enum NACalType) - Calibration type. Choose from:
  - 0 - naCalType_Response_Open
  - 1 - naCalType_Response_Short
  - 2 - naCalType_Response_Thru
  - 3 - naCalType_Response_Thru_And_Isol
  - 4 - naCalType_OnePort
  - 5 - naCalType_TwoPort_SOLT
  - 6 - naCalType_TwoPort_TRL
  - 7 - naCalType_None
  - 8 - naCalType_ThreePort_SOLT
  - 9 - Custom
  - 10 - naCalType_FourPort_SOLT

Note: For 1-port cals, the source port = receiver port. For 2, 3,4-port SOLT and TRL, it doesn't matter which port is specified as source and receiver

- **rcvPort**: (long integer) - Receiver Port
- **srcPort**: (long integer) - Source Port

Return Type
NACalType

Default
7- naCalType_None

Examples
```vbnet
cal.setCalInfo(naCalType_Response_Open, 1, 1)
```
C++ Syntax  HRESULT SetCalInfo(tagNACalType calType,long portA, long portB)

Interface  ICalibrator
### SetCalInfoEx Method (for source power cals)

**Description**
This command replaces SetCalInfo2 Method.

Specifies the channel and the source port to be used for the source power calibration about to be performed.

**VB Syntax**
```vbnet
powerCalibrator.SetCalInfoEx channel, srcPort, [powerOffset,] [display]
```

**Variable**
- **(object)** - A `SourcePowerCalibrator` object
- **channel** *(long integer)* - Number of the PNA channel (not power meter channel) on which the source power cal will be performed. If the channel does not already exist, it will be created.
- **srcPort** *(long integer)* - Port number on which the source power cal will be performed.

**Note:** If the port is defined by a string name, such as an external source, a balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-x model, then you must use `chan.getPortNumber` to translate the string into a port number. To learn more see [Remotely Specifying a Source Port](#).

- **[powerOffset]** *(double)* - Optional argument. Sets or returns a power level offset from the PNA test port power. This can be a gain or loss value (in dB) to account for components you connect between the source and the reference plane of your measurement. For example, specify 10 dB to account for a 10 dB amplifier at the input of your DUT. Following the calibration, the PNA power readouts are adjusted by this value. This argument performs the same function as `chan.SourcePowerCalPowerOffset Property`.

- **[display]** *(boolean)* - Optional argument. Enables and disables the display of power readings on the PNA screen. After the source power cal data is acquired, this setting is reset to ON. If unspecified, value is set to ON.
  - **True** - Display of power readings is ON
  - **False** - Display of power readings is OFF

**Return Type**
None

**Default**
Not Applicable

**Examples**
```vbnet
powerCalibrator.SetCalInfoEx 1, 1, -10, True
```

**C++ Syntax**
```cpp
HRESULT SetCalInfoEx( long Channel, long SourcePort, double PowerOffset = 0.,
VARIANT_BOOL bDisplay = VARIANT_TRUE);
```

**Interface**
`ISourcePowerCalibrator4`

---

Last Modified: 1994
24-Apr-2008  Added note for string names
30-Apr-2007  Edited for src strings
SetCenter Method

**Description**
Changes the center stimulus to the stimulus value of the marker. The start stimulus stays the same and the stop is adjusted.

This command does not work with channels that are in CW or Segment Sweep mode.

**VB Syntax**
mark.SetCenter

**Variable (Type) - Description**

*mark*  A Marker (object)

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
mark.SetCenter

**C++ Syntax**
HRESULT SetCenter()

**Interface**
IMarker
### SetCW Method

**Description**  
Changes the analyzer to sweep type CW mode and sets the CW frequency to the marker’s frequency. Does not change anything if current sweep type is other than a frequency sweep.

**VB Syntax**  
`mark.SetCW`

**Variable**  
*(Type)* - Description

- `mark`  
  A Marker *(object)*

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
`mark.SetCW`

**C++ Syntax**  
`HRESULT SetCW()`

**Interface**  
IMarker
**SetCWFreq Method**

**Description**
Sets the CW frequency to the frequency of the active marker. Does NOT change sweep type.

Use ONLY when the current sweep type is sweeping frequency - NOT available in CW or Power Sweep.

Use this command to first set the CW Frequency to a value that is known to be within the current calibrated range, THEN set Sweep Type to naPowerSweep or naCWTimeSweep.

**VB Syntax**
mark.SetCWFreq

**Variable** *(Type) - Description*
mark A Marker *(object)*

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
mark.SetCWFreq

**C++ Syntax**
HRESULT SetCWFreq()

**Interface**
IMarker3

Last Modified:
3-Oct-2008    MX New topic
SetDUTPorts Method

Description
Sets the PNA to DUT port map for FCA measurements. Use DeviceInputPort and DeviceOutputPort to read these values. Changing the ports may limit your ability to use an internal second source. If a selected port is shared by one of the sources, then that source will not be available as an LO source. Learn more about Internal second sources.

VB Syntax
mixer.SetDUTPorts (inputPort, outputPort)

Variable (Type) - Description
mixer A IMixer Interface pointer to the Meas (object)
inputPort (Long) PNA port to be connected to the DUT input.
  - For SMC, choose any unused PNA port.
  - For VMC, set to 1
outputPort (Long) PNA port to be connected to the DUT output. Choose any unused port for SMC and VMC.

Return Type Not Applicable
Default 1,2
Examples mixer.SetDUTPorts =2,1

C++ Syntax HRESULT SetDutPorts(long inputPort, long OutputPort);

Interface IMixer8

Last Modified:
23-Apr-2008 New topic.
### SetElectricalDelay Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Changes the measurement's electrical delay to the marker's delay value.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>mark.SetElectricalDelay</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - Description</td>
</tr>
<tr>
<td>mark</td>
<td>A Marker <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>mark.SetElectricalDelay</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT SetElectricalDelay()</td>
</tr>
<tr>
<td>Interface</td>
<td>IMarker</td>
</tr>
</tbody>
</table>

2000
### SetFailOnOverRange Method

**Description**  
When set TRUE, configures the analyzer to report outOfRange conditions with an error code. Any overrange error will return E\_NA\_LIMIT\_OUTOF RANGE\_ERROR

**Note:** This method is for the benefit of VB clients. The analyzer automatically adjusts overrange conditions to the closest acceptable setting. The VB user will not see that an overrange occurred because the HRESULT is not returned if it has a success code. For more information, see Events/OverRange.

<table>
<thead>
<tr>
<th>VB Syntax</th>
<th>app.SetFailOnOverRange state</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Variable</strong></td>
<td>(Type) - Description</td>
</tr>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
<tr>
<td>state</td>
<td>(boolean) -</td>
</tr>
<tr>
<td></td>
<td>True (1) - Overrange conditions report an error code</td>
</tr>
<tr>
<td></td>
<td>False (0) - Overrange conditions report a success code</td>
</tr>
</tbody>
</table>

**Return Type**  
Not Applicable

**Default**  
False (0)

**VB Example**  
```vbnet
app.SetFailOnOverRange TRUE
On Error Goto ERRHANDLER

' the following overrange will cause ERRHANDLER to be invoked

channel.StartFrequency = 9.9 GHZ
exit

ERRHANDLER:
  print "something failed"
```

**C++ Syntax**  
HRESULT put_SetFailOnOverRange(VARIANT_BOOL mode)

**Interface**  
IAplication
Write only
SetIsolationPaths Method

Description
Adjusts the list of paths (port pairings) for which isolation standards will be measured during calibration.

VB Syntax
`guidedCal.SetIsolationPaths specifier, pathList`

Variable (Type) - Description

obj
Any of the following:

GuidedCalibration (object)

specifier (Enum) - Choose from:

0 - naPathsAll - Measure isolation on all pairings of the ports that are to be calibrated.

1 - naPathsNone - Do not measure isolation on any pairing of the ports to be calibrated.

2 - naPathsAdd - Add one or more specific pairings of ports to the list of port pairings for which isolation will be measured.

3 - naPathsRemove - Remove one or more specific pairings of ports from the list of port pairings for which isolation will be measured.

pathlist (Variant) - port numbers in pairs. One-dimensional array of Long Integers.

Note: pathList is evaluated only when specifier is naPathsAdd or naPathsRemove. For naPathsAll and naPathsNone, pathList is ignored.

Return Type
Not Applicable

Default
Not Applicable

Examples
```
Dim pathList
'selecting to measure isolation on all possible paths for the
ports about to be calibrated
guidedCal.SetIsolationPaths naPathsAll, pathList

'now removing the paths 1-to-2, 2-to-3 and 2-to-4 from the set of all paths
pathList = Array(1,2,2,3,2,4)
guidedCal.SetIsolationPaths naPathsRemove, pathList
```

C++ Syntax
`HRESULT SetIsolationPaths(enum NAPortPathSpecifier specifier, VARIANT pathList);`

Interface
IGuidedCalibration3

Last Modified:
16-Apr-2007  MX New topic
SetIPConfiguration Method

Description
Modifies settings of the PNA computer networking configuration.

VB Syntax
```vb
app.SetIPConfiguration AutoIPAddress, DNSServer1, DNSServer2, HostName, DomainName, IPAddress, SubNet, Gateway, DNSSuffix1, DNSSuffix2
```

or
```vb
retStr = app.SetIPConfiguration (AutoIPAddress, DNSServer1, DNSServer2, HostName, DomainName, IPAddress, SubNet, Gateway, DNSSuffix1, DNSSuffix2)
```

Variable (Type) - Description

- **app** An **Application** (object)

  **AutoIPAddress** (boolean) - Choose either:
  
  **True** - PNA is assigned an IP address by a DHCP server, or will use AutoIP (Dynamic Link-Local Addressing) if DHCP server not found.
  
  **False** – PNA will use the static IP address value specified by IPAddress.

  **DNSServer1** (string) IP address of primary DNS server.

  When AutoIPAddress = True and an empty string is specified for DNSServer1, the PNA will attempt to obtain the addresses of primary and secondary DNS servers automatically.

  When AutoIPAddress = False, an IP address must be specified for DNSServer1 and/or DNSServer2 or else the PNA’s host name will not be resolvable on the computer network.

  **DNSServer2** (string) IP address of secondary DNS server. When specifying an empty string for DNSServer1, then specify an empty string here also.

  **HostName** (string) DNS host name (computer name) to be assigned to this PNA.

  **Note**: If specifying a name different than the PNA’s current host name, the change will not take effect until after you reboot the PNA.

  **DomainName** (string) DNS domain name associated with this PNA.

  **IPAddress** (string) Static IP address to assign to this PNA when AutoIPAddress = False. When AutoIPAddress = True, the value of IPAddress is ignored.

  **SubNet** (string) Subnet mask value to assign to the PNA network configuration.

  **Gateway** (string) Gateway address to assign to the PNA network configuration.

  **DNSSuffix1** (string) Primary suffix to set in the PNA DNS suffix search order. An empty string is allowed.
**DNSSuffix2** *(string)* Secondary suffix to set in the PNA DNS suffix search order. An empty string is allowed.

**retStr** *(string)* String returned by this method should be ignored. It is intended for Agilent diagnostic use.

**Return Type** String

**Default** Not Applicable

**Examples**
```plaintext
app.SetIPConfiguration True, "", "", "MyHostName", "MyRegion.MyCompany.com", "", "255.255.255.0", "123.45.67.890", "", ""

app.SetIPConfiguration False, "123.456.78.90", "234.56.78.901", "MyHostName", "MyRegion.MyCompany.com", "123.456.789.0", "255.255.255.0", "123.45.67.890", "MyCompany.com", ""
```

**C++ Syntax**
```c++
HRESULT SetIPConfiguration(VARIANT_BOOL AutoIPAddress, BSTR DNSServer1, BSTR DNSServer2, BSTR HostName, BSTR DomainName, BSTR IPAddress, BSTR SubNet, BSTR Gateway, BSTR DNSSuffix1, BSTR DNSSuffix2, BSTR *pRetStr);
```

**Interface** IApplication14

---

Last Modified:

2-Jun-2008    MX New topic
### SetPowerAcquisitionDevice Method

**Description**
Sets the power sensor channel (A or B) to be used. This performs the same function as the **Use this sensor only** checkbox in the Power Sensor Settings dialog.

*Note: This method is only necessary when performing an SMC calibration.*

**VB Syntax**
```
pwrCal. SetPowerAcquisitionDevice sensor
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pwrCal</td>
<td>(Object) A SourcePowerCalibrator object</td>
</tr>
<tr>
<td>sensor</td>
<td>(enum NAPowerAcquisitionDevice) The power sensor channel. Choose from:  0 – naPowerSensor_A  1 – naPowerSensor_B</td>
</tr>
</tbody>
</table>

**Default**
Not Applicable

**Examples**
```
pwrCal.PowerAcquisitionDevice naPowerSensor_A
```

**C++ Syntax**
```
HRESULT SetPowerAcquisitionDevice( tagNAPowerAcquisitionDevice enumAcqDevice);
```

**Interface**
ISourcePowerCalibrator3
### SetFrequencyLowPass Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Set the start frequencies when <code>trans.Mode = LowPass</code>.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>trans.SetFrequencyLowPass</code></td>
</tr>
<tr>
<td>Variable</td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>trans</code></td>
<td>A Transform <em>(object)</em></td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td><code>trans.SetFrequencyLowPass</code></td>
</tr>
<tr>
<td>C++ Syntax</td>
<td><code>HRESULT SetFrequencyLowPass(void)</code></td>
</tr>
<tr>
<td>Interface</td>
<td>ITransform</td>
</tr>
</tbody>
</table>
Write-only

SetPortMap Method

Description
Set the DUT-to-PNA port mapping for the Noise Figure, Gain Compression, IMD, IMDx, IMS, or IMSx measurement.

For Noise Figure:
Port mapping is allowed ONLY when NoiseReceiver is set to naStandardReceiver.

When setting IMD and IMS channels:

- When input is 1, output can be 2 or 4.
- When input is 3, output must be 4.
- This setting is necessary only when using the limited port mapping feature. Learn more.

VB Syntax
obj.SetPortMap in,out

Variable (Type) - Description

obj
A GainCompression (object) or
A SweptIMD (object) or
An IMSpectrum (object)
A NoiseFigure (object) - See example program

in
PNA port which is connected to the DUT input.

out
PNA port which is connected to the DUT output.

Return Type
Not Applicable

To read port map, use:
DeviceInputPort Property
DeviceOutputPort Property

Default
1,2

Examples
gca.SetPortMap 2,1

C++ Syntax
HRESULT SetPortMap(long input_port,long output_port);
Interface  IGainCompression
         ISweptIMD
         ISpectrum
         INoiseFigure6

Last Modified:
  30-Apr-2010  Added NF Opt 028
  11-Aug-2009  Added IMD and IMSpectrum (9.0)
  29-Nov-2007  MX New topic
### SetReferenceLevel Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Changes the measurement's reference level to the marker's Y-axis value.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>mark.SetReferenceLevel</code></td>
</tr>
<tr>
<td><strong>Variable</strong> A</td>
<td><code>mark</code> A Marker (object)</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>mark.SetReferenceLevel</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT SetReferenceLevel()</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IMarker</td>
</tr>
</tbody>
</table>

2010
SetSBPorts Method

**Description**
For a Single-ended - Balanced device type, maps the PNA ports to the DUT ports.
Set the Single-ended - Balanced device type using the DUTTopology Property.

**VB Syntax**
`balTopology.SetSBPorts se, bPos, bNeg`

**Variable (Type) - Description**

- `balTopology` A **BalancedTopology** **(object)**
- `se, bPos, bNeg` PNA port number that connects to each of the following DUT ports:

![Diagram](image)

**Return Type**
Not applicable - To read port mappings, use the **BalancedTopology** properties.

**Default**
Not Applicable

**Examples**
`balTop.SetSBPorts 1, 2, 3`

**C++ Syntax**
`HRESULT SetSBPorts (long se, long bPos, long bNeg)`

**Interface**
IBalancedTopology
SetSSBPorts Method

**Description**
For a Single-ended - Single-ended - Balanced device type, maps the PNA ports to the DUT ports.

Set the Single-ended - Single-ended - Balanced device type using the `DUTTopology Property`.

**VB Syntax**
`balTopology.SetSSBPorts se, se2, bPos, bNeg`

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>balTopology</code></td>
<td>A <code>BalancedTopology</code> (object)</td>
</tr>
<tr>
<td><code>se, se2, bPos, bNeg</code></td>
<td>PNA port number that connects to each of the following DUT ports:</td>
</tr>
</tbody>
</table>

![Diagram of Single-ended Port 1, Single-ended Port 2, DUT, Bal Port 3, <bPos>, <bNeg>]

**Return Type**
Not applicable - To read port mappings, use the `BalancedTopology` properties.

**Default**
Not Applicable

**Examples**
`balTop.SetSSBPorts 1, 2, 3, 4`

**C++ Syntax**
`HRESULT SetSSBPorts (long se, long se2, long bPos, long bNeg)`

**Interface**
`IBalancedTopology`
SetupMeasurementsForStep Method

Description
Show the Cal Window, and optionally one or more other specific windows, before acquiring a Cal standard. This command will cause the Cal Window to display the specific measurements that are to be made for that particular Cal standard.

See custom Cal window commands.

VB Syntax
`guidedCal.SetupMeasurementsForStep (n)`

Variable

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>guidedCal</code></td>
<td>A <code>GuidedCalibration</code> (object)</td>
</tr>
</tbody>
</table>

`n` Step number in the calibration process.
Use `GenerateSteps` to determine the total number of steps.
Use `GetStepDescription` to read the description of each step.

Return Type
Not Applicable

Default
Not Applicable

Examples
`guidedCal.SetupMeasurementsForStep 3`

See example using this command

C++ Syntax
`HRESULT SetupMeasurementsForStep(long step);`

Interface
`IGuidedCalibration4`
Write-only

**SetStandardsForClass Method**

**Description**  Set the calibration standard numbers for a specified calibration class. To read the calibration standard numbers use `GetStandardsForClass Method`.

**VB Syntax**  `calKit.SetStandardsForClass (calclassorder, std1, std2, std3, std4, std5, std6, std7)`

**Variable (Type) - Description**

- `calKit`  A CalKit (object)

  - `calclassorder`  (enum NACalClassOrder)  Cal Class. Choose from:
    - 0 - naRefl_1_S11
    - 1 - naRefl_2_S11
    - 2 - naRefl_3_S11
    - 3 - naTran_1_S21
    - 4 - naRefl_1_S22
    - 5 - naRefl_2_S22
    - 6 - naRefl_3_S22
    - 7 - naTran_1_S12
    - 8 - naRefl_1_S33
    - 9 - naRefl_2_S33
    - 10 - naRefl_3_S33
    - 11 - naTran_1_S32
    - 12 - naTran_1_S23
    - 13 - naTran_1_S31
    - 14 - naTran_1_S13
    - 15 - naTRL_T
    - 16 - naTRL_R
    - 17 - naTRL_L

  - `std1…std7`  (long)  Calibration Standard Number. Choose from 1 through 30. Std2 through Std7 are optional.

**Return Type**  Not applicable
**Default**  Not applicable

**Examples**  
- `calkit.SetStandardsForClass naRefl_3_S11, 3, 5, 6`
- `calkit.SetStandardsForClass naTran_1_S21, 4`

**C++ Syntax**  
`HRESULT SetStandardsForClass(NACalClassOrder calclassorder, long std1, long std2, long std3, long std4, long std5, long std6, long std7)`

**Interface**  
ICalKit
**SetStart Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Changes the start stimulus to the stimulus value of the marker. The stop stimulus stays the same and the span is adjusted. This command does not work with channels that are in CW or Segment Sweep mode.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td>mark.SetStart</td>
</tr>
<tr>
<td>Variable</td>
<td>(Type) - Description</td>
</tr>
<tr>
<td>mark</td>
<td>A Marker (object)</td>
</tr>
<tr>
<td>Return Type</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Examples</td>
<td>mark.SetStart</td>
</tr>
<tr>
<td>C++ Syntax</td>
<td>HRESULT SetStart()</td>
</tr>
<tr>
<td>Interface</td>
<td>IMarker</td>
</tr>
</tbody>
</table>
### SetStop Method

**Description**
Changes the stop stimulus to the stimulus value of the marker. The start stimulus stays the same and the span is adjusted. This command does not work with channels that are in CW or Segment Sweep mode.

**VB Syntax**
`mark.SetStop`

**Variable**
- **(Type)**: Description
- **mark**: A Marker (object)

**Return Type**: Not Applicable

**Default**: Not Applicable

**Examples**
`mark.SetStop`

**C++ Syntax**
`HRESULT SetStop()`

**Interface**: IMarker
ShowMarkerReadout Method

Description
Shows and Hides the Marker readout for the active marker in the upper-right corner of the window.

VB Syntax
`win.ShowMarkerReadout state`

Variable
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>win</code></td>
<td>A NAWindow (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean) -</td>
</tr>
<tr>
<td></td>
<td>True (1) - Show the Marker readout</td>
</tr>
<tr>
<td></td>
<td>False (0) - Hide the Marker readout</td>
</tr>
</tbody>
</table>

Return Type
Not Applicable

Default
Not Applicable

Examples
`win.ShowMarkerReadout True`

C++ Syntax
`HRESULT ShowMarkerReadout(VARIANT_BOOL bState)`

Interface
INAWindow
ShowStatusBar Method

**Description**
Shows and Hides the Status Bar. The Status Bar is located across the bottom of the display. The following information is shown for the active measurement:

- Channel number
- Parameter
- Correction On or Off
- Remote or Local operation

**VB Syntax**

```vbnet
app.ShowStatusBar state
```

**Variable**

<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean) -</td>
</tr>
<tr>
<td></td>
<td>True (1) - Show the Status Bar</td>
</tr>
<tr>
<td></td>
<td>False (0) - Hide the Status Bar</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**

```vbnet
app.ShowStatusBar True
```

**C++ Syntax**

```c++
HRESULT ShowStatusBar (VARIANT_BOOL bState)
```

**Interface**

IApplication
**ShowStimulus Method**

**Description**
Shows and Hides the Stimulus (X-axis) information located at the bottom of the display. The start and stop stimulus values are shown for the active measurement.

**VB Syntax**
```
app.ShowStimulus state
```

**Variable**
(Type) - Description

- **app**
  An Application (object)

- **state**
  (boolean) -
  True (1) - Show the Stimulus information
  False (0) - Hide the Stimulus information

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```
app.ShowStimulus True
```

**C++ Syntax**
```
HRESULT ShowStimulus(VARIANT_BOOL bState)
```

**Interface**
IApplciation
ShowTable Method

Description
Shows or hides the specified table for the window's active measurement in the lower part of the window.

VB Syntax
```
win.ShowTable value
```

Variable (Type) - Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>win</td>
<td>A NAWindow (object)</td>
<td></td>
</tr>
<tr>
<td>value</td>
<td>(enum naTable) - The table to show or hide. Choose from:</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0 - naTable_None</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 - naTable_Marker</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 - naTable_Segment</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3 - naTable_Limit</td>
<td></td>
</tr>
</tbody>
</table>

Return Type
Not Applicable

Default
Not Applicable

Examples
```
win.ShowTable naTable_limit
```

C++ Syntax
HRESULT ShowTable (tagNATableType table)

Interface
INAWindow
## ShowTitleBars Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Shows and Hides the Title Bars. The Title Bars are across the top of the Network Analyzer Window and each of the measurement windows. The Window name is shown in the Title Bar.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>app.ShowTitleBars state</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>app</code></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><code>state</code></td>
<td>(boolean)</td>
</tr>
<tr>
<td></td>
<td><strong>True (1)</strong> - Show the Title Bars</td>
</tr>
<tr>
<td></td>
<td><strong>False (0)</strong> - Hide the Title Bars</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>app.ShowTitleBars True</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td>HRESULT ShowTitleBars/(VARIANT_BOOL bState)</td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IApplication</td>
</tr>
</tbody>
</table>
ShowToolbar Method

**Description**
Shows and Hides the specified Toolbar.

**VB Syntax**
```vbnet
app.ShowToolbar toolbar, state
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
<tr>
<td>toolbar</td>
<td>(enum NAToolbarType) - The toolbar to show or hide. Choose from:</td>
</tr>
<tr>
<td>state</td>
<td>(boolean) -</td>
</tr>
<tr>
<td>true (1)</td>
<td>Show the specified toolbar</td>
</tr>
<tr>
<td>false (0)</td>
<td>Hide the specified toolbar</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
1 - naToolbar_ActiveEntry showing; all others hiding.

**Examples**
```vbnet
app.ShowToolbar 1,1 'shows the active entry toolbar
```

**C++ Syntax**
```cpp
HRESULT ShowToolbar(tagNAToolbarType toolbar, VARIANT_BOOL bState)
```

**Interface**
IAplication
Single Method

Description
Sets the trigger count to 1 which will cause the channel to respond once to the trigger source.

How the channel responds to a single trigger depends on the trigger mode (point, trace, and so forth.)

With the exception of the 'sync' argument, this command behaves like the channel 'single' setting from the user interface.

This setting has implications on Calibration. Learn more.

VB Syntax
\[
\text{chan.Single \{sync\}}
\]

Variable (Type) - Description

\text{chan} \quad \text{A Channel (object)}

\{sync\} \quad \text{boolean} \quad \text{-Optional argument.}

- \textbf{True} - The PNA waits (blocks execution) until the entire acquisition process is completed.
- \textbf{False} - The PNA returns immediately - does NOT wait for acquisition to complete (non-blocking). Default setting.

When trigger source is set to Manual:

- with \texttt{sync = True}, trigger source automatically changes to Internal which sends AND allows one trigger signal, then changes back to Manual.
- with \texttt{sync = False}, a trigger signal must also be sent using \texttt{app.ManualTrigger Method}.

Return Type
Not Applicable

Default
Not Applicable

Examples
\[
\text{sync = True} \\
\text{chan.Single sync}
\]

C++ Syntax
\[
\text{HRESULT Single(VARIANT_BOOL bWait)}
\]

Interface
IChannel
Store Method

Description
Saves the path configuration currently associated with channel (ch) to the specified configuration name.

This command is identical to PathConfigurationManager.StoreConfiguration Method.

VB Syntax
pathMgr.StoreConfiguration ch, name

Variable
(pathMgr - type - Description)

pathMgr  PathConfigurationManager (object)

ch  (Long) Channel number of the configuration to be saved.

name  (String) Configuration name. Factory configurations can NOT be overwritten. Specifying the name of a pre-defined factory configuration will result in an error.

Return Type
Not Applicable

Default
Not Applicable

Examples
path.StoreConfiguration(2) "myMixer"

C++ Syntax
HRESULT StoreConfiguration( long channelNum, BSTR configName );

Interface
IPathConfigurationManager

Last modified:
Dec.12, 2006  MX New Command
### StoreConfiguration Method

<table>
<thead>
<tr>
<th>Description</th>
<th>Saves the path configuration currently associated with channel (ch) to the specified configuration name.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>pathMgr.StoreConfiguration ch, name</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type) - Description</strong></td>
</tr>
<tr>
<td><code>pathMgr</code></td>
<td><code>PathConfigurationManager</code> <em>(object)</em></td>
</tr>
<tr>
<td><code>ch</code></td>
<td><em>(Long)</em> Channel number of the configuration to be saved.</td>
</tr>
<tr>
<td><code>name</code></td>
<td><em>(String)</em> Configuration name. Factory configurations can NOT be overwritten. Specifying the name of a pre-defined factory configuration will result in an error.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>path.StoreConfiguration(2) &quot;myMixer&quot;</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT StoreConfiguration( long channelNum, BSTR configName );</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td><code>IPathConfigurationManager</code></td>
</tr>
</tbody>
</table>

Last Modified: 14-Dec-2006    MX New topic
# StoreTheme Method

<table>
<thead>
<tr>
<th><strong>Description</strong></th>
<th>Saves the current color theme to a disc file.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VB Syntax</strong></td>
<td><code>colors.StoreTheme (filename)</code></td>
</tr>
<tr>
<td><strong>Variable</strong></td>
<td><strong>(Type)</strong> - <strong>Description</strong></td>
</tr>
<tr>
<td><code>colors</code></td>
<td>A <code>ComColors</code> <em>(object)</em></td>
</tr>
<tr>
<td><code>filename</code></td>
<td><em>(String)</em> - Path and filename of the theme to save.</td>
</tr>
<tr>
<td><strong>Return Type</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>Not Applicable</td>
</tr>
<tr>
<td><strong>Examples</strong></td>
<td><code>colors.StoreTheme = (&quot;c:/Program Files/Agilent/Network Analyzer/Colors/Theme1.colors&quot;)</code></td>
</tr>
<tr>
<td><strong>C++ Syntax</strong></td>
<td><code>HRESULT StoreTheme(BSTR filename);</code></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>IComColors</td>
</tr>
</tbody>
</table>
StringToNACalClass Method

Description
Converts the returned strings from GetStandardsList into the enumeration (NACalClass) and the port numbers required for PutStandard and GetStandard methods that transmit data in and out of the Cal Set.

Learn more about reading and writing Cal data using COM.

VB Syntax
CalSet.StringToNACalClass (list, std, rcv, src)

Variable (Type) - Description

CalSet (object) - A Cal Set object

list (string) - a string containing the textual description of the standard.

std (enum NACalClass) Choose from:

1 - naClassA
2 - naClassB
3 - naClassC
4 - naClassD
5 - naClassE
6 - naReferenceRatioLine
7 - naReferenceRatioThru

SOLT Standards
1 - naSOLT_Open
2 - naSOLT_Short
3 - naSOLT_Load
4 - naSOLT_Thru
5 - naSOLT_Isolation

TRL Standards
1 - naTRL_Reflection
2 - naTRL_Line_Reflection
3 - naTRL_Line_Tracking
4 - naTRL_Thru
5 - naTRL_Isolation
rcv (long) - port number of the receiver

src (long) - port number of the source

Return Type Not Applicable

Default Not Applicable

Examples guid = CalSet.StringToNACalClass(list, std, rcv, src)

C++ Syntax HRESULT StringToNACalClass( BSTR* str, NACalClass* item, long *rcv, long *src);

Interface ICalSet
StringToNAErrorTerm2 Method

Description  Converts the returned strings from GetErrorTermList into the enumeration (NAErrorTerm2) and the port numbers required for PutErrorTerm and GetErrorTerm methods that transmit data in and out of the Cal Set.

Learn more about reading and writing Cal data using COM

VB Syntax  
```vbnet
CalSet.StringToNAErrorTerm2 (list, eterm, rcv, src)
```

Variable (Type) - Description

Cal Set (object) - A Cal Set object

list (string) - a string containing the textual description of the error term.

eterm (enum As NaErrorTerm2). Choose from:
- 0 - naET_Directivity (rcv = src)
- 1 - naET_SourceMatch (rcv = src)
- 2 - naET_ReflectionTracking (rcv = src)
- 3 - naET_TransmissionTracking (rcv != src)
- 4 - naET_LoadMatch (rcv != src)
- 5 - naET_Isolation (rcv != src)

rcv (long) - port number of the receiver

src (long) - port number of the source

Return Type  Not Applicable

Default  Not Applicable

Examples  
```vbnet
CalSet.StringToNAErrorTerm2 str, term, rcv, src
```

C++ Syntax  
```cpp
HRESULT StringToNAErrorTerm2 (BSTR* str, NAErrorTerm2* item, long *rcv, long *src);
```

Interface  ICalSet
Write-only

SweepOnlyCalChannelDuringCalAcquisition Method

Description
Clears ALL flags for channels to sweep during calibration except the Cal channel. To flag a channel, see AllowChannelToSweepDuringCalAcquisition Method.

VB Syntax
`calMgr.SweepOnlyCalChannelDuringCalAcquisition`

Variable
- **Type** - Description
  - `calMgr` (object) - A CalManager object

Return Type
Not Applicable

Default
Not Applicable

Example
`calMgr.SweepOnlyCalChannelDuringCalAcquisition`

See example using this command

C++ Syntax
`HRESULT SweepOnlyCalChannelDuringCalAcquisition()`

Interface
ICalManager5

Last Modified:
8-Nov-2007    MX New topic
TestsetCatalog Method

Description
Returns a list of supported testsets.

VB Syntax
\[ data = Tsets.TestsetCatalog \]

Variable
\textbf{(Type) - Description}

\begin{itemize}
  \item \textit{data} \textbf{(variant array)} - Variable to store the returned data.
  \item \textit{Tsets} \textbf{(object)} - An \texttt{ExternalTestSets} collection
\end{itemize}

Return Type
Variant

Default
Not Applicable

Examples
\texttt{value = Tsets.TestsetCatalog}

C++ Syntax
\texttt{HRESULT TestsetCatalog (VARIANT* Data);}

Interface
\texttt{IEexternaTestSets}
**UserPreset Method**

**Description**
Performs a User Preset. There must be an active User Preset state file (see UserPresetLoadFile and UserPresetSaveState) or an error will be returned.

Regardless of the state of the User Preset Enable checkbox, the app.Preset command will always preset the PNA to the factory preset settings, and app.UserPreset will always perform a User Preset.

**VB Syntax**
```vbnet
app.UserPreset
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
```vbnet
app.UserPreset
```

**C++ Syntax**
```c++
HRESULT UserPreset()
```

**Interface**
IAplication6
UserPresetLoadFile Method

**Description**
Loads an existing instrument state file (.sta or .cst) to be used for User Preset. Subsequent execution of `app.UserPreset` will cause the PNA to assume this instrument state.

Regardless of the state of the User Preset Enable checkbox, the `app.Preset` command will always preset the PNA to the factory preset settings, and `app.UserPreset` will always perform a User Preset.

**VB Syntax**

```vbnet
app.UserPresetLoadFile (file)
```

**Variable**

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>app</strong></td>
<td>An Application (object)</td>
</tr>
<tr>
<td><strong>file</strong></td>
<td>(String) Full path, name, and extension of the file to be loaded.</td>
</tr>
</tbody>
</table>

**Return Type**

Not Applicable

**Default**

Not Applicable

**Examples**

```vbnet
app.UserPresetLoadFile ("C:/Program Files/Agilent/Network Analyzer/Documents/10MHzto20GHz.sta")
```

**C++ Syntax**

```cpp
HRESULT UserPresetLoadFile (BSTR bstrFile)
```

**Interface**

IApplication6
Write-only

UserPresetSaveState Method

**Description**  Saves the current instrument settings as UserPreset.sta. Subsequent execution of `app.UserPreset` will cause the PNA to assume this instrument state.

Regardless of the state of the User Preset Enable checkbox, the `app.Preset` command will always preset the PNA to the factory preset settings, and `app.UserPreset` will always perform a User Preset.

**VB Syntax**  
```
app.UserPresetSaveState
```

**Variable**  
(Type) - Description

- `app`  An `Application` (object)

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**  
```
app.UserPresetSaveState
```

**C++ Syntax**  
```
HRESULT UserPresetSaveState()
```

**Interface**  `IApplication6`
## WriteData Method

**Description**  
Writes a 13-bit value to the specified address using the AD0 through AD12 lines of the external test set connector. The PNA generates the appropriate timing signals. It automatically controls timing signals LDS, LAS and RLW to strobe the address, then the data, to the external test set. See the [timing diagram](#) for Address and Data I/O read.

**VB Syntax**  
```
ExtIO.WriteData (address) = value
```

**Variable (Type) - Description**

| ExtIO  | (object) - An External IO object |
| address | (variant) - Address to be written to. |
| value   | (variant) - 13-bit word to write |

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
```
ExtIO.WriteData (15) = 12
```

**C++ Syntax**  
```
HRESULT WriteData(VARIANT Address, VARIANT Data);
```

**Interface**  
IHWExternalTestSetIO
Write-only

About the ExtTestSetIO connector

**WriteRaw Method**

<table>
<thead>
<tr>
<th>Description</th>
<th>Writes a 16-bit value to the external test set connector lines AD0 - AD12, RLW, LAS and LDS. The analyzer does NOT generate the appropriate timing signals. The user has control of all 16 lines using this write method.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Note:</strong> When RLW (pin 25) is set to 1 (high) it causes lines AD0 - AD12 to float. It disables their output latches and sets the hardware for reading. LDS and LAS are not affected by this behavior.</td>
<td></td>
</tr>
</tbody>
</table>

Below is the format of data that is written with WriteRaw:

* This Output will float if RLW (bit-13) is set high

<table>
<thead>
<tr>
<th>Pin</th>
<th>Bit</th>
<th>Signal name</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>0</td>
<td>AD0*</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>AD1*</td>
</tr>
<tr>
<td>11</td>
<td>2</td>
<td>AD2*</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>AD3*</td>
</tr>
<tr>
<td>9</td>
<td>4</td>
<td>AD4*</td>
</tr>
<tr>
<td>21</td>
<td>5</td>
<td>AD5*</td>
</tr>
<tr>
<td>20</td>
<td>6</td>
<td>AD6*</td>
</tr>
<tr>
<td>19</td>
<td>7</td>
<td>AD7*</td>
</tr>
<tr>
<td>6</td>
<td>8</td>
<td>AD8*</td>
</tr>
<tr>
<td>5</td>
<td>9</td>
<td>AD9*</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>AD10*</td>
</tr>
<tr>
<td>17</td>
<td>11</td>
<td>AD11*</td>
</tr>
<tr>
<td>3</td>
<td>12</td>
<td>AD12*</td>
</tr>
<tr>
<td>25</td>
<td>13</td>
<td>RLW</td>
</tr>
<tr>
<td>24</td>
<td>14</td>
<td>LDS</td>
</tr>
<tr>
<td>8</td>
<td>15</td>
<td>LAS</td>
</tr>
</tbody>
</table>
VB Syntax  
\textit{ExtIO.WriteRaw value}

**Variable**  
(\textbf{Type}) - Description

\textit{ExtIO} (\textbf{object}) - An External IO object

\textit{value} (\textbf{variant}) - Data to be written

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
\texttt{ExtIO.WriteRaw 12}

**C++ Syntax**  
\texttt{HRESULT WriteRaw( VARIANT Output );}

**Interface**  
IHWExternalTestSetIO
Write-only

**WriteSnpFileWithSpecifiedPorts Method**

**Description**  
**Note:** This command replaces `app.Save (.snp)`. This command is more explicit regarding the data to be saved, and works for PNAs with multiport test sets.  
Saves SnP data to the specified file. Learn more about SnP data.

**VB Syntax**  
`data = meas.WriteSnpFileWithSpecifiedPorts ports, filename`

**Variable**  
<table>
<thead>
<tr>
<th>(Type)</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>data</em></td>
<td>(Variant) array to store the data.</td>
</tr>
<tr>
<td><em>meas</em></td>
<td>A Measurement (object)</td>
</tr>
<tr>
<td><em>ports</em></td>
<td>(Variant Array) One dimensional array containing a list of port numbers for which snp data is requested.</td>
</tr>
<tr>
<td><em>filename</em></td>
<td>(string) Full path, filename, and suffix to store the data.</td>
</tr>
</tbody>
</table>

  The suffix is not checked for accuracy. If saving 2 ports, specify "filename.s2p"; If saving 3 ports, specify "filename.s3p." and so forth.

  SnP data can be output using several data formatting options. See [SnPFormat Property](#).

**Return Type**  
Variant array - automatically dimensioned to the size of the data.

**Default**  
Not Applicable

**Examples**  
'This VBScript example can be pasted into a notepad file and run on the PNA as a macro. Learn how.

```vbscript
Set pna = CreateObject("AgilentPnA835x.application")
Set meas = pna.ActiveMeasurement

'List the port numbers for required data
ports = Array(1,2,4)

'specify where to save the data
filename="C:/Program Files/Agilent/Network Analyzer/Documents/MyData.s3p"
meas.WriteSnpFileWithSpecifiedPorts ports, filename
```

**C++ Syntax**  
`HRESULT WriteSnpFileWithSpecifiedPorts(VARIANT portsToMeasure,BSTR filename);`

**Interface**  
IMeasurement7
9/18/06   MQ Added for multiport
OnCalEvent

**Description**  Triggered by a calibration event. See a list of [CAL Events](#).

**Note:** Some Severe Events are also used as Error Messages

**VB Syntax**

```vbnet
Sub app_OnCalEvent(ByVal eventID As Variant, ByVal chanNum As Variant, ByVal measNum As Variant)

Variable (Type) - Description

- **app**  An Application (object)
- **eventID**  Code number of the event which occurred
- **chanNum**  Channel Number of the event
- **measNum**  Measurement Number of the event

**Return Type**  Not Applicable

**Default**  Not Applicable

**Examples**

```vbnet
Sub pna_OnCalEvent(ByVal eventID As Variant, ByVal channelNumber As Variant, ByVal measurementNumber As Variant)
    MsgBox ("A Calibration event has occured")
End Sub
```

**C++ Syntax**

```csharp
HRESULT OnCalEvent(VARIANT eventID, VARIANT channelNumber, VARIANT measurementNumber)
```

**Interface**  IApplication

**Selected Cal Events**

- 512 naEventID_CAL_QUESTIONABLE
- 513 naEventID_CAL_STD_NEEDED
- 514 naEventID_CAL_STATE_NOT_HW_COMPATIBLE
- 515 naEventID_CAL_REQUIRED
- 516 naEventID_CAL_CORRECTION_TURNED_OFF
- 517 naEventID_CAL_CORRECTION_TURNED_OFF_INTERPOLATION_OFF
- 518 naEventID_CAL_CORRECTION_RESTORED
- 519 naEventID_CAL_CORRECTION_TURNED_OFF_FREQRANGE_EXCEEDED
See Also

Errors and the SCPIStringParser Object

Last modified:

Nov. 6, 2006 Added events
OnChannelEvent

Description  Triggered by a channel event.

VB Syntax  Sub app_OnChannelEvent(ByVal eventID As Variant, ByVal chanNum As Variant)

Variable (Type) - Description

app  An Application (object)

eventID  Code number of the event which occurred

chanNum  Channel Number of the event

Return Type  Not Applicable

Default  Not Applicable

Examples  Sub pna_OnChannelEvent(ByVal eventID As Variant, ByVal channelNumber As Variant)

End If
End Sub

C++ Syntax  HRESULT OnChannelEvent(VARIANT eventID, VARIANT channelNumber)

Interface  IApplication

Selected Channel Events
1792 naEventID_CHANNEL_SWEEP_COMPLETE
1793 naEventID_CHANNEL_TRIGGER_COMPLETE
1796 naEventID_SET_CHANNEL_DIRTY
1797 naEventID_CLEAR_CHANNEL_DIRTY
1801 naEventID_ALL_SWEEPS_COMPLETED_AND_PROCESSED
1805 naEventID_CHANNEL_CREATED
1806 naEventID_CHANNEL_DELETED
1876 naEventID_NO_SOURCE_ATTEN
1879 naEventID_FREQ_OFFSET_OVERRANGE_SO_TURNED_OFF
1883 naEventID_PORT_NUMBER_OUT_OF_RANGE

See Also
Errors and the SCPIStringParser Object
Last modified:

March 2, 2007    Added channel create and delete

Nov. 6, 2006    Added events
About Analyzer Events

OnDisplayEvent

Description
Triggered by a display event.

VB Syntax
Sub app_OnDisplayEvent(ByVal eventID As Variant, ByVal winNum As Variant, ByVal traceNum As Variant)

Variable (Type) - Description

app  An Application (object)

eventID  Code number of the event which occurred

winNum  Window Number of the event

traceNum  Trace Number of the event

Return Type
Not Applicable

Default
Not Applicable

Examples
Sub pna_OnDisplayEvent(ByVal eventID As Variant, ByVal windowNumber As Variant, ByVal traceNumber As Variant)
MsgBox("A Display event has occurred")
End Sub

C++ Syntax
HRESULT OnDisplayEvent(VARIANT eventID, VARIANT windowNumber, VARIANT traceNumber)

Interface  IApplication

Selected Display Events
1541 naEventID_PRINT_SETUP_FAILURE
1542 naEventID_PRINT_CANCELED

See Also
Errors and the SCPIStringParser Object

Last modified:
Nov. 6, 2006  Added events
### OnHardwareEvent

**Description**  
Triggered by a hardware event. See a list of [Hardware Events](#).  

**Note:** Some Severe Events are also used as Error Messages

**VB Syntax**  
Sub `app_OnHardwareEvent(ByVal eventID As Variant)`

<table>
<thead>
<tr>
<th>Variable</th>
<th>(Type) - Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>app</code></td>
<td>An Application <em>(object)</em></td>
</tr>
<tr>
<td><code>eventID</code></td>
<td>Code number of the event which occurred</td>
</tr>
</tbody>
</table>

**Return Type**  
Not Applicable

**Default**  
Not Applicable

**Examples**  
Private Sub pna_OnHardwareEvent(ByVal eventID As Variant)  
MsgBox ("A Hardware event has occurred")  
End Sub

**C++ Syntax**  
`HRESULT OnHardwareEvent(VARIANT eventID)`

**Interface**  
`IApplication`

### Selected Hardware Events

848 `naEventID_PHASELOCK`  
852 `naEventID_RFPOWEROFF`  
853 `naEventID_RFPOWERON`  
855 `naEventID_UNLEVELED`  
857 `naEventID_OVERLOAD`  
914 `naEventID_TRIGGERQUIRES_EDGELIVETRIGGER`  
915 `naEventID_TRIGGERREQUIRESTRIGGEROUT`

### See Also

[Errors and the SCPIStringParser Object](#)

Last modified: 2048
Nov. 6, 2006   Added events
OnMeasurementEvent

Description
Triggered by a measurement event.

VB Syntax
Sub app_OnMeasurementEvent(ByVal eventID As Variant, ByVal measNum As Variant)

Variable (Type) - Description
app An Application (object)
eventID Code number of the event which occurred
measNum Measurement Number of the event

Return Type Not Applicable
Default Not Applicable

Examples
Private Sub pna_OnMeasurementEvent(ByVal eventID As Variant, ByVal measurementNumber As Variant)
    MsgBox ("A Measurement event has occurred")
End Sub

C++ Syntax
HRESULT OnMeasurementEvent(VARIANT eventID, VARIANT measurementNumber)

Interface IApplication

Selected Measurement Events
1024 naEventID_NO_VALID_MEMORY_TRACE
1028 naEventID_LIMIT_FAILED
1029 naEventID_LIMIT_PASSED
1034 naEventID_MEMORY_NOT_SAVED
1035 naEventID_SET_AVERAGE_COMPLETE
1036 naEventID_CLEAR_AVERAGE_COMPLETE
1111 naEventID_MARKER_BANDWIDTH_NOT_FOUND
1112 naEventID_PEAK_NOT_FOUND
1113 naEventID_TARGET_VALUE_NOT_FOUND
See Also

Errors and the SCPIStringParser Object

Last modified:

Nov. 6, 2006  Added events
OnSCPIEvent

Description
Triggered by a SCPI event.

Note: Some Severe Events are also used as Error Messages

VB Syntax
Sub app_OnSCPIEvent(ByVal eventID As Variant)

Variable (Type) - Description

app An Application (object)
eventID Code number of the event which occurred

Return Type Not Applicable

Default Not Applicable

Examples
Private Sub pna_OnSCPIEvent(ByVal eventID As Variant)
MsgBox ("A SCPI event has occurred")
End Sub

C++ Syntax
HRESULT OnSCPIEvent(VARIANT eventID )

Interface IApplication

Selected SCPI Parser Events
1281 naEventID NOTHING_TO_SAY
1284 naEventID SCPI_STATUS_BYTE_CHANGE
1360 naEventID BAD_SCPI_EXECUTE
1375 naEventID CALC_MEASUREMENT_SET_TO_NONE

See Also
Errors and the SCPIStringParser Object

Last modified:
Nov. 6, 2006 Added events
About Analyzer Events

**OnSystemEvent**

**Description**
Triggered by a system event. See a list of System Events, also known as general events.
See also EnableSourceUnleveledEvents Property

**Note:** Some Severe Events are also used as Error Messages

**VB Syntax**
Sub app_OnSystemEvent(ByVal eventID As Variant)

**Variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>app</td>
<td>An Application (object)</td>
<td></td>
</tr>
<tr>
<td>eventID</td>
<td>Code number of the event which occurred</td>
<td></td>
</tr>
<tr>
<td>chanNum</td>
<td>Channel Number of the event</td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**
Not Applicable

**Default**
Not Applicable

**Examples**
Private Sub pna_OnSystemEvent(ByVal eventID As Variant)
MsgBox ("A System event has occured")
End Sub

**C++ Syntax**
HRESULT OnSystemEvent(VARIANT eventID)

**Interface**
IApplication

**Selected System Events**
- 2048 naEventID_OPTION_NOT_INSTALLED
- 2049 naEventID_FEATURE_NOT_AVAILABLE
- 2050 naEventID_FEATURE_NOT_VALID
- 2051 naEventID_SAVEFILE_OK
- 2063 naEventID_RECALLFILE_SUCCESS
- 2130 naEventID_PRINTER_TROUBLE
- 2133 naEventID_TRIGGERDENIED
- 2134 naEventID_MACRO_FAILED
2144 naEventID_NO_LICENSE
2163 naEventID_PRESET
2166 naEventID_TRIGGERFAILED

See Also
Errors and the SCPIStringParser Object

Last modified:

Nov. 6, 2006  Added events
### OnUserEvent

<table>
<thead>
<tr>
<th>Description</th>
<th>Reserved for future use.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VB Syntax</td>
<td><code>Sub app_OnUserEvent</code></td>
</tr>
</tbody>
</table>
C++ Example

The following example uses the smart pointer created by Microsoft Visual Studio. The calls to CoInitialize and CoUninitialize open and close the COM libraries.

Also notice that the pointers local to the main routine are explicitly released. When smart pointers go out of scope, they will perform this duty implicitly. However, we are calling CoUninitialize before they have the chance to be destroyed, so we are obliged to release them.

```c++
// An example program to illustrate the use of #import to bind to the
// PNA type library.
//

#ifndef _UNICODE
#define _UNICODE
#endif
#include "stdafx.h"
#include "stdio.h"
#include "math.h"

/////////////////////////////////////////////
// import the network analyzer type library
/////////////////////////////////////////////
#import "C:/Program Files/Common Files/Agilent/Pna/835x.tlb" no_namespace,
named_guids
/////////////////////////////////////////////
// include the error definitions for the PNA so we can implement
// error handling.
/////////////////////////////////////////////
#include "C:/Program Files/Common Files/Agilent/Pna/errorsystemmessage.h"

IApplicationPtr pNA; // top level application pointer
float fScalarData [1601]; // global buffer for data retrieval
float fScalarData2[1601];
DWORD dwCookie;

/////////////////////////////////////////////
// SetupChannel:
/////////////////////////////////////////////
// input: pointer to the channel
// function: sets properties on the channel
/////////////////////////////////////////////
void SetupChannel(IChannelPtr pChannel)
{
  pChannel->put_StartFrequency( 1.2E9 );
  pChannel->put_StopFrequency ( 4.2E9 );
  pChannel->put_NumberOfPoints ( 201);
}
```
void AcquireData( IChannelPtr pChannel )
{
    pChannel->Single( TRUE );
}

void ReadScalarData(IMeasurementPtr pMeas )
{
    IArrayTransferPtr pDataTransfer;
    pDataTransfer = pMeas;
    long numVals = 1601;
    float* pData = fScalarData;

    if(pDataTransfer){
        pDataTransfer->getScalar( naMeasResult, naDataFormat_LogMag, &numVals, pData);

        for (int i = 0; i < numVals; i++)
            printf("%d/%f/n",i,pData[i]);
    }
    TCHAR msg[100];
    BSTR param;
    pMeas->get_Parameter(&param);
    swprintf(msg,L"Review %s data",param);
    MessageBox(NULL,msg,L"User Message",0);
    ::SysFreeString(param);
}

void ReadComplexData(IMeasurementPtr pMeas )
{
    IArrayTransferPtr pDataTransfer;
    pDataTransfer = pMeas;
    long numVals = 1601;
    float* pReal= fScalarData;
    float* pImag = fScalarData2;

    for (int i = 0; i < numVals; i++)
        printf("%d/%f/%f/n",i,pReal[i],pImag[i]);
}
if(pDataTransfer){

pDataTransfer->getPairedData( naRawData, naRealImaginary, &numVals, pReal, pImag);

for (int i = 0; i < numVals; i++)
printf("%d/%f/%f/n",i,pReal[i], pImag[i]);
}

TCHAR msg[100];
BSTR param;
pMeas->get_Parameter(&param);
swprintf(msg,L"Review %s data",param);
MessageBox(NULL,msg,L"User Message",0);
::SysFreeString(param);
} 
} 

void PutData( IMeasurementPtr pMeas )
{
IArrayTransferPtr pDataTransfer;
pDataTransfer = pMeas;
long numVals = 201;

if(pDataTransfer){
NAComplex* pComplex = new NAComplex[numVals];
pComplex[0].Im = 0;
pComplex[0].Re = 1;
for (int i = 1; i < numVals; i++)
{
 pComplex[i].Im = (float)sin(i)/i;
pComplex[i].Re = (float)cos(i)/i;
}
}
}

void printError( HRESULT hr)
{
BSTR text;
hr = pNA->get_MessageText ((NAEventID) hr, &text);
MessageBox(NULL, text, L"Network Analyzer error", 0);
::SysFreeString(text);
}

// main
int main(int argc, char* argv[])
{
    HRESULT hr;
    const long channel1 = 1;
    const long window1 = 1;
    const long srcport = 1;
    IMeasurementPtr pMeasurement;
    IChannelPtr pChannel;

    // initialize COM libraries
    CoInitialize(NULL);

    try {
        pNA = IApplicationPtr("AgilentPNA835x.Application.1");
        pNA->put_Visible(TRUE);
        pNA->Reset();

        pNA->CreateMeasurement (channel1, "S21", srcport, 3);
        hr = pNA->get_ActiveChannel( &pChannel);

        if (SUCCEEDED (hr))
        {
            SetupChannel( pChannel);
            AcquireData(pChannel);
        }

        hr= pNA->get_ActiveMeasurement( &pMeasurement);
        if (SUCCEEDED(hr))
        {
            pMeasurement->put_Format( naDataFormat_Polar);
            ReadScalarData( pMeasurement);
            ReadComplexData( pMeasurement);
            PutData(pMeasurement);
        }
        if (FAILED(hr))
        {
            printError(hr);
        }
    }

    // make sure to release the remaining pointers
// before calling CoUninitialize

pMeasurement.Release();
pChannel.Release();
pNA.Release();
}
catch (_com_error err)
{
    printError( err.Error() );
}

CoUninitialize();
return 0;
}
Reading Cal Set Data using COM

This example iterates over the entire collection of Cal Sets that currently reside in the PNA. It reads the entire list of error term strings from each Cal Set and queries the data for each term. It then does the same for the standards data.

Learn more about Reading and Writing Calibration data using COM.
Learn more about Cal Sets.
See example: Writing Cal Set Data using COM

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as CalSets.vbs. Learn how to setup and run the macro.

```vbs
Dim pna
Dim cset
Dim calsets

' create the pna object
' to run on a remote PC, substitute 'name' for the full computer name of your PNA
' to run as a macro on the PNA, remove ,"name"
Set pna = CreateObject("AgilentPNA835x.Application", "name")

wscript.echo pna.IDString

' obtain the calset collection
Set calsets = pna.GetCalManager.calsets

' loop thru the calsets
Dim c
For c = 1 To calsets.count
Set cset = calsets.Item(c)

' wscript.echo prints values to a message box
wscript.echo "calset = ", cset.GetGUID, cset.Description

' iterate through error terms data
Dim vterms,fdata
vterms = cset.GetErrorTermList2(0, "")
if (Not IsEmpty(vterms)) then
For i = LBound(vterms) To UBound(vterms)
    wscript.echo vterms(i)
    vdata = cset.GetErrorTermByString(0,vterms(i))
fdata = cset.GetErrorTermStimulus(0,vterms(i))
    wscript.echo vdata(1,0), vdata(1,1)
    wscript.echo fdata(1,0), fdata(1,1)
Next
end if

' iterate through standards data
```
vterms = cset.GetStandardList2("")
if (Not IsEmpty(vterms)) then
For i = LBound(vterms) To UBound(vterms)
  wscript.echo vterms(i)
  vdata = cset.GetStandardByString( vterms(i) )
  wscript.echo vdata(1,0), vdata(1,1)
Next
end if
Next

Last modified:

  2-May-2011   Added GetErrorTermStimulus
Intro to Examples

Getting Trace Data from the Analyzer

This Visual Basic program:

- Retrieves Scalar Data from the Analyzer and plots it.
- Retrieves Paired Data from the Analyzer and plots it.
- Retrieves Complex Data from the Analyzer and plots it.

To use this code, prepare a form with the following:

- Two MSCharts named MSChart1 and MSChart2
- Three buttons named GetScalar, GetPaired, GetComplex

Note: You can get MSChart in Visual Basic by clicking Project / Components / Microsoft Chart Control

```vbnet
'Put this in a module
Public dlocation As NADataStore
Public numpts As Long
Public fmt As NADataFormat
Public app As Application
Public measData As IArrayTransfer
Public chan As Channel

Sub Form_Load()
    'Change analyzerName to your analyzer's full computer name
    Set app = CreateObject("AgilentPNA835x.Application", "analyzerName")
    Set measData = app.ActiveMeasurement
    Set chan = app.ActiveChannel

    'To pick a location to get the data from remove the comment from one of these
    dlocation = naDataStore
    'dlocation = naCorrectedData
    'dlocation = naMeasResult
    'dlocation = naRawMemory
    'dlocation = naMemoryResult

    'setup MSChart1 and MSChart2
    'right click on the chart and select:
    ' - line chart
    ' - series in rows
End Sub

Sub GetComplex_Click()
    ReDim Data(numpts) As NAComplex
    Dim Real(201) As Single
    Dim Imag(201) As Single
    numpts = chan.NumberOfPoints
```
'You cannot change the format of Complex Data
Call trigger
'get data
measData.GetNAComplex dlocation, numpts, Data(0)
'plot data
Dim i As Integer

For i = 0 To numpts - 1
    Real(i) = Data(i).Re
    Imag(i) = Data(i).Im
Next i
MSChart1 = Real()
MSChart2.Visible = True
MSChart2 = Imag()
Call Sweep
End Sub

Sub GetPaired_Click()
ReDim Real(numpts) As Single
ReDim Imag(numpts) As Single
numpts = chan.NumberOfPoints

' To pick a format, remove the comment from one of these
fmt = naLogMagPhase
'fmt = naLinMagPhase
Call trigger
'Get data
measData.getPairedData dlocation, fmt, numpts, Real(0), Imag(0)
'Plot Scalar
MSChart1 = Real()
MSChart2.Visible = True
MSChart2 = Imag()
Call Sweep
End Sub

Sub GetScalar_Click()
ReDim Data(numpts) As Single
numpts = chan.NumberOfPoints

'To pick a format remove the comment from one of these
fmt = naDataFormat_LogMag
'fmt = naDataFormat_LinMag
'fmt = naDataFormat_Delay
'fmt = naDataFormat_Phase
'fmt = naDataFormat_Real
'fmt = naDataFormat_Imaginary
Call trigger
'Get data
measData.GetScalar dlocation, fmt, numpts, Data(0)
'Plot Data
MSChart1 = Data()
MSChart2.Visible = False
Call Sweep
End Sub

Sub trigger()
    'The analyzer sends continuous trigger signals
    app.TriggerSignal = naTriggerInternal
    'The channel will only accept one, then go into hold
    'Sync true will wait for the sweep to complete
    sync=True
    chan.Single sync
End Sub

Sub Sweep()
    'The channel goes back to accepting all triggers
    chan.Continuous
End Sub
Perform a Guided Calibration using COM

This example uses the GuidedCalibration interface to perform either a 2-port or 4-port calibration. Learn more about Reading and Writing Calibration data using COM.

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as Calibrate.vbs. Learn how to setup and run the macro.

```
Set pna = CreateObject("AgilentPNA835x.Application")
Set calMgr = pna.GetCalManager
Set guidedCal = calMgr.GuidedCalibration
Set chan = pna.ActiveChannel
chanNum = chan.ChannelNumber
' Initialize guided cal to be performed on the active channel.
' The boolean argument of True indicates to store the cal only
' in the channel's calibration register.  If instead you wish
' to create a new calset that the new cal will get stored to,
' comment out this next line and uncomment the three lines below it.
guidedCal.Initialize chanNum, True
'Set calset = calMgr.CreateCalSet(chanNum)
'chan.SelectCalSet calset.GetGUID, True
'guidedCal.Initialize chanNum, False

' To perform 2-port cal, Uncomment the following
' Then comment the 4-port cal

' Do 2-port cal
'TwoPortGuidedCal

' Do 4-port cal
FourPortGuidedCal

Sub TwoPortGuidedCal()
' Select the connectors
guidedCal.ConnectorType(1) = "APC 3.5 female"
guidedCal.ConnectorType(2) = "APC 3.5 male"
For i = 3 To pna.NumberOfPorts
  guidedCal.ConnectorType(i) = "Not used"
Next
value = MsgBox("Connectors defined for Ports 1 and 2")
' Select the Cal Kit for each port being calibrated.
guidedCal.CalKitType(1) = "85052D"
guidedCal.CalKitType(2) = "85052D"
' To use an ECal module instead, comment out the above two lines
```
' and uncomment the appropriate lines below:
' Your ECal module must already be connected
' via USB to the PNA.
'guidedCal.CalKitType(1) = "N4691-60004 ECal"
'guidedCal.CalKitType(2) = "N4691-60004 ECal"
' Non-factory characterizations are specified as follows:
'guidedCal.CalKitType(1) = "N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
'guidedCal.CalKitType(1) = "N4691-60004 ECal 01234"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'guidedCal.CalKitType(1) = "N4691-60004 MyDskChar ECal 01234"

MsgBox("Cal kits defined for Ports 1 and 2")

' Initiate the calibration and query the number of steps
numSteps = guidedCal.GenerateSteps
' Measure the standards, compute and apply the cal
MeasureAndComplete(numSteps)
End Sub

Sub FourPortGuidedCal()
' Select the connectors
guidedCal.ConnectorType(1) = "APC 3.5 female"
guidedCal.ConnectorType(2) = "APC 3.5 female"
guidedCal.ConnectorType(3) = "APC 3.5 female"
guidedCal.ConnectorType(4) = "APC 3.5 female"
' If a PNA which has more than 4 ports
For i = 5 To pna.NumberOfPorts
  guidedCal.ConnectorType(i) = "Not used"
Next
value = MsgBox("Connectors defined for Ports 1 to 4")
' Select the Cal Kit for each port being calibrated.
guidedCal.CalKitType(1) = "85052D"
guidedCal.CalKitType(2) = "85052D"
guidedCal.CalKitType(3) = "85052D"
guidedCal.CalKitType(4) = "85052D"
' To use an ECal module instead, comment out the above four lines
' and uncomment the following four lines.
' Replace N4691-60003 with your own ECAL model followed by 'Ecal'.
' Your ECal module must already be connected to a PNA USB port.
' See above for ECal options
'guidedCal.CalKitType(1) = "N4431-60003 ECal"
'guidedCal.CalKitType(2) = "N4431-60003 ECal"
'guidedCal.CalKitType(3) = "N4431-60003 ECal"
'guidedCal.CalKitType(4) = "N4431-60003 ECal"

value = MsgBox("Cal kits defined for Ports 1 to 4")
' Initiate the calibration

guidedCal.GenerateSteps

' If your selected cal kit is not a 4-port ECal module which can
' mate to all 4 ports at once, then you may want to choose which
'thru connections to measure for the cal. You must measure at
'least 3 different thru paths for a 4-port cal (for greatest
'accuracy you can choose to measure a thru connection for all 6
'pairings of the 4 ports). If you omit this command, the default
'is to measure from port 1 to port 2, port 1 to port 3, and
'port 1 to port 4. For this example we select to measure
'from port 1 to port 2, port 2 to port 3, and port 2 to port 4.

portList = Array(1,2,2,3,2,4)
guidedCal.ThruPortList = portList

' Re-generate the connection steps to account for the thru changes

numSteps = guidedCal.GenerateSteps

' Measure the standards, compute and apply the cal

MeasureAndComplete(numSteps)

End Sub

Sub MeasureAndComplete(ByVal numSteps)

value = MsgBox("Number of steps is " + CStr(numSteps))

' Measure the standards
'The following series of commands shows that standards
'can be measured in any order. These steps acquire
'measurement of standards in reverse order.
'It is easiest to iterate through standards using
'a For-Next Loop.

For i = NumSteps To 1

step = "Step " + CStr(i) + " of " + CStr(numSteps)
strPrompt = guidedCal.GetStepDescription(i)
value = MsgBox(strPrompt, vbOKOnly, step)
guidedCal.AcquireStep i

Next

' Conclude the calibration

guidedCal.GenerateErrorTerms

MsgBox ("Cal is done!")

End Sub

Last Modified:

5-Apr-2011   Edited for ECal options

20-Jan-2007   Added any order to steps.
Perform a Source Power Cal using COM

This program can be run in either Visual Basic 6 or as a VBScript program. The PNA can run *.vbs programs as macros.

This program demonstrates:

- Performing a source power calibration of Port 2 for Channel 1.
- Reading the calibration data.

Learn more about Power Calibrations
See an example that Uploads a Source Power Cal

---

See Other COM Example Programs

---

To run this program, you need:

- One of the following power meters connected to the PNA through GPIB: E4416A, E4417A, E4418A/B, E4419A/B, 437B, 438A, EPM-441A, EPM-442A

Note: If your power meter is other than these, you can create your own Power Meter Driver using our template.

- Your PC and PNA both connected to a LAN (for communicating with each other).

To make this program work in VBS, save the following code in a text editor file such as Notepad and save as *.vbs.

To make this program work in Visual Basic 6:

1. Create a new project
2. Click Project, Add New Module, click Open.
3. Paste the following code into the code window.
4. Delete the first two lines (comment and Main)
5. Click Project, Properties. Under Startup Object, select Sub Main
6. Click Project, References, and select the Agilent PNA Series Type Library.

```vbnet
' Run the Main subroutine
Main
Public Sub Main()
Dim PNA, chan, pwrcal ' PNA COM objects
Const naPowerMeter = 0, naPowerMeterAndReceiver = 1 ' enum NASourcePowerCalMethod
Const naPowerSensor_A = 0 ' enum NAPowerAcquisitionDevice
Const naCorrectionValues = 0 ' enum NASourcePowerCalBuffer
```
Const port = 2  ' PNA port #2 as source port
Const offset = 0  ' cal power offset value
Const bDisplay = True  ' whether to display data during acquire

Dim stimulus, calvalues, strResult

' Instantiate our PNA COM objects
Set PNA = CreateObject("AgilentPNA835x.Application")
Set chan = PNA.Channels(1)
Set pwrcal = PNA.SourcePowerCalibrator

' Set the number of sweep points to 21 on Channel 1.
chan.NumberOfPoints = 21

' Specify the GPIB address of the power meter  
' that will be used in performing the calibration.
pwrcal.PowerMeterGPIBAddress = 13

' Turn use of the loss table OFF (this assumes there is  
' virtually no loss in the RF path to the power sensor  
' due to a splitter, coupler or adapter).
pwrcal.UsePowerLossSegments = False

' Turn frequency checking OFF (so one power sensor is used for the entire cal  
' acquisition sweep regardless of frequency span).
pwrcal.UsePowerSensorFrequencyLimits = False

' Specify a nominal power accuracy tolerance (IterationsTolerance) in dB for the  
' calibration, and the maximum number of iterations to adjust power at each point,  
' attempting to achieve within tolerance of the desired power. If at any stimulus  
' point the power fails to reach within the set tolerance of the desired power  
' after the maximum number of iterations, the power at that point will be set to the  
' value determined by the last iteration (the Source Power Cal dialog box will  
' indicate the FAIL, but we can still apply the cal if desired when it's complete).  
' Each iteration is based upon a SETTLED power reading (see comments preceding the  
' next two properties below).
pwrcal.IterationsTolerance = 0.1
pwrcal.MaximumIterationsPerPoint = 3

' The worst-case window of power uncertainty (for a calibration which meets  
' tolerance) is the sum of the iteration tolerance and the power meter settling  
' tolerance (which is described below).
' At each stimulus point, the PNA takes power meter readings and determines when  
' they have settled by comparing the magnitude difference between consecutive  
' readings versus a nominal dB tolerance limit (ReadingsTolerance) on that magnitude  
' difference. When consecutive readings are within tolerance of each other, or  
' if they are not within tolerance but we've taken a maximum number of readings  
' (ReadingsPerPoint), the PNA does a weighted average of the readings taken at that  
' stimulus point and that is considered our settled power reading.
pwrcal.ReadingsTolerance = 0.1
pwrcal.ReadingsPerPoint = 5
Setup of information pertaining to this specific cal acquisition. Includes the method (type of devices) that will be used to perform the cal -- choose either naPowerMeter or naPowerMeterAndReceiver. naPowerMeterAndReceiver uses the power meter for the first iteration of each point and the PNA's reference receiver for subsequent iterations, so is much faster than using power meter only naPowerMeter.

But the power meter accounts for compression when calibrating at the output of an active device, whereas the reference receiver cannot unless it is coupled to the cal reference plane (on a PNA which allows direct access to the receivers).

'offset' specifies if the cal power level is offset (positive value for a gain, negative value for a loss) from the PNA port power setting on the channel when no source power cal is active. This is to account for components between the PNA test port and cal reference plane. In this example, we will calibrate at the PNA test port, so there is no offset (it is zero).

'bDisplay' indicates whether to display the source power cal dialog during the source power cal acquisition (the dialog will chart the corrected power readings).

```
pwrcal.SetCalInfo2 naPowerMeter, chan.channelNumber, port, offset, bDisplay
```

Perform synchronous source power cal acquisition sweep using the sensor attached to Channel A of the power meter. This assumes that the power sensor is already connected to Port 2 of the PNA.

```
pwrcal.AcquirePowerReadings naPowerSensor_A, True
```

Conclude the calibration. This applies the cal data to PNA channel memory, and turns the correction ON for Port 2 on Channel 1, but does NOT save the calibration.

```
pwrcal.ApplyPowerCorrectionValues
```

At this point, if you choose to save the instrument state as a ".CST" file, the calibration will be saved with the instrument state in that file.

```
stimulus = chan.GetXAxisValues
```

Read the stimulus values from Channel 1.

```
calvalues = chan.getSourcePowerCalDataEx(naCorrectionValues, port)
```

Read the source power correction data.

```
strResult = "Stimulus" & Chr(9) & Chr(9) & "Cal Value" & Chr(10)
For i = 0 To UBound(stimulus)
    strResult = strResult & stimulus(i) & Chr(9) & calvalues(i) & Chr(10)
Next
MsgBox strResult
End Sub
Perform an Unguided Cal using COM

This example uses the ICalibrator interface to do the following:

- perform a two port calibration
- retrieve the error term data
- retrieve the standard data (cal acquisition data)

Learn more about Reading and Writing Calibration data using COM.

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as Calibrate.vbs. Learn how to setup and run the macro.

dim pna
' To run from an external PC, substitute your PNA Name and use the following command.
' set pna = CreateObject("AgilentPNA835x.Application", "PNA Name")
set pna = CreateObject("AgilentPNA835x.Application")
dim calibrator
set calibrator=pna.activechannel.calibrator

wscript.echo "setcalinfo for two port cal"
calibrator.setcalinfo 5,1, 2

' only have one set of standards
calibrator.Simultaneous2PortAcquisition = false

'first acquire forward reflection standards, then reverse
dim p
for p = 1 to 2
if (p = 1) then
    calibrator.AcquisitionDirection = 0
else
    calibrator.AcquisitionDirection = 1
end if

wscript.echo "connect open to port ", p
calibrator.acquirecalstandard 1

wscript.echo "connect short to port ", p
calibrator.acquirecalstandard 2
wscript.echo "connect load to port ", p
calibrator.acquirecalstandard 3

next

wscript.echo "connect a thru1"
calibrator.acquirecalstandard 4

'Optional - perform isolation
wscript.echo "connect loads to both ports"
calibrator.acquirecalstandard 5

wscript.echo "calculating"
calibrator.CalculateErrorCoefficients

'Calibration complete
' Now read error terms and standard data

dim termName
termName = Array("Directivity", "SourceMatch", "ReflectionTracking")
dim vardata

' iterate over error terms
dim t
for t = 0 to 2 ' per error term
for p = 1 to 2 ' per port
wscript.echo "Requesting ", termName(t), p, p
vardata = calibrator.GetErrorTerm( t, p, p)
next
next

' now get the path terms: iterator each one request
termName = Array("Isolation", "LoadMatch", "TransmissionTracking")
for t = 0 to 2

wscript.echo "Requesting Forward term", termName(t), 1, 2
vardata = calibrator.GetErrorTerm( t, 1, 2)
wscript.echo "Requesting Reverse Term", termName(t), 2, 1
vardata = calibrator.GetErrorTerm( t, 2, 1)
next

dim stdname
stdname = Array("", "Open", "Short", "Load", "Thru", "Isolation")

' iterate over the port standards
for t = 1 to 3
for p = 1 to 2
' request the standard term for each port of interest
wscript.echo "Requesting", stdname(t), p, p
vardata = calibrator.GetStandard( t, p, p)
next
next

' now get the path standards: iterator each one request
for t = 4 to 5

wscript.echo "Requesting Forward",stdname(t),1,2
vardata = calibrator.GetStandard(t, 1,2)

wscript.echo "Requesting Reverse",stdname(t),2,1
vardata = calibrator.GetStandard(t, 2,1)

next
Perform an Unknown Thru or TRL Cal

The following program performs either a 2-port SOLT Unknown Thru Cal or a 2-port TRL Cal. The 85052C Cal Kit used in this program contains both types of standards. This program can be run on 2-port or 4-port PNAs. When run on a multiport (4 or more ports) PNA, which does not have a reference receiver per port, a Delta Match Cal is required. See example of Delta Match Cal.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unknown.vbs. Learn how to setup and run the macro.

```
Sub PerformUnknownThruOrTRLCal()
   ' Create / Get the PNA application.
   Set app = CreateObject("AgilentPNA835x.Application")
   ' Get the cal manager object
   Set calMgr = app.GetCalManager
   ' Get the guided cal object
   Set guidedCal = calMgr.GuidedCalibration
   Set chan = app.ActiveChannel
   chanNum = chan.ChannelNumber
   
   ' Initialize guided cal to be performed on the active channel.
   ' The boolean argument of True specifies the creation of a new calset
   ' for storing the new calibration.
   guidedCal.Initialize chanNum, True

   ' Specify connectors for Ports 1 and 2
   guidedCal.ConnectorType(1) = "APC 3.5 female"
   guidedCal.ConnectorType(2) = "APC 3.5 male"

   ' If your PNA has more than 2 ports, uncomment one or both of
   ' these next two lines, to explicitly specify this is
   ' just a 2-port cal.
   'guidedCal.ConnectorType(3) = "Not used"
   'guidedCal.ConnectorType(4) = "Not used"

   ' Specify cal kit for Ports 1 and 2
   guidedCal.CalKitType(1) = "85052C"
   guidedCal.CalKitType(2) = "85052C"

   ' Since the 85052C cal kit contains SOLT standards and also TRL
   ' standards, these next lines determine whether the cal becomes
   ' unknown thru (SOLT), or TRL.
   ' Specify cal and Thru method
   guidedCal.PathCalMethod (1,2) = "SOLT"
   guidedCal.PathThruMethod (1,2) = "Undefined Thru"

   ' To set up the cal as TRL, comment the previous line and uncomment this next line.
   ' The Thru method is set by default.
   guidedCal.PathCalMethod (1,2) = "TRL"
End Sub
```
' Initiate the calibration and query the number of steps
numSteps = guidedCal.GenerateSteps
MsgBox "Number of steps is " + CStr(numSteps)

' Query the list of ports that need delta match
portList = guidedCal.PortsNeedingDeltaMatch
If portList contains just one element and it's value is 0, then that indicates none of the ports being calibrated require delta match data.
' If each testport on the PNA has it's own reference receiver (R channel), then delta match is never needed, so portList will always be just 0.
lowerBound = LBound(portList)
If (UBound(portList) <> lowerBound) Or (portList(lowerBound) <> 0) Then
  ' Delta match data is required for at least one port.
  ' For this example, we assume a Global Delta Match Cal has previously been performed so the Global Delta Match CalSet exists.
  ' Supplying an empty string to ApplyDeltaMatchFromCalSet indicates to use the Global Delta Match CalSet.
  MsgBox "here"
guidedCal.ApplyDeltaMatchFromCalSet ""
End If

' Measure the standards
For i = 1 To numSteps
  step = "Step " + CStr(i) + " of " + CStr(numSteps)
  strPrompt = guidedCal.GetStepDescription(i)
  retVal = MsgBox(strPrompt, vbOKCancel, step)
  If retVal = vbCancel Then Exit Sub
  guidedCal.AcquireStep i
Next

' Conclude the calibration
guidedCal.GenerateErrorTerms
MsgBox "Cal is done!"

End Sub
Perform Global Delta Match Cal

The following program performs a Global Delta Match Calibration. This is required when performing an Unknown Thru cal or TRL cal on PNAs without a reference receiver for each test port. See example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Delta.vbs. Learn how to setup and run the macro.

```vbs
Sub PerformGlobalDeltaMatchCal()
    ' Create / Get the PNA application.
    Set app = CreateObject("AgilentPNA835x.Application")
    ' Get cal manager object
    Set calMgr = app.GetCalManager
    ' Get guided cal object
    Set guidedCal = calMgr.GuidedCalibration

    ' Initiate a Global Delta Match calibration, choosing connector and cal kit
    numSteps = guidedCal.GenerateGlobalDeltaMatchSequence("APC 3.5 female", "85033D/E")
    MsgBox "Number of steps is " + CStr(numSteps)

    ' Measure the standards
    For i = 1 To numSteps
        step = "Step " + CStr(i) + " of " + CStr(numSteps)
        strPrompt = guidedCal.GetStepDescription(i)
        retVal = MsgBox(strPrompt, vbOKCancel, step)
        If retVal = vbCancel Then Exit Sub
        guidedCal.AcquireStep i
    Next

    ' Conclude the calibration
    guidedCal.GenerateErrorTerms
    MsgBox "Cal is done!"

End Sub
```
Perform a Guided Cal with C#

The following example performs a 2-port or 4-port Guided Cal using C#.

**Note:** Replace `<remote host name>` with the full computer name of your PNA.

Learn more about using .NET with the PNA

```csharp
AgilentPNA835x.IApplication pna;
    AgilentPNA835x.ICalManager3 calMgr;
    AgilentPNA835x.IGuidedCalibration guidedCal;
    AgilentPNA835x.IChannel chan;
    int chanNum;
    AgilentPNA835x.ICalSet calset;
    void PerformGuidedCal()
    {
        Type pnaType = Type.GetTypeFromProgID("AgilentPNA835x.Application", "<remote host name>");
        pna = (AgilentPNA835x.IApplication)Activator.CreateInstance(pnaType);
        calMgr = (AgilentPNA835x.ICalManager3)pna.GetCalManager();
        guidedCal = (AgilentPNA835x.IGuidedCalibration)calMgr.GuidedCalibration;
        chan = pna.ActiveChannel;
        chanNum = chan.channelNumber;
        // Initialize guided cal to be performed on the active channel.
        // The boolean argument of True indicates to store the cal only
        // in the channel's calibration register. If instead you wish
        // to create a new calset that the new cal will get stored to,
        // comment out this next line and uncomment the three lines below it.
        guidedCal.Initialize(chanNum, true);
        calset = calMgr.CreateCalSet(chanNum);
        chan.SelectCalSet(calset.GetGUID(), true);
        guidedCal.Initialize(chanNum, false);

        // To perform 2-port cal, Uncomment the following
        // Then comment the 4-port cal
```
void TwoPortGuidedCal()
{
    // Select the connectors
    guidedCal.set_ConnectorType(1, "APC 3.5 female");
guidedCal.set_ConnectorType(2,"APC 3.5 male");
    for (int i = 3; i <= pna.NumberOfPorts; i++)
        guidedCal.set_ConnectorType(i, "Not used");
    MessageBox.Show("Connectors defined for Ports 1 and 2");
    // Select the Cal Kit for each port being calibrated.
guidedCal.set_CalKitType(1,"85052D");
guidedCal.set_CalKitType(2,"85052D");
    // To use an ECal module instead, comment out the above two lines
    // and uncomment the following two lines.
    // Replace N4691-60004 with your own ECAL model followed by 'ECal'
    // Your ECal module must already be connected to a PNA USB port.
    // guidedCal.CalKitType(1) = "N4691-60004 ECal"
    // guidedCal.CalKitType(2) = "N4691-60004 ECal"
    MessageBox.Show("Cal kits defined for Ports 1 and 2");
    // Initiate the calibration and query the number of steps
    int numSteps = guidedCal.GenerateSteps();
    // Measure the standards, compute and apply the cal
    MeasureAndComplete(numSteps);
}

void FourPortGuidedCal()
{
    //Select the connectors
    guidedCal.set_ConnectorType(1,"APC 3.5 female");
guidedCal.set_ConnectorType(2,"APC 3.5 female");
guidedCal.set_ConnectorType(3,"APC 3.5 female");
guidedCal.set_ConnectorType(4,"APC 3.5 female");

// If a PNA which has more than 4 ports
for (int i = 5;i<=pna.NumberOfPorts;++i)
    guidedCal.set_ConnectorType(i,"Not used");

MessageBox.Show("Connectors defined for Ports 1 to 4");

// Select the Cal Kit for each port being calibrated.
guidedCal.set_CalKitType(1,"85052D");
guidedCal.set_CalKitType(2,"85052D");
guidedCal.set_CalKitType(3,"85052D");
guidedCal.set_CalKitType(4, "85052D");

// To use an ECal module instead, comment out the above four lines
// and uncomment the following four lines.
// Replace N4691-60003 with your own ECAL model followed by 'ECal'.
// Your ECal module must already be connected to a PNA USB port.
//guidedCal.CalKitType(1) = "N4431-60003 ECal";
//guidedCal.CalKitType(2) = "N4431-60003 ECal";
//guidedCal.CalKitType(3) = "N4431-60003 ECal";
//guidedCal.CalKitType(4) = "N4431-60003 ECal";
MessageBox.Show("Cal kits defined for Ports 1 to 4");

// Initiate the calibration
guidedCal.GenerateSteps();

// If your selected cal kit is not a 4-port ECal module which can
// mate to all 4 ports at once, then you may want to choose which
// thru connections to measure for the cal. You must measure at
// least 3 different thru paths for a 4-port cal (for greatest
// accuracy you can choose to measure a thru connection for all 6
// pairings of the 4 ports). If you omit this command, the default
// is to measure from port 1 to port 2, port 1 to port 3, and
// port 1 to port 4. For this example we select to measure
// from port 1 to port 2, port 2 to port 3, and port 2 to port 4.
long[] portList = new long[6]{1,2,2,3,2,4};
guidedCal.ThruPortList = portList;
// Re-generate the connection steps to account for the thru changes
int numSteps = guidedCal.GenerateSteps();
MeasureAndComplete(numSteps);

}

void MeasureAndComplete(int numSteps)
{
    MessageBox.Show("Number of steps is " + numSteps.ToString());

    // Measure the standards
    // The following series of commands shows that standards
    // can be measured in any order. These steps acquire
    // measurement of standards in reverse order.
    // It is easiest to iterate through standards using
    // a For-Next Loop.
    for (int i = numSteps; i >= 1; --i)
    {
        string strPrompt = guidedCal.GetStepDescription(i);
        MessageBox.Show(strPrompt);
        guidedCal.AcquireStep(i);
    }

    // Conclude the calibration
    guidedCal.GenerateErrorTerms();
    MessageBox.Show("Cal is done");
}
Perform an ECal using COM

This example uses the GuidedCalibration interface to perform a 2-port ECal calibration. Learn more about Reading and Writing Calibration data using COM.

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as ECal.vbs. Learn how to setup and run the macro.

```vbs
Set pna = CreateObject("AgilentPNA835x.Application")
Set calMgr = pna.GetCalManager
Set guidedCal = calMgr.GuidedCalibration
Set chan = pna.ActiveChannel
chanNum = chan.ChannelNumber
' Initialize guided cal to be performed on the active channel.
' The boolean argument of True indicates to create a new calset
guidedCal.Initialize chanNum, True
' To perform 3-port cal, Uncomment the following
' Then comment the 2-port cal

' Do 2-port cal
TwoPortGuidedCal

' Do 3-port cal
' ThreePortGuidedCal

Sub TwoPortGuidedCal()
'Change the following to match the connectors on your ECal module
guidedCal.ConnectorType(1) = "APC 3.5 female"
guidedCal.ConnectorType(2) = "APC 3.5 female"
For i = 3 To pna.NumberOfPorts
guidedCal.ConnectorType(i) = "Not used"
Next
value = MsgBox("Connectors defined for Ports 1 and 2")

' Select the ECal module for each port being calibrated.
' Replace N4691-60004 with your own ECAL model followed by 'ECal'.
' Your ECal module must already be connected
' via USB to the PNA.
guidedCal.CalKitType(1) = "N4691-60004 ECal"
guidedCal.CalKitType(2) = "N4691-60004 ECal"
' Non-factory characterizations are specified as follows:
```
'guidedCal.CalKitType(1) = "N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
'guidedCal.CalKitType(1) = "N4691-60004 ECal 01234"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'guidedCal.CalKitType(1) = "N4691-60004 MyDskChar ECal 01234"
' Turn on auto orientation for the ECal (default behavior).
'guidedCal.AutoOrient = 1'
MsgBox("Cal kits defined for Ports 1 and 2")

' Initiate the calibration and query the number of steps
numSteps = guidedCal.GenerateSteps
' Measure the standards, compute and apply the cal
MeasureAndComplete(numSteps)
End Sub

Sub ThreePortGuidedCal()
' 'Change the following to match the connectors on your ECal module
guidedCal.ConnectorType(1) = "APC 3.5 female"
guidedCal.ConnectorType(2) = "APC 3.5 female"
guidedCal.ConnectorType(3) = "APC 3.5 female"

' Select the ECal module for each port being calibrated.
' Replace N4691-60003 with your own ECAL model followed by 'ECal'.
' Your ECal module must already be connected to a PNA USB port.
guidedCal.CalKitType(1) = "N4431-60003 ECal"
guidedCal.CalKitType(2) = "N4431-60003 ECal"
guidedCal.CalKitType(3) = "N4431-60003 ECal"

value = MsgBox("Cal kits defined for Ports 1 to 3")
' Initiate the calibration
numSteps = guidedCal.GenerateSteps
' Measure the standards, compute and apply the cal
MeasureAndComplete(numSteps)
End Sub

Sub MeasureAndComplete(ByVal numSteps)
value = MsgBox("Number of steps is " + CStr(numSteps))
' Measure the standards
For i = 1 To numSteps
step = "Step " + CStr(i) + " of " + CStr(numSteps)
strPrompt = guidedCal.GetStepDescription(i)
value = MsgBox(strPrompt, vbOKOnly, step)
guidedCal.AcquireStep i
Next
' Conclude the calibration
guidedCal.GenerateErrorTerms
MsgBox ("Cal is done!")
End Sub
Perform an ECal User Characterization

This example performs a user-characterization and stores it to both the ECal module memory and PNA disk memory.

It then performs two 2-port cals: the first using the characterization from module memory, then using the characterization from disk memory.

**Note:** This example requires that channel 1 be already calibrated.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as ECal.vbs.

**See Also**

How to setup and run the macro.

ECalUserCharacterizer Object

About User Characterization

---

```vbs
Option Explicit
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Dim calMgr
Set calMgr = pna.GetCalManager

' Get ECal User Characterizer COM object
Dim ecalCharacterizer
Set ecalCharacterizer = calMgr.GetECalUserCharacterizer

' Substitute here the model number and serial number of your own ECal.
' Note that this example corresponds to a 4-port ECal module with
' serial number 00001. If instead you have a 2-port ECal module,
' their model numbers are '5x5' numbers -- for example, 'N4691-60001'.
Dim ecalModelNum
ecalModelNum = "N4433A"
Dim ecalSerialNum
ecalSerialNum = "00001"
ecalCharacterizer.ECalID = ecalModelNum & "," & ecalSerialNum
MsgBox "ECal module to be characterized is: " & ecalCharacterizer.ECalID

' Set which user characterization number (1-12) the new characterization
' will be stored to in the ECal module when it is done. If you intend to
```
' store your user characterization just to PNA Disk Memory and NOT the 
' ECal module's memory, then omit the setting of this property.

calCharacterizer.CharacterizationNumber = 1

' The following commented-out lines of code show how you can access 
' the list of connector type names you can set for the ports of an 
' ECal when you user-characterize it. However, please note that if 
' you are writing the user characterization to the ECal module's memory, 
' as of yet only the Factory Defined set of connector choices will work 
' properly (see the ValidConnectorType property).

' If you will be saving your characterization to just 
' PNA Disk Memory only, then all connector names returned by this call 
' will work, user-defined connector names as well as factory-defined.

'Dim connTypeArray
'connTypeArray = calCharacterizer.ValidConnectorTypes
'MsgBox connTypeArray(1)

' Access element 1 in the string array
' For each port of the ECal module, specify which connector type 
' is at the end of the adapter (or cable or fixture) that is 
' connected to that port of the ECal for the characterization 
' (must be one of the connector types that is included in the 
' list that the ValidConnectorTypes method returns). The default 
' is "No adapter", which assumes you are characterizing that port 
' of the ECal "as is" (nothing attached to it). So in this example, 
' Ports C and D of the ECal are being characterized to just the 
' ECal's connectors.

calCharacterizer.ConnectorType(1) = "APC 3.5 male" ' ECal Port A
calCharacterizer.ConnectorType(2) = "APC 3.5 male" ' ECal Port B

' As with the connector types, the information set in these next 
' few properties also gets stored within the characterization.
' Set the name of the person and/or company that is producing 
' this characterization.

calCharacterizer.UserName = "John Doe, Acme Inc."

' Set user-specified description of the PNA being used.

calCharacterizer.UserDescriptionOfPNA = "SN US12345678"

' Set descriptions of what you have connected to the ECal module's 
' ports for the characterization.
ecalCharacterizer.PortDescription(1) = "3.5 mm adapter, SN 00001" ' Port A of the ECal
ecalCharacterizer.PortDescription(2) = "3.5 mm adapter, SN 00002" ' Port B of the ECal
' Begin a user characterization on Channel 1.
' If you will be storing this characterization to the ECal module's memory, then
' the boolean argument to this command should be set to True. If you will be storing
' this characterization to PNA disk memory ONLY, then you should specify False for
' that argument. In this example we will be storing the characterization to both
' module memory and PNA disk memory, so we use True.
ecalCharacterizer.InitializeEx 1, True
' Generate the measurement steps for the user characterization.
Dim numSteps
numSteps = ecalCharacterizer.GenerateSteps
' Measure the steps.
' You must ensure you have already applied the appropriate calibration to the channel
' already, or else an error will be thrown indicating that.
Dim i
For i = 1 To numSteps
    MsgBox ecalCharacterizer.GetStepDescription(i)
    ecalCharacterizer.AcquireStep(i)
    MsgBox "Acquire is complete"
Next
MsgBox "Now the user characterization will be saved to the ECal module and to PNA disk memory"
' Save the user characterization to the ECal module's memory.
' Note that this can take multiple minutes, depending on how many sweep points the channel has.
ecalCharacterizer.SaveToECal
' Save the user characterization to PNA Disk Memory.
Dim characterizationName
characterizationName = "test"
ecalCharacterizer.SaveToDiskMemory(characterizationName)
MsgBox "User characterization is complete. Now we will calibrate using it. First we will use it from ECal module memory."
Dim moduleMemCalKitName

moduleMemCalKitName = GetCalKitName("User " & CStr(ecalCharacterizer.CharacterizationNumber))

DoTwoPortCal moduleMemCalKitName

MsgBox "Now we will calibrate using the characterization from PNA Disk Memory."

Dim pnaDiskMemCalKitName

pnaDiskMemCalKitName = GetCalKitName(characterizationName)

DoTwoPortCal pnaDiskMemCalKitName

MsgBox "Example has completed"

Function GetCalKitName(characterizationName)

Dim calKitName

calKitName = ecalModelNum

If Len(characterizationName) > 0 Then calKitName = calKitName & " " & characterizationName

calKitName = calKitName & " ECal " & ecalSerialNum

GetCalKitName = calKitName

End Function

Sub DoTwoPortCal(calKitName)

' Initialize guided cal to be performed on Channel 1.

Dim guidedCal

Set guidedCal = calMgr.GuidedCalibration

guidedCal.Initialize 1, True

' Specify the DUT connector for each PNA port to be calibrated (DUT connector = ECal characterization's connector)

guidedCal.ConnectorType(1) = "APC 3.5 male"

guidedCal.ConnectorType(2) = "APC 3.5 male"

' Specify the "cal kit" for each of those ports

guidedCal.CalKitType(1) = calKitName

guidedCal.CalKitType(2) = calKitName

' We know this example will result in a calibration sequence of a single "connection step"

Dim numSteps

numSteps = guidedCal.GenerateSteps

' Acquire the cal connection step

guidedCal.AcquireStep 1
' Conclude the cal and turn it on
guidedCal.GenerateErrorTerms
End Sub
Perform a Comprehensive 2-Port Guided Cal

This example program performs a Guided Calibration on the active channel between ports 1 and 2. The following calibration features are demonstrated:

- Guided Power Cal
- Optional functions when using ECal
- Select the Thru method
- Save to a new CalSet

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as CompGuided.vbs. Learn How to setup and run the macro.

**See Also**
- PNA Object Model
- CalManager Object
- GuidedCalibration Object
- Calibrator Object

**See Other COM Example Programs**

```vbscript
' Performing a Guided 2-port cal (Ports 1 and 2)
TwoPortGuidedCal
Sub TwoPortGuidedCal
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set calMgr = app.GetCalManager
Set guidedCal = calMgr.GuidedCalibration
Set chan = app.ActiveChannel
chanNum = chan.ChannelNumber
' Initialize guided cal to be performed on the active channel.
' The boolean argument of True indicates to create a new calset
' for storing the new calibration to.
guidedCal.Initialize chanNum, True
' Query the connectors that the PNA system recognizes
conns = guidedCal.ValidConnectorTypes
```
'Format the list string with linefeed characters between each substring
connList = FormatList(conns)

'Select the connector for Port 1
selectedConn1 = InputBox("Enter your DUT connector for Port 1. Choose from this list:" & _
                      Chr(10) & Chr(10) & connList)
If selectedConn1 = "" Then Exit Sub
guidedCal.ConnectorType(1) = selectedConn1

'Select the connector for Port 2
selectedConn2 = InputBox("Enter your DUT connector for Port 2. Again, choose from this list:" & _
                        Chr(10) & Chr(10) & connList)
If selectedConn2 = "" Then Exit Sub
guidedCal.ConnectorType(2) = selectedConn2

'Note: If your PNA has more than 2 ports, you would need to uncomment one or both of these next two lines, to explicitly specify this is just a 2-port cal.
'guidedCal.ConnectorType(3) = "Not used"
'guidedCal.ConnectorType(4) = "Not used"

'Query the list of acceptable cal kits and ECal module characterizations for Port 1.
kits = guidedCal.GetCompatibleCalKits(selectedConn1)

'Format the list string with linefeed characters between each substring
kitList = FormatList(kits)

'Select the Cal Kit or ECal module characterization to use for Port 1.
selectedKit = InputBox("Enter your cal kit or ECal module characterization for Port 1. " & _
                      "Choose from this list:" & Chr(10) & Chr(10) & kitList)
If selectedKit = "" Then Exit Sub
guidedCal.CalKitType(1) = selectedKit

'Query the list of acceptable cal kits and ECal module characterizations for Port 2.
kits = guidedCal.GetCompatibleCalKits(selectedConn2)

'Format the list string with linefeed characters between each substring
kitList = FormatList(kits)

'Select the Cal Kit or ECal module characterization to use for Port 2.
selectedKit = InputBox("Enter your cal kit or ECal module characterization for Port 2. " & _
"Choose from this list:" & Chr(10) & Chr(10) & kitList)
If selectedKit = "" Then Exit Sub
guidedCal.CalkitType(2) = selectedKit
' This determines whether the cal will be a "Guided Power Cal"
' or just a traditional S-parameter cal.
message = "On which port number shall power be measured?  
message = message & "For a traditional guided cal without power cal, enter 0"
Dim powerPort
powerPort = CInt( InputBox(message) )
If powerPort > 0 Then
    guidedCal.PerformPowerCalibration(powerPort) = True
    Dim retVal
    retVal = MsgBox("Is the power sensor's connector type or gender different from 
    retVal = vbYesNo)
    If retVal = vbYes Then
        message = "Enter your power sensor's connector.  Choose from this list:" 
        message = message & Chr(10) & Chr(10) & connList
        ' Select the sensor's connector.
        selectedConn1 = InputBox(message)
        If selectedConn1 = "" Then Exit Sub
guidedCal.PowerSensorConnectorType(powerPort) = selectedConn1
        ' Query the list of acceptable cal kits and ECal module characterizations
        ' that are applicable for the sensor's connector.
        kits = guidedCal.GetCompatibleCalKits(selectedConn1)
        ' Format the list string with linefeed characters between each substring
        kitList = FormatList(kits)
        message = "Enter your cal kit or ECal module characterization to use for 
        message = message & "Choose from this list:" 
        message = message & Chr(10) & Chr(10) & kitList
        ' Select the Cal Kit or ECal module characterization to use for de-embed of 
        selectedKit = InputBox(message)
        If selectedKit = "" Then Exit Sub
guidedCal.PowerSensorCalkitType(powerPort) = selectedKit
    Else
        guidedCal.PowerSensorConnectorType(powerPort) = "Ignored"
End If ' End of block that considers the sensor's connector

' Ask for the power level to perform the power cal at
' (if this command is omitted, the default is 0 dBm).
Dim powerLevel

powerLevel = InputBox("Enter the power level for the power cal to be performed at")

If powerLevel = "" Then Exit Sub

guidedCal.PowerCalibrationPowerLevel(powerPort) = CDbl(powerLevel)
Else

guidedCal.PerformPowerCalibration(1) = False
End If ' End of block that considers if the cal will include power calibration

'--------------------------------------------------------------------------------
---

' This next block of commented-out code shows optional functions when using ECal.
' These OrientECALModule and ECALPortMapEx properties would need to be set prior to
' calling GenerateSteps on the guidedCal object.
' Read the information about the Agilent factory characterization data
' of ECal module #1 on the USB bus
'Set calibrator = chan.Calibrator
'Const ECalModule1 = 1
'module1Info = calibrator.GetECALModuleInfoEx(ECalModule1)
'MsgBox "Description of ECal module #1:" & Chr(10) & Chr(10) & module1Info
' By default, during calibration the PNA automatically determines the orientation of
' the ECal module (senses which port of the module is connected to which port of
' the PNA).
' However, since this setting could have recently been overridden by another user of
' the instrument, use this next line to ensure the auto orientation setting is enabled.
'calibrator.OrientECALModule = True
' Alternatively, if you are measuring at very low power levels where
' the PNA fails to sense the module's orientation, you may need to turn off the auto
' orientation and specify how the module is connected (as in these next two lines of code,
' "A1,B2" would indicate Port A of the module is connected to Port 1 and
' Port B is connected to Port 2).
'calibrator.OrientECALModule = False
'calibrator.ECALPortMapEx(ECalModule1) = "A1,B2"
' End of optional ECal setup
'-------------------------------------------------------------------------------------------------------------------
---

' Select the thru method of Default. This instructs the PNA to determine which thru
' standard measurement technique to use, based upon the selected connectors and
' calibration kit(s) and what model of PNA this is.
guidedCal.ThruCalMethod = 0 ' 0 = naDefaultCalMethod
' Initiate the calibration and query the number of steps
numSteps = guidedCal.GenerateSteps
MsgBox "Number of steps is " + CStr(numSteps)
' Measure the standards
For i = 1 To numSteps
    step = "Step " + CStr(i) + " of " + CStr(numSteps)
    strPrompt = guidedCal.GetStepDescription(i)
    MsgBox strPrompt, vbOKOnly, step
    guidedCal.AcquireStep i
Next
' Conclude the calibration
guidedCal.GenerateErrorTerms
MsgBox "Cal is done!"

End Sub

Function FormatList(tokens)
    For i = 0 To UBound(tokens)
        list = list & tokens(i) & Chr(10)
    Next
    FormatList = list
End Function

Last Modified:
8-Oct-2010    MX New topic
**ECAL Confidence Check**

This Visual Basic program:

- Initializes the PNA objects.
- Performs a complete ECAL confidence check

Before using this code:

- The active channel must contain an S11 measurement with a 1-port or N-port calibration
- Prepare a form with two buttons named `cmdRun` and `cmdQuit`

**Note:** A confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

```vbnet
Private oPNA As AgilentPNA835x.Application
Private oChan As Channel
Private oCal As Calibrator
Private oMeas As Measurement

Private Sub cmdRun_Click()
    Dim iMeasIndex As Integer
    Set oPNA = CreateObject("AgilentPNA835x.Application", "MachineName")
    Set oChan = oPNA.ActiveChannel
    Set oCal = oChan.Calibrator

    iMeasIndex = 1

    ' Loop through measurements until an S11 on the active channel
    ' is found, or the end of the measurement collection is reached.
    Do
        Set oMeas = oPNA.Measurements(iMeasIndex)
        If oMeas.Parameter = "S11" And _
            oMeas.channelNumber = oChan.channelNumber Then Exit Do
        iMeasIndex = iMeasIndex + 1
        If iMeasIndex > oPNA.Measurements.Count Then
            MsgBox "No S11 measurement found on the active channel." _
                & " Create an S11 measurement, then try again."
    Loop
```

2097
Exit Sub
End If
Loop

' Set up trace view so we are viewing only the data trace.
oMeas.View = naData

' Acquire the S11 confidence check data from ECal Module A
' into the memory buffer.
oCal.AcquireCalConfidenceCheckECALEx "S11", 1

' Turn on trace math so the trace shows data divided by memory.
' You can be confident the S11 calibration is reasonably good if
' the displayed trace varies no more than a few tenths of a dB
' from 0 dB across the entire span.
oMeas.TraceMath = naDataDivMemory
End Sub

Sub cmdQuit_Click()

' Turn off trace math
' in case someone clicks Quit without having clicked Run
If oMeas <> Nothing Then oMeas.TraceMath = naDataNormal

' Conclude the confidence check to set the ECal module
' back to it's idle state.
If oCal <> Nothing Then oCal.DoneCalConfidenceCheckECAL

' End the program
End
End Sub
Writing Cal Set Data using COM

This example creates a Cal Set and then writes data to the Cal Set.

Learn more about Reading and Writing Calibration data using COM.
Learn more about Cal Sets.
See example: Reading Calset Data

---

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as CalSetsWrite.vbs. Learn how to setup and run the macro.

```vbs
' This sub creates phony data
Sub InitPhonyData()
    Dim i
    Dim numpts
    WScript.Echo "init phony"
    numpts = PNA.ActiveChannel.NumberOfPoints
    ReDim v(numpts - 1, 1)
    For i = 0 To numpts - 1
        v(i, 0) = i
        v(i, 1) = 0
    Next
End Sub

' This sub creates a Cal Set, then writes the phony data to it
Sub PutPhonyData()
    Dim cmgr
    Dim cset
    WScript.Echo "putphony"
    Set cmgr = PNA.GetCalManager
    Set cset = cmgr.CreateCalSet(1)
    Cset.OpenCalSet naCalType_OnePort, 1
    const directivity = 0
    const sourcematch = 1
    const reflectiontracking = 2
    cset.putErrorTerm directivity, 1, 1, v
    cset.putErrorTerm sourcematch, 1, 1, v
    cset.putErrorTerm reflectiontracking, 1, 1, v
    cset.CloseCalSet
```

2099
cset.Description = "Phony One Port"
cset.save
End Sub
Upload a Source Power Cal using COM

This program can be run in either Visual Basic 6 or as a VBScript program. The PNA can run *.vbs programs as macros.

This program demonstrates:

- Uploading a source power calibration of Port 2 for Channel 1.
- Reading the calibration data.

Learn more about Power Calibrations

See Other COM Example Programs

To run this program you need:

- Your PC and PNA both connected to a LAN (for communicating with each other).

To make this program work in VBS, save the following code in a text editor file such as Notepad and save as *.vbs.

To make this program work in Visual Basic 6:

1. Create a new project
2. Click Project, Add New Module, click Open.
3. Paste the following code into the code window.
4. Delete the first two lines (comment and Main)
5. Click Project, Properties. Under Startup Object, select Sub Main
6. Click Project, References, and select the Agilent PNA Series Type Library.

```vbs
Public Sub Main()
    Dim PNA, chan ' PNA COM objects
    Const naCorrectionValues = 0 ' enum NASourcePowerCalBuffer
    Const port = 2 ' PNA port #2 as source port
    Dim stimulus, calvalues
    Dim power, calpower, strResult
    ' Instantiate our PNA COM objects
    Set PNA = CreateObject("AgilentPNA835x.Application")
    Set chan = PNA.Channels(1)
    ' Code for uploading source power calibration...
    ' Code for reading calibration data...
End Sub
```
' Set the number of sweep points to 2 on Channel 1.
chan.NumberOfPoints = 2

' Ensure there's currently no source power cal on for this channel and port.
chan.SourcePowerCorrection(port) = False

' Specify if the cal power level is offset (positive value for a gain, negative value for a loss) from the PNA port power setting on the channel when no source power cal is active. This is to account for components between the PNA test port and cal reference plane.
' In this example, let's set up our calibration at the output of an amplifier with 15 dB gain.
chan.SourcePowerCalPowerOffset(port) = 15

' Send our source power correction data to the PNA. For purpose of simplicity in this example, we'll set up for no correction (0) at our start stimulus and 0.5 dB at our stop stimulus (recall that our sweep currently has just 2 points).
calvalues = Array(0, 0.5)
chan.putSourcePowerCalDataEx naCorrectionValues, port, calvalues

' Set the number of sweep points to 21 on Channel 1.
chan.NumberOfPoints = 21

' Read the fixed power level for this port on Channel 1.
power = chan.TestPortPower(port)

' Turn the source power cal on.
chan.SourcePowerCorrection(port) = True

' Again read the fixed power level for this port on Channel 1 (with our calibration turned on, this should now include the 15 dB offset we indicated our power amplifier provides).
calpower = chan.TestPortPower(port)

' Read the stimulus values from Channel 1.
stimulus = chan.GetXAxisValues

' Read back the source power correction data, now interpolated for 21 points
calvalues = chan.getSourcePowerCalDataEx(naCorrectionValues, port)

' Print the data using a message box (here, Chr returns the ASCII characters for Tab (9) and Linefeed (10)).
strResult = "PNA port power = " & power & Chr(10)
strResult = strResult & "Power at reference plane = " & calpower & Chr(10) & hr(10)
strResult = strResult & "Stimulus" & Chr(9) & Chr(9) & "Cal Value" & Chr(10)
For i = 0 To UBound(stimulus)
  strResult = strResult & stimulus(i) & Chr(9) & calvalues(i) & Chr(10)
Next
MsgBox strResult
End Sub
Upload and Download Segment Table

These example programs use the `SetAllSegments_Method` and `GetAllSegments Method` to do the following:

- Creates a 2-dimensional array (7 x 10) 7 data elements that define each segment x 10 segments
- Uploads the data to the PNA
- **Downloads a segment table** from the PNA

This program does not make sweep type = segment or show the segment table. The comments indicate the order in which the segment elements are specified: Index 0 - segment state, Index 4 is IFBW, and so forth.

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as *.vbs. Learn how to setup and run the macro.

```vbs
' Create the application instance, and preset the application
Set app = CreateObject("AgilentPNA835x.Application")
app.Preset

Dim chan
Set chan = app.ActiveChannel
chan.sweeptype = 4

Dim segs
Set segs = chan.Segments

Dim win
Set win = app.NAWindows(1)
win.ShowTable 2

' Multipliers
kHz = 1000
MHz = kHz*1000
GHz = MHz*1000
' Create segments from 10MHz to 3GHz
StartFreq = 10 * MHz
StopFreq = 3 * GHz
'*
'* Create 10 segments between StartFreq and StopFreq
'*
' Create a 2-D array of segments.
```
Dim segdata(6, 9)

' Width of frequency segment, used below
SegmentWidth = (StopFreq - StartFreq) / 10

' Fill up all 10 segments (indices 0 to 9) with data
For i = 0 To 9
' element 0=segment state (on or off)
segdata(0, i) = True

' element 1=Num Points in this segment
segdata(1, i) = 500

' element 2=Start Freq
segdata(2, i) = StartFreq + i * SegmentWidth

' element 3=Stop Freq
segdata(3, i) = segdata(2, i) + SegmentWidth

' element 4=IFBW
segdata(4, i) = 35000

' element 5=Dwell Time
segdata(5, i) = 0

' element 6=Power
segdata(6, i) = 0

Next

' Configure Independent segment settings
segs.IFBandwidthOption = 1
segs.SourcePowerOption = 1

' Push the segment data into the PNA's Active Channel
segs.SetAllSegments segdata

Option Explicit
Dim app
Set app = CreateObject("AgilentPNA835x.Application")
Dim chan
Set chan = app.ActiveChannel
chan.sweeptype = 4
Dim segs
Set segs = chan.Segments
Dim win
Set win = app.NAWindows(1)
win.ShowTable 2
Dim segData
segData = segs.GetAllSegments
' Get lower bound and upper bound on the data values per each segment
Dim segDataLB, segDataUB
segDataLB = LBound(segData,1)
segDataUB = UBound(segData,1)
' Get lower bound and upper bound corresponding to how many segments
Dim segArrayLB, segArrayUB
segArrayLB = LBound(segData,2)
segArrayUB = UBound(segData,2)
' If the VB LBound and UBound functions didn't generate an error
' before reaching this point, that implies a valid two-dimensional
' array was returned into 'segData'.
WScript.Echo "Number of segments = " & segArrayUB - segArrayLB + 1
WScript.Echo "Number of data values per segment = " & segDataUB - segDataLB + 1
Dim index
index = segDataLB
Dim segInfStr
segInfStr = "Segment 1: state = " & segData(index, segArrayLB)
index = index + 1
segInfStr = segInfStr & ", num points = " & segData(index, segArrayLB)
index = index + 1
segInfStr = segInfStr & ", start freq = " & segData(index, segArrayLB)
index = index + 1
segInfStr = segInfStr & ", stop freq = " & segData(index, segArrayLB)
index = index + 1
segInfStr = segInfStr & ", IFBW = " & segData(index, segArrayLB)
index = index + 1
segInfStr = segInfStr & ", dwell time = " & segData(index, segArrayLB)
' In case of a measurement receiver PNA like N5264A
' which has no source ports, chan.SourcePortNames will
' return an empty variant (no array)
Dim srcPortNames
srcPortNames = chan.SourcePortNames
Dim srcPortNamesLB, srcPortNamesUB
srcPortNamesUB = -1
On Error Resume Next
srcPortNamesLB = LBound(srcPortNames)
srcPortNamesUB = UBound(srcPortNames)
On Error GoTo 0
If (srcPortNamesUB >= 0) And ((srcPortNamesUB - srcPortNamesLB + 1) <> (segDataUB - index)) Then
    WScript.Echo "Mismatch in number of source port names!"
End If
Dim j
For j = index + 1 To segDataUB
    segInfStr = segInfStr & ", " & srcPortNames(j - (index + 1) + srcPortNamesLB) & " power = " & segData(j, segArrayLB)
Next
WScript.Echo segInfStr

Last Modified:

28-Apr-2009   Added Download example
Create and Cal an SMC Measurement

This example creates and calibrates an SMC measurement. To run this example without modification you need the following:

- A Mixer setup file saved on the PNA: C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr.
- If the mixer file uses an external LO source, it must be connected and configured.
- An ECal module that covers the frequency range of the measurement.
- A power meter must be available to the PNA. This can be accomplished either by attaching the meter to the PNA via a GPIB cable, or by using SCPI over LAN.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as SMC.vbs. Learn how to setup and run the macro.

See Also
Create an SMC Fixed Output Meas
Use Existing Power Cal for SMC

```
Dim App
Set App = CreateObject("AgilentPNA835x.Application")
App.Preset
Dim Meas
Set Meas = App.ActiveMeasurement
Meas.Delete
App.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","SC21"
'Other valid strings that can be specified to create a measurement with a parameter
'other than"SC21" are: "S11", "S22", "IPwr", and "OPwr"
Set Meas = App.ActiveMeasurement
'You can perform mixer setup here or
'recall a previous mixer setup from the PNA Hard drive.
'This is how the mixer could be configured through the IMixer interface
Dim mix
Set mix = Meas  ' reference to IMixer object
mix.ActiveXAxisRange = 0  ' 0 = mixINPUT (Input frequency range)
' Alternatively, recall a previous mixer setup from the PNA Hard drive
Meas.LoadFile "C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr"
```
app.activechannel.numberofpoints = 21
Dim CalMgr
Set CalMgr = App.GetCalManager
Dim SMC
Set SMC = CalMgr.CreateCustomCal("SMC")
SMC.Initialize 1, 1
SMC.ConnectorType(1) = "APC 3.5 male"
SMC.ConnectorType(2) = "APC 3.5 female"
' Use Mechanical cal kits
SMC.CalKitType(1) = "85033D/E"
SMC.CalKitType(2) = "85033D/E"
' To use an ECal module instead, comment out the above two lines
' and uncomment the appropriate lines below:
' Your ECal module must already be connected
' via USB to the PNA.
' SMC.CalKitType(1) = "N4691-60004 ECal"
' SMC.CalKitType(2) = "N4691-60004 ECal"
' Non-factory characterizations are specified as follows:
' SMC.CalKitType(1) = "N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
' SMC.CalKitType(1) = "N4691-60004 ECal 01234"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
' SMC.CalKitType(1) = "N4691-60004 MyDskChar ECal 01234"
' Turn on auto orientation for the ECal (default behavior).
' SMC.AutoOrient = 1'
MsgBox("Cal kits defined for Ports 1 and 2")
' Omit the isolation part of the 2-port cal (default behavior).
SMC.OmitIsolation = 1
Dim steps
steps = SMC.GenerateSteps
For i = 1 To steps
    MsgBox SMC.GetStepDescription(i)
    SMC.AcquireStep i
Dim calset

calset = SMC.GenerateErrorTerms

Msgbox("SMC Cal Complete!")

Last Modified:

5-Apr-2011   Edited for ECal options

8-Mar-2011   Updated for A.09.33
Create and Cal a VMC Measurement

The following example program sets up a 1-stage mixer, then performs a VMC calibration using an N4691-60004 ECAL module.

By removing the comments ( ` ) at the start of the BLUE code, it can also do the following:

- Use a mechanical cal kit
- Perform manual ECAL orientation
- Load a Mixer Characterization
- Perform a Mixer Characterization ONLY

See Also
Converter Object
VMCType Object

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as VMC.vbs. Learn how to setup and run the macro.

dim NASWEPT: NASWEPT = 0
dim NAFIXED: NAFIXED = 1
dim LOWSIDE: LOWSIDE = 0
dim HIGHSIDE: HIGHSIDE = 1
dim MIXEROUT: MIXEROUT = 2
dim pna: set pna = CreateObject("AgilentPNA835x.application")
pna.reset
' Create a VMC channel
' Other valid measurement strings are: "S11", and "S22"
pna.CreateCustomMeasurementEx 1, "Vector Mixer/Converter","VC21",1
' Setup Stimulus
dim chan: set chan = pna.activechannel
dim cv: set cv = chan.Converter
chan.NumberOfPoints = 11
chan.IFBandwidth = 1000
cv.InputStartFrequency = 3.6e9
cv.InputStopFrequency = 3.9e9
cv.LOFixedFrequency(1) = 1e9
cv.LOPower(1) = 10
cv.OutputSideband = LOWSIDE
cv.Calculate MIXEROUT
cv.LOName(1) =  "Port 3"
cv.Apply()
DoBasicVMCCal (chan.channelNumber)

sub DoBasicVMCCal( channel )
dim myMixerCharFile: myMixerCharFile = "C:\Program Files\Agilent\Network Analyzer\Documents\MyMixerS2P.s2p"

' construct a VMC calibration object
dim calmanager: set calmanager = pna.GetCalManager
dim guidedCal: set guidedCal = calmanager.CreateCustomCalEx( channel )
dim vmc: set vmc = guidedCal.CustomCalConfiguration

' Initialize the cal object.
' Choose to respect or ignore the Preference: Cal: Auto Save to User Calset
' if you set this true, the behavior will be dependent on the setting
' of the preference.
dim useCalSetPreference: useCalSetPreference = false
vmc.Initialize channel, useCalSetPreference

' Define the DUT connectors and kits at ports 1 and 2 of the PNA
vmc.ConnectorType (1) = "APC 3.5 female"
vmc.ConnectorType (2) = "APC 3.5 male"
' Use Mechanical cal kits
vmc.CalKitType(1) = "85033D/E"
vmc.CalKitType(2) = "85033D/E"
' To use an ECal module instead, comment out the above two lines
' and uncomment the appropriate lines below:
' Your ECal module must already be connected
' via USB to the PNA.
' vmc.CalKitType(1) = "N4691-60004 ECal"
' vmc.CalKitType(2) = "N4691-60004 ECal"
' Non-factory characterizations are specified as follows:
' vmc.CalKitType(1) = "N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
' vmc.CalKitType(1) = "N4691-60004 ECal 01234"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
' vmc.CalKitType(1) = "N4691-60004 MyDskChar ECal 01234"
'
MsgBox("Cal kits defined for Ports 1 and 2")
' By default, VMC requires the measurement of a Calibration Mixer.
' To determine the conversion loss of the calmixer, the cal wizard
' will add a step to perform a 1 port cal at the output of the mixer.
' To avoid performing this step, provide the wizard with a
' mixer characterization file. Uncomment the following two lines to
' specify the characterization file. This is an .S2P file.
' vmc.CharFileName = myMixerCharFile ' this file will be read
' vmc.LoadCharFromFile = true
'
' OR to perform a mixer characterization ONLY
' uncomment the following two lines.
' The outcome of the calibration is an S2P file.
' vmc.CharacterizeMixerOnly = true
' vmc.CharFileName = MyMixerCharFile ' this file will be written
'
' By default, auto orientation of the ecal module is performed
' Uncomment the following lines to manually orient the ecal
' vmc.autoorient = false
' for 2-port portion, ecal port A connected to PNA port 1
' vmc.EcalOrientation2Port(1) ="A1,B2"
' for mixer char, ecal port A connected to cal mixer output
' vmc.EcalOrientation1Port(1) = "A1"
'
' the main calibration loop
' a description for the connection instructions is read
' and then the standard is acquired

dim steps, connectionPrompt
steps = vmc.GenerateSteps
wscript.echo "Number of Steps = " + cstr(steps)
if (steps > 0) then ' otherwise an error condition occurred
for i = 1 to steps
    connectionPrompt = vmc.GetStepDescription( i )
    wscript.echo connectionPrompt
    vmc.AcquireStep( i )
next
vmc.GenerateErrorTerms
end if
end sub
Create an SMC Fixed Output Measurement with COM

This VBScript example creates a calibrated SMC fixed output measurement using a controlled LO. Then a single sweep is taken and data is retrieved.

Requirements:

- If an external LO is used, it should be configured to match the LOName property of the mixer object.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as SMC.vbs. Learn how to setup and run the macro.

```vb
option explicit
' Utility function
function ToString(complexDataArray)
    dim dataAsString
    dim point
    for point = 0 to UBound(data)
        dataAsString = dataAsString & "(" & data(point,0) & "," & data(point,1) & ")"
    next
    ToString = dataAsString
end function
dim app
set app = createobject("agilentpna835x.application")
app.preset
' Put the channel in hold (highly recommended)
app.ActiveChannel.Hold 1
' Delete the standard measurement
app.ActiveMeasurement.Delete
' Create an SC21 measurement
app.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","SC21"
' Set the number of points to 11
app.ActiveChannel.NumberOfPoints = 11
' Setup the mixer parameters for a swept LO, fixed output measurement
dim mixer
set mixer = app.ActiveMeasurement
mixer.InputStartFrequency = 200e6
```
mixer.InputStopFrequency = 700e6
mixer.LORangeMode(1) = 0  ' 0 = Swept mode
mixer.OutputFixedFrequency = 3.4e9
mixer.InputPower = -17
mixer.LOPower(1) = 10
'mixer.LOName(1) = "8360"
'The CALCULATE method calculates the LO frequency from the other parameters,
'It also applies ALL mixer parameters to the channel.
mixer.Calculate 3  ' Calculate the LO range
' Create an S11 in the same channel
app.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","S11"
dim S11Meas
set S11Meas = app.ActiveMeasurement
' Create an IPwr in the same channel
app.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","IPwr"
' Create an OPwr in the same channel
app.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","OPwr"
' Perform a single sweep synchronously.
app.ActiveChannel.Single 1
' Retrieve the SC21 data
dim data
'Get the calibrated values in polar format
data = mixer.GetData(1,3)  ' 1 = naCorrectedData, 3 = naDataFormat_Polar
wscript.echo "SC21=" & ToString(data)
' Retrieve the S11 data
'Get the calibrated values in polar format
data = S11Meas.GetData(1,3)  ' 3 = naDataFormat_Polar
wscript.echo "S11=" & ToString(data)
Create a Segmented Sweep for Mixers

This example program shows how to setup a segment sweep in FCA.
This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as Seg.vbs.

Learn how to setup and run the macro.

See Also

Converter Object

See Other COM Example Programs

```vbs
option explicit
Dim app, chan, conv
Set app = CreateObject("AgilentPNA835x.Application")
Set chan = app.ActiveChannel
Set conv = chan.GetConverter
app.Reset
' Create FCA Scalar Mixer/Converter channel with an SC21 measurement:
app.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter", "SC21", 1
' Delete all existing segments, and create three new ones
conv.DeleteAllSegments()
conv.AddSegment 1, 3
' Set segment sweep
' This command discards the changes made to the scratch mixer
' Therefore, precede with Apply
' Also, always do this before setting the LO port
conv.Apply
chan.SweepType = 4  'segment sweep
' Setup segment #1
' Input is swept from 1.1GHz to 1.39GHz
conv.SegmentStartFrequency(1,0)=1.1e9
conv.SegmentStopFrequency(1,0)=1.39e9
'Swept input
conv.SegmentRangeMode(1,0)=0
' Input power is -10 dBm
```
conv.SegmentFixedPower(1,0)=-10.0
' LO1 is fixed: 2.2 GHz
conv.SegmentFixedFrequency(1,2)=2.2e9
' LO1 power is 10.0 dBm
conv.SegmentFixedPower(1,2)=10.0
' Number of points is 21
conv.SegmentPoints(1)=21
' Output is swept
conv.SegmentRangeMode(1,1)=0
' Output is low-side
conv.SegmentMixingMode(1,1)=0
' Output is calculated from input and lo1
conv.SegmentCalculate(1,2)
' Turn on segment 1
conv.SegmentState(1)=True
' Setup segment #2 from 1.40 to 1.49 GHz
' All else the same
conv.SegmentStartFrequency(2,0)=1.4e9
conv.SegmentStopFrequency(2,0)=1.49e9
conv.SegmentRangeMode(2,0)=0
conv.SegmentFixedPower(2,0)=-10.0
conv.SegmentFixedFrequency(2,2)=2.2e9
conv.SegmentFixedPower(2,2)=10.0
conv.SegmentPoints(2)=21
conv.SegmentRangeMode(2,1)=0
conv.SegmentMixingMode(2,1)=0
conv.SegmentCalculate(2,2)
conv.SegmentState(2)=True
' Setup segment #3 from 1.50 to 1.59 GHz
' All else the same
conv.SegmentStartFrequency(3,0)=1.5e9
conv.SegmentStopFrequency(3,0)=1.59e9
conv.SegmentRangeMode(3,0)=0
conv.SegmentFixedPower(3,0)=-10.0
conv.SegmentFixedFrequency(3,2)=2.2e9
conv.SegmentFixedPower(3,2)=10.0
conv.SegmentPoints(3)=21
conv.SegmentRangeMode(3,1)=0
conv.SegmentMixingMode(3,1)=0
conv.SegmentCalculate(3,2)
conv.SegmentState(3)=True
' Mixer Input to be port 1
' Mixer output to Port 2
' Mixer LO to Port 3
conv.LOName(1)="Port 3"
' Apply the scratch mixer
conv.Apply
Use an Existing Power Cal During an SMC Cal

This example shows how to use an existing Source Power Cal instead of the power cal that is performed during an SMC calibration. To run this program without modification, you need the following:

- A Mixer setup file saved on the PNA: C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr.
- If the mixer file uses an external LO source, it must be connected and configured.
- An ECal module that covers the frequency range of the measurement.
- An SMC cal set named "SMC_CAL". This is the cal set that source power correction data will be imported from. The input and output frequency ranges of the cal set must cover the corresponding ranges used during calibration, or guided cal initialization will fail.

Error Messages

- If you attempt to import power cal data from an SMC calset that uses different ports than the ones currently in use, the message “The necessary calibration standards were not found.” will appear.
- If the imported Cal Set does not cover the frequency range of the current cal, the message “Interpolation target is out of range. Cannot interpolate.” will appear.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as SMC.vbs. Learn how to setup and run the macro.

See Also

SMC Type Object
ImportDataSet Method

See Other COM Example Programs

Dim App
Set App = CreateObject("AgilentPNA835x.Application")
App.Preset
Dim Meas
Set Meas = App.ActiveMeasurement
Meas.Delete
App.CreateCustomMeasurementEx 1, "Scalar Mixer/Converter","SC21"
'Other valid strings that can be specified to create a measurement with a parameter
'other than"SC21" are: "S11", "S22", "IPwr", and "OPwr"
Set Meas = App.ActiveMeasurement
'You can perform mixer setup here or
'recall a previous mixer setup from the PNA Hard drive.
'This is how the mixer could be configured through the IMixer interface
Dim mix
Set mix = Meas ' reference to IMixer object
mix.ActiveXAxisRange = 0 ' 0 = mixINPUT (Input frequency range)
' Alternatively, recall a previous mixer setup from the PNA Hard drive
Meas.LoadFile "C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr"
app.activechannel.numberofpoints = 21
Dim CalMgr
Set CalMgr = App.GetCalManager
Dim SMC
Set SMC = CalMgr.CreateCustomCal("SMC")
SMC.Initialize 1, 1
SMC.ConnectorType(1) = "APC 3.5 male"
SMC.ConnectorType(2) = "APC 3.5 female"
SMC.CalKitType(1) = "N4691-60004 ECal"
SMC.CalKitType(2) = "N4691-60004 ECal"
' Import power cal data from an existing SMC calset.
SMC.ImportDataSet "SMC_CAL","POWER_STEP"
'Omit the isolation part of the 2-port cal (default behavior).
SMC.OmitIsolation = 1
'Turn on auto orientation for the ECal (default behavior).
SMC.AutoOrient = 1
Dim steps
steps = SMC.GenerateSteps
For i = 1 To steps
    MsgBox SMC.GetStepDescription(i)
    SMC.AcquireStep i
Next
Dim calset
calset = SMC.GenerateErrorTerms
Msgbox("SMC Cal Complete!")
Create a Pulsed Measurement

**Note:** This example applies only to the E836x Opt H11. For PNA-X see [PNA-X Create a Narrowband Pulsed Measurement](#).

The following example demonstrates how to create a narrowband pulsed measurement using the Pulsed Application DLL. It first gets valid configuration settings and then uses those settings to configure the PNA and external pulsed generators.

To run this program, you need:

- **Pulsed Application** (Option H08)
- External Pulse Generators
- External Pulse Modulator / Pulse Bias

**See Also**

- Learn how to [install and register the pulsed .dll](#) on your PC
- See the [ConfigureNarrowBand3](#) Method for sending and returning parameters to the .dll.
- See the documentation for the following [COM IF Configuration](#) commands.
- See the [SCPI IF Configuration](#) commands.
- Learn about the **Pulsed Application**.

```vba
' Interfaces
Dim OApp As AgilentPNA835x.Application
Dim OIF As AgilentPNA835x.IFConfiguration

' Pulsed parameters
Dim DPRF As Double
Dim DOffset As Double
Dim DSampleRate As Double
Dim LNumTaps As Long
Dim LBW As Long
Dim IPrecision as Integer
Dim BPG81110 As Boolean
Dim BFixedPRF As Boolean

' pulsed DLL interface
Dim OPulsed As New AgilentPNAPulsed.Application
DPRF=5123 'Hz
LBW=100 'Hz
BPG81110=True 'Using the Agilent 81110A Pulse Generator
BFixedPRF=True 'Do not change the PRF during filter alignment. Only adjust the IFBW.

' Calculate precision of pulse generators so that the config function returns the correct precision with the right filter. For example, DPRF=5000 Hz with a pulse
```
generator that will only take a total of four numeric values
' (5.123 kHz)
'->log10(DPRF)=3.709
'->int(3.709)=3
'->3-3=0

'The algorithm will use a 10^x value for decrementing the PRF for null computation. This means that the first numeric digit from the right should be the one that is decremented by the pulsed algorithm (i.e. 5.122 kHz) to compute the filter nulls. This ensures that the pulse generators receive a PRF that is within their precision with the associated nulling IFBW.
IPrecision = Int(Flog10(CSng(DPRF))) - 3

'Send desired pulsed parameters to the pulsed configuration DLL. The DLL will return a new set of pulse parameters that provide the proper filter nulling.
OPulsed.ConfigNarrowBand3 DPRF, LNumTaps, LBW, DOffset, DSampleRate, IPrecision, BFixedPRF,BPG81110

'Send configuration to PNA

'Set PNA IFBW close to that returned by pulsed algorithm. This ensures that the proper settling time is set on the PNA.
OApp.ActiveChannel.IFBandwidth = LBW
OIF.IFFilterSamplePeriodMode = naMANUAL
OIF.IFFilterSamplePeriod = DSampleRate
OIF.IFFilterSource = naIFFilterSourceManual
OIF.IFFilterSampleCount = LNumTaps
OIF.IFGateEnable = True
OApp.ActiveChannel.FrequencyOffsetState = naON
OApp.ActiveChannel.FrequencyOffsetFrequency = DOffset

'Set receivers to medium gain setting
OIF.IFGainMode("ALL") = naMANUAL
OIF.IFGainLevel("A") = 1
OIF.IFGainLevel("B") = 1
OIF.IFGainLevel("R1") = 1
OIF.IFGainLevel("R2") = 1

Public Function Flog10(SGNum As Single) As Single
Flog10 = Log(SGNum) / Log(10)
End Function

'Enter Code here to send configuration to external pulse generators
Create a Balanced Measurement using COM

The following program creates several Balanced measurements in separate windows, generates markers, calculates statistics, and sets limit lines and queries results.

Note: By their nature, balanced measurements are extremely sensitive to phase differences between the two RF paths that make up the balanced port, especially at higher frequencies. A good calibration (not performed in this example) is critical to achieving good balanced measurement results.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as BalancedCOM.vbs. Learn how to setup and run the macro.

```vbs
' PNA application object
Dim app

' Channel 1 object
Dim chan1

' start of marker/limit testing range
Dim minTestStimulus

' end of marker/limit testing range
Dim maxTestStimulus

' Set to true if you want additional balanced measurements.
Dim AdditionalMeasurements
AdditionalMeasurements = 1

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
' Preset the instrument
app.Preset
' Get the Channel 1 object
Set chan1 = app.Channels(1)
' Stop data taking for now.
chan1.Hold true
' Set up the start / stop frequency for Channel 1 sweep.
MHZ = 1000000
GHZ = 1000*MHZ
chan1.StartFrequency = 10 *MHZ
chan1.StopFrequency = 1 *GHZ
chan1.NumberOfPoints = 801
' Define our test frequency range
minTestStimulus = 100*MHZ
maxTestStimulus = 900*MHZ
' This example uses DUT topology Bal-Bal -
' a DUT with a balanced input and balanced output.
' Port mapping for our DUT:
' logical port 1 = physical ports 1 and 4
```
logical port 2 = physical ports 2 and 3
The default is:
logical port 1 = physical ports 1 and 2
logical port 2 = physical ports 3 and 4

logical 1             logical 2
1 --------|                |---- 2 +
|          | DUT               |
4 --------|------------------|---- 3 -

chan1.BalancedTopology.SetBBPorts 1, 4, 2, 3

Now we create some Bal-Bal measurements.
By creating Bal-Bal measurements ("BBAL:..."),
the channel is set to Bal-Bal topology,
so it is not necessary to do this explicitly
with the BalancedTopology.DUTTopology command.
We do it here just for clarity:

chan1.BalancedTopology.DUTTopology = 2
0 == SE-Bal, 1 == SE-SE-Bal, 2 == Bal-Bal

Create four windows, each showing one category of balanced measurement:
Create Forward Transmission Measurements in Bal-Bal topology on Channel 1, window 1

differential mode transmission
app.CreateMeasurement 1, "BBAL:SDD21",1,1
Set sdd21_1 = app.ActiveMeasurement

differential to common mode conversion
app.CreateMeasurement 1, "BBAL:SCD21",1,1
Set scd21_1 = app.ActiveMeasurement

common to differential mode conversion
app.CreateMeasurement 1, "BBAL:SDC21",1,1
Set sdc21_1 = app.ActiveMeasurement

common mode transmission
app.CreateMeasurement 1, "BBAL:SCC21",1,1
Set scc21_1 = app.ActiveMeasurement

Optionally create some additional measurements
If AdditionalMeasurements Then

Create (logical) Port 1 reflection measurements, channel 1, window 2
app.CreateMeasurement 1, "BBAL:SDD11",1,2 ' differential mode reflection
app.CreateMeasurement 1, "BBAL:SCD11",1,2 ' C to D mode conversion reflection
app.CreateMeasurement 1, "BBAL:SDC11",1,2 ' D to C mode conversion reflection
app.CreateMeasurement 1, "BBAL:SCC11",1,2 ' common mode reflection
' Create Reverse Transmission Measurements, channel 1, window 3
app.CreateMeasurement 1, "BBAL:SDD12",1,3 ' differential mode transmission
app.CreateMeasurement 1, "BBAL:SCD12",1,3 ' differential to common mode conversion
app.CreateMeasurement 1, "BBAL:SDC12",1,3 ' common to differential mode conversion
app.CreateMeasurement 1, "BBAL:SCC12",1,3 ' common mode transmission

' Create (logical) Port 2 reflection measurements in window 4
app.CreateMeasurement 1, "BBAL:SDD22",1,4 ' differential mode reflection
app.CreateMeasurement 1, "BBAL:SDC22",1,4 ' C to D mode conversion reflection
app.CreateMeasurement 1, "BBAL:SCD22",1,4 ' D to C mode conversion reflection
app.CreateMeasurement 1, "BBAL:SCC22",1,4 ' common mode reflection

End If

' Set up some limit lines to verify a minimum differential insertion loss
sdd21_1.LimitTest(1).BeginStimulus = minTestStimulus
sdd21_1.LimitTest(1).EndStimulus = maxTestStimulus
sdd21_1.LimitTest(1).BeginResponse = -2
sdd21_1.LimitTest(1).EndResponse = -2
sdd21_1.LimitTest(1).Type = 2 ' minimum limit
sdd21_1.LimitTest.State = 1

' Limit lines for maximum common mode to differential conversion
sdc21_1.LimitTest(1).BeginStimulus = minTestStimulus
sdc21_1.LimitTest(1).EndStimulus = maxTestStimulus
sdc21_1.LimitTest(1).BeginResponse = -20
sdc21_1.LimitTest(1).EndResponse = -20
sdc21_1.LimitTest(1).Type = 1 ' maximum limit
sdc21_1.LimitTest.State = 1

' Take a (synchronous) single sweep on channel 1
chan1.Single true

' Show differential forward transmission statistics.
sdd1_1.ShowStatistics = true

' Set up user range 1 to limit marker’s search range.
chan1.UserRangeMin(0,1) = minTestStimulus
chan1.UserRangeMax(0,1) = maxTestStimulus

' Find/Show max common mode to differential conversion, and read back the frequency.
sdc21_1.MarkerState(1) = true
  Set marker 1 to use user range 1
  sdc21_1.Marker(1).UserRange = 1
  sdc21_1.Marker(1).SearchMax

' Find/Show max differential mode insertion loss, and read back the frequency.
sdd21_1.MarkerState(1) = true
  Set marker 1 to use user range 1
  sdd21_1.Marker(1).UserRange = 1
  sdd21_1.Marker(1).SearchMin
If sdd21_1.LimitTestFailed Then
End If
If sdc21_1.LimitTestFailed Then
End If
Create a PMAR Device and Measurement

The following program creates a new External Device: a Power Meter as Receiver, makes several power meter settings, and then creates a PMAR measurement.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as PMAR.vbs. Learn how to setup and run the macro.

See Also

ExternalDevices Collection
ExternalDevice Object
PowerSensorAsReceiver Object
PowerSensorCalFactorSegmentPMAR Object
PowerLossSegmentsPMAR Collection
PowerLossSegmentPMAR Object

dim app
Set app = CreateObject("AgilentPNA835x.Application")
dim externalDevices
Set externalDevices = app.ExternalDevices
dim devicecount
devicecount = externalDevices.count
externalDevices.Add "NewPMAR"
dim newExternalDevice
Set newExternalDevice = externalDevices.Item("NewPMAR")
newExternalDevice.DeviceType = "Power Meter"
newExternalDevice.IOConfiguration= "GPIB0::14::INSTR"
'newExternalDevice.IOConfiguration = "USB0::2391::4865::GB45100278::0::INSTR"
newExternalDevice.IOEnable = true
dim PMAR
Set PMAR = newExternalDevice.ExtendedProperties
PMAR.SensorIndex = 1
PMAR.ReadingsPerPoint = 10
dim avr
avr = PMAR.ReadingsPerPoint
PMAR.ReadingsTolerance = 0.1
dim tole
tole = PMAR.ReadingsTolerance
PMAR.MinimumFrequency = 100000000
PMAR.MaximumFrequency = 10000000000
PMAR.LimitFrequency = false
PMAR.referenceCalFactor = 99
Set powerCalFactorSegments = PMAR.CalFactorSegments
powerCalFactorSegments.Add 1,10
Set calpair = powerCalFactorSegments(1)
calpair.Frequency = 1e9
calpair.CalFactor = 99
Set calpair = powerCalFactorSegments(2)
calpair.Frequency = 2e9
calpair.CalFactor = 98
powerCalFactorSegments.Remove 3,8
PMAR.UsePowerLossSegments = true
Set pls = PMAR.PowerLossSegments
pls.Add 1,5
Set pl = pls(1)
pl.Loss = -1
pl.Frequency = 1e9
Set pl = pls(2)
pl.Loss = -2
pl.Frequency = 2e9
pls.Remove 3,3
newExternalDevice.active = true

'Create a PMAR trace with power meter connected to port 3
app.CreateMeasurement 1,"NewPMAR",3,1
Create a Wideband Pulsed Measurement using the PNA-X

This Visual Basic COM example shows you how to configure the PNA-X internal pulse generators and modulators to make wideband pulsed measurements in pulse profile mode using the PNA-X.

Visit the PNA website where you can download a free Wideband Pulsed Application that performs this measurement on the PNA-X.

See all COM Pulsed examples

```vba
'Create an PNA Application instance
Dim pnaApp As New AgilentPNA835x.Application

'Create a PathConfiguration instance
Dim pathConf As AgilentPNA835x.PathConfiguration

'Create an PulseGenerator instance
Dim pulseGen As AgilentPNA835x.PulseGenerator

'Create a Channel instance
Dim myChan As AgilentPNA835x.Channel

'Preset PNA-X
pnaApp.Preset

'Assign current active channel to myChan object
Set myChan = pnaApp.ActiveChannel

'Let PNA-X work in CW mode because of doing pulse profile measurement
myChan.SweepType = naCWTimeSweep

'Set CW Freq to 4 GHz
myChan.CWFrequency = 4000000000#

'Set IF Bandwidth to 5 MHz to get the best time resolution
myChan.IFBandwidth = 5000000#

'Assign current active channel path configuration to pathConf object
Set pathConf = myChan.PathConfiguration

'Let PNA-X source work in ALC Open Loop mode
myChan.ALCLevelingMode(1) = naALCOpenLoop

'Make the Pulse1 as modulation pulse to generate Pulsed-RF signal
pathConf.Element("PulseModDrive").Value = "Pulse1"

'Enable pulse modulation at Source1Out1 path
pathConf.Element("Src1Out1PulseModEnable").Value = "Enable"

'Assign current active channel pulse generator to pulseGen object
Set pulseGen = myChan.PulseGenerator
```
'Internal pulse generator has five channels,
'default the channel 0 use as internal ADC trigger signal
'Enable channel 0 of internal pulse generator as trigger signal
pulseGen.State(0) = True
'Enable channel 1 of internal pulse generator as modulation signal
pulseGen.State(1) = True
'Set pulse period to 10 us
pulseGen.Period = 0.00001 '10 us
'Set pulse width of channel 0 to 1 us
pulseGen.Width(0) = 0.000001 ' 1 us
'Set pulse width of channel 1 to 5 us
pulseGen.Width(1) = 0.000005 '5 us

End Sub

Last Modified:

4-Jan-2008   Added point trigger note
2-Oct-2007   MX New topic
Create an IM Spectrum Measurement

This VBScript example creates IM Spectrum measurement based on passed parameters. This subprogram is extracted from the macro on the PNA that produces an IM spectrum channel from the Marker function. You can see the entire program at C:/Program Files/Agilent/Network Analyzer/Applications/IMD/IMD.VBS. This VBScript (*.vbs) program must be used as part of a program that supplies the required parameters. When complete, it can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as IMD.vbs.

Learn how to setup and run the macro.

See SweptIMD Object.

```
'' SetupIMSpectrum
'' Setup an IM Spectrum (non-converter) channel based upon supplied parameters
sub SetupIMSpectrum(app, MkrPos, ToneSpacing, TonePower)
    dim objIMXChan, objIMDChan
    dim Fstart, Fstop, NumPoints, ToneFc

    set objIMXChan = objIMSChan.CustomChannelConfiguration
    set objIMDChan = objSIMDChan.CustomChannelConfiguration

    NumPoints = objSIMDChan.NumberOfPoints
    select case objIMDChan.SweepType
        case naIMDToneCWSweep
            ToneFc = objIMDChan.FrequencyCenter
        case naIMDTonePowerSweep
            Fstart = objIMDChan.TonePowerStart(0)
            Fstop = objIMDChan.TonePowerStop(0)
            TonePower = CalcMkrValue(Fstart, Fstop, MkrPos, NumPoints)
            ToneFc = objIMDChan.FrequencyCenter
        case naIMDToneCenterFreqSweep
            Fstart = objIMDChan.FrequencyCenterStart
```

2132
Fstop = objIMDChan.FrequencyCenterStop
ToneFc = CalcMkrValue(Fstart, Fstop, MkrPos, NumPoints)

case naIMDDeltaFrequencySweep
    ToneFc = objIMDChan.FrequencyCenter
    Fstart = objIMDChan.DeltaFrequencyStart
    Fstop = objIMDChan.DeltaFrequencyStop
    ToneSpacing = CalcMkrValue(Fstart, Fstop, MkrPos, NumPoints)
end Select

objIMXChan.FrequencyCenter = ToneFc
objIMXChan.DeltaFrequency = ToneSpacing
objIMXChan.TonePower(0) = TonePower
objIMXChan.TonePower(1) = TonePower
app.ActiveMeasurement.Trace.ReferenceValue = TonePower + 10
objIMSChan.continuous
end Sub
Create an iTMSA Measurement

The following VB Script example shows how to create an iTMSA measurement with Power Sweep. Click each link to see a detailed description of each command.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as Testset.vbs. Learn how to setup and run the macro.

```vbs
dim app 'PNA App
dim meas 'Measurement
dim balancemeas
dim balstimulus
dim chan

set app = createobject("agilentpna835x.application")
set chan = app.activechannel
chan.SweepType = 2 ' Set the sweep type to power sweep

set meas = app.ActiveMeasurement
set balancemeas = meas.BalancedMeasurement
balancemeas.BalancedTopology.DUTTopology = 2 ' Bal-Bal topology
balancemeas.BalancedStimulus.Mode = 1 ' Turn on true mode

' The PNA-X balanced port numbers are always (0)=Bal 1; (-1)=Bal2
chan.StartPowerEx(0) = -5 ' Set the balanced port 1 start power to -5 dbm
chan.StopPowerEx(0) = 5 ' Set the balanced port 1 stop power to 5 dbm
chan.StartPowerEx(-1) = -10 ' Set the balanced port 2 start power to -5 dbm
chan.StopPowerEx(-1) = 0 ' Set the balanced port 2 stop power to 5 dbm
```

Last Modified: 2-Jun-2008  MX New topic
Create and Cal a Gain Compression Measurement

This VBScript example creates and calibrates a Gain Compression measurement and performs Compression analysis.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as GCA.vbs. Learn how to setup and run the macro.

See Gain Compression Object.

```vbnet
option explicit

dim CompLevel, Tolerance, StartFreq, StopFreq, NumFreqs, Scale, LinearPower

dim AcqMode, BackOff, StartPower, StopPower, NumPowers, EnableInterp, CompAlg

dim DwellTime, IFBandwidth, ShowIterations, host, app

' GCA Settings/Values

'' Acquisition Mode:
'' naSmartSweep = 0
'' naSweepPowerAtEachFreq2D = 1
'' naSweepFreqAtEachPower2D = 2
''
'' Compression Algorithm
'' naCompressionFromLinearGain = 0
'' naCompressionFromMaximumGain = 1
'' naBackoffCompression = 2
'' naXYCompression = 3
''
'' EndOfSweepOperation
'' naDefaultPowerSet = 0
'' naSetToStartPower = 1
'' naSetToStopPower = 2
'' naSetRFOff = 3
''

CompLevel = 1 ' 1 dB compression level
Tolerance = 0.05 ' SMART Sweep tolerance
```
StartFreq = 1E9
StopFreq = 9E9
NumFreqs = 201
Scale = 0.1
LinearPower = -20
BackOff = 10
StartPower = -20
StopPower = 8
NumPowers = 60
DwellTime = 0.0005
IFBandwidth = 1000
EnableInterp = False
AcqMode = 0
CompAlg = 0
ShowIterations = False

dim objargs
set objargs = wscript.Arguments
if (objArgs.Count = 1) then host = objargs(0)

set app = CreateObject("Agilentpna835x.application")
call SetupGCA(app, StartFreq, StopFreq, NumFreqs, EnableInterp, Scale, CompLevel, LinearPower, AcqMode, BackOff, StartPower, StopPower)
NumPowers, _
CompAlg, _
DwellTime, _
IFBandwidth, _
ShowIterations )
call CalGCA ( app )
call Analysis ( app )
```
'' GCA Setup
```
sub SetupGCA ( app, StartFreq, StopFreq, NumFreqs, EnableInterp, Scale, CompLevel, LinearPower, _
    AcqMode, BackOff, StartPower, StopPower, NumPowers, CompAlg, DwellTime, IFBandwidth, _
    ShowIterations )
dim chan, gca
app .reset
app .CreateCustomMeasurementEx 1, "Gain Compression", "S21", 1
set chan = app .channels (1)
chan .hold 1
app .CreateCustomMeasurementEx 1, "Gain Compression", "CompIn21", 1
app .CreateCustomMeasurementEx 1, "Gain Compression", "DeltaGain21", 1
app .nawindows (1).traces (3).YScale = Scale
app .nawindows (1).traces (3).ReferenceValue = -CompLevel
set gca = chan .CustomChannelConfiguration
gca .InputLinearPowerLevel = LinearPower
gca .AcquisitionMode = AcqMode
gca .CompressionLevel = CompLevel
gca .CompressionBackoff = BackOff
gca .CompressionDeltaX = BackOff
gca .CompressionDeltaY = BackOff - CompLevel
gca .CompressionAlgorithm = CompAlg
gca .NumberOfPowerPoints = NumPowers
gca .CompressionInterpolation = EnableInterp
gca .SmartSweepSettlingTime = DwellTime
gca .SmartSweepShowIterations = ShowIterations
chan .IFBandwidth = IFBandwidth
chan .DwellTime = DwellTime
chan .StartPower = StartPower
chan .StopPower = StopPower
chan .StartFrequency = StartFreq
chan .StopFrequency = StopFreq
chan .NumberOfPoints = NumFreqs
chan .single 1
end sub

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
sub Analysis (app)
Dim meass
Dim ana
Set meass = app.Measurements ' get the measurements
Set ana = meass(1).CustomMeasurementConfiguration ' get the measurement
ana.AnalysisEnable = true ' enable the analysis mode
ana.AnalysisCWFreq = 3e9 ' set the analysis cw frequency to 3GHz
Set ana = meass(2).CustomMeasurementConfiguration
ana.AnalysisEnable = true
ana.AnalysisCWFreq = 4e9
ana.AnalysisXAxis = naPsourceAsXAxis ' set the XAxis as the source power setting
Set ana = meass(3).CustomMeasurementConfiguration
ana.AnalysisEnable = true
ana.AnalysisIsDiscreteFreq = false ' turn off the discrete frequency option
ana.AnalysisCWFreq = 4.5e9
end sub
Create and Cal a GCX Measurement

This VBScript example creates and calibrates a GCX measurement and performs Compression analysis.
This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as GCX.vbs. Learn how to setup and run the macro.

See Gain Compression Object and Converter Object

```vbnet
option explicit
dim CompLevel, Tolerance, StartFreq, StopFreq, LOFreq, NumFreqs, Scale, LinearPower
dim AcqMode, BackOff, StartPower, StopPower, NumPowers, EnableInterp, CompAlg
dim DwellTime, IFBandwidth, ShowIterations, host, app, parser
CompLevel = 1 ' 1 dB compression level
Tolerance = 0.05 ' SMART Sweep tolerance
StartFreq = 2.5E9
StopFreq = 2.6E9
LOFreq = 1.7E9
NumFreqs = 21
Scale = 0.1
LinearPower = -10
BackOff = 10 ' Not used for Deviation from linear gain
StartPower = -20
StopPower = 8
NumPowers = 60 ' Not used for SMART Sweep
DwellTime = 0.0005 ' Allow some time for DUT bias/thermal effects
IFBandwidth = 1000 ' Reasonable trace noise at -20 dBm
EnableInterp = False ' Disable interpolation
AcqMode = 0 ' Smart Sweep
CompAlg = 0 ' Deviation from linear gain
ShowIterations = False ' Configure SMART to not show iteration results
dim objargs
set objargs = wscript.Arguments
if (objArgs.Count = 1) then host = objargs(0)
'----------------------------------------------------------
```

2140
' Create and Configuration GCX Channel:
'--------------------------------------------------------------------------
set app = CreateObject("Agilentpna835x.application")
call SetupGCX( parser, _
    StartFreq, _
    StopFreq, _
    LOFreq, _
    NumFreqs, _
    EnableInterp, _
    Scale, _
    CompLevel, _
    LinearPower, _
    AcqMode, _
    BackOff, _
    StartPower, _
    StopPower, _
    NumPowers, _
    CompAlg, _
    DwellTime, _
    IFBAndwidth, _
    ShowIterations )
call CalGCX( parser )
'--------------------------------------------------------------------------
' GCAX Setup
'--------------------------------------------------------------------------
sub SetupGCX( parser, StartFreq, StopFreq, LOFreq, NumFreqs, EnableInterp, Scale, CompLevel, LinearPower, _
    AcqMode, BackOff, StartPower, StopPower, NumPowers, CompAlg, DwellTime, IFBAndwidth, _
    ShowIterations )
dim chan, gca
app.reset
app.CreateCustomMeasurementEx 1, "Gain Compression Converters", "SC21", 1
set chan = app.channels(1)
dim converter
set converter = chan.Converter()
chan.hold 1
app.CreateCustomMeasurementEx 1, "Gain Compression Converters", "CompIn21", 1
app.CreateCustomMeasurementEx 1, "Gain Compression Converters", "DeltaGain21", 1
app.nawindows(1).traces(3).YScale = Scale
app.nawindows(1).traces(3).ReferenceValue = -CompLevel
set gca = chan.CustomChannelConfiguration
gca.InputLinearPowerLevel = LinearPower
gca.AcquisitionMode = AcqMode
gca.CompressionLevel = CompLevel
gca.CompressionBackoff = BackOff
gca.CompressionDeltaX = BackOff
gca.CompressionDeltaY = BackOff - CompLevel
gca.CompressionAlgorithm = CompAlg
gca.NumberOfPowerPoints = NumPowers
gca.CompressionInterpolation = EnableInterp
gca.SmartSweepSettlingTime = DwellTime
gca.SmartSweepShowIterations = ShowIterations
chan.IFBandwidth = IFBandwidth
chan.DwellTime = DwellTime
chan.StartPower = StartPower
chan.StopPower = StopPower
chan.TestPortPower(1) = LinearPower
chan.StartFrequency = StartFreq
chan.StopFrequency = StopFreq
chan.NumberOfPoints = NumFreqs
set converter properties
converter.InputRangeMode = 0 'swept
converter.LORangeMode(1) = 1 'fixed
converter.OutputRangeMode = 0 'swept
converter.InputStartFrequency = StartFreq
converter.InputStopFrequency = StopFreq
converter.LOFixedFrequency(1) = LOFreq
converter.LOName(1) = "Port 3"
converter.LOPower(1) = -10
converter.Calculate 2 'calculateOutput
chan.Single 1
end sub
sub CalGCAX ( parser )
Dim CalMgr
Set CalMgr = app.GetCalManager
Dim SMC
Set SMC = CalMgr.CreateCustomCal("SMC")
SMC.Initialize 1, 1
SMC.Do2PortEcal = 1 'specify 0 for mechanical cal, 1 for ecal
' use Factory Characterization
SMC.ECALCharacterization(1) = 0
SMC.OmitIsolation = 1
SMC.AutoOrient = 1
' 1- forward, 2-reverse, or Both
SMC.CalibrationPort = "1"
Dim steps
steps = SMC.GenerateSteps
Dim i
For i = 1 To steps
    MsgBox SMC.GetStepDescription(i)
    SMC.AcquireStep i
Next
Dim calset
calset = SMC.GenerateErrorTerms
MsgBox("SMC Cal Complete!")
end sub
Create and Cal a Noise Figure Measurement

This example program creates a Noise Figure measurement, then calibrates the measurement. You MUST change the ECal Identification strings (in Blue font).

Optional: Uncomment the following lines (in Blue font) to change these settings:

- Noise Receiver = Noise Receiver to Std (PNA) Receiver
- Cal Method = "Vector" to "Scalar"
- Receiver Characterization Method = "NoiseSource" to "Power Meter"

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Noise.vbs. Learn how to setup and run the macro.

See Also
NoiseFigure Object.
Accessing the NoiseFigure object and NoiseCal object using C#

See other COM Examples

```vbscript
windowNum = 1
channelNum = 1
set pna=CreateObject("AgilentPNA835x.Application")
set scpi = pna.ScpiStringParser
pna.reset
' Create noise figure measurement
set noise = pna.createcustommeasurementex(channelNum, "NoiseFigure", "NF", windowNum)
set noisechan = pna.activechannel
' Create object to access noise-specific channel attributes
set noiseConfig = pna.activechannel.CustomChannelConfiguration
' Create guided noise calibration object on our channel
set noiseCal = pna.GetCalmanager.CreateCustomCalEx(channelNum)
set noiseCalExtension = noiseCal.CustomCalConfiguration
noiseCalExtension.NoiseSourceCold  = 300
' Substitute appropriate ECal identification strings here
tunerEcal = "N4691-60004 ECal 02821"
pullEcal = "N4691-60004 ECal 02297"
```
' configuration
ConfigureChannel
ConfigureNoiseSettings
' perform calibration
SetupCalAttributes_Insertable
SetupNoiseSource
FinishCalibration
' ----- Support subroutines ------
' Configure noise channel
sub ConfigureChannel
  noisechan.startfrequency = 500e6
  noisechan.stopfrequency = 5.0e9
  noisechan.numberofpoints = 201
  noisechan.IFBandwidth = 1.0E3
end sub
' Configure noise-specific channel settings
sub ConfigureNoiseSettings
  noiseConfig.NoiseReceiver = 1 'Noise Receiver
  noiseConfig.NoiseReceiver = 0 'Std PNA Receiver
  noiseConfig.noiseaveragestate = true
  noiseConfig.NoiseAverageFactor = 40
  noiseConfig.NoiseTuner = tunerEcal
  noiseConfig.NoiseTunerIn = "B"
  noiseConfig.NoiseTunerOut = "A"
  noiseConfig.NoiseBandwidth = 8e6
end sub
sub SetupCalAttributes_Insertable
  noisecal.Initialize channelNum, true
  noisecal.ConnectorType( 1 ) = "APC 3.5 female"
  noisecal.ConnectorType( 2 ) = "APC 3.5 male"

  noisecal.CalKitType (1) = pullEcal
  noisecal.CalKitType (2) = pullEcal

  noiseCalExtension.NoiseSourceConnectorType = "APC 3.5 male"
  noiseCalExtension.NoiseSourceCalKitType = pullEcal
noiseCalExtension.CalMethod = "Vector"
'    noiseCalExtension.CalMethod = "Scalar"
    noiseCalExtension.RcvCharMethod = "NoiseSource"  'Can NOT be used with Std PNA Rcvr
    noiseCalExtension.RcvCharMethod = "PowerMeter"
end sub

sub SetupNoiseSource
    ' specify the ENR file for the noise source
    noiseCalExtension.ENRFile = "C:/Program Files/Agilent/Network Analyzer/Documents/346C_MY44420454.enr"
    noiseCalExtension.NoiseSourceCold = 301.1
end sub

' Build the connection list and acquire the calibration
sub FinishCalibration
    steps = noisecal.GenerateSteps
    for i = 1 to steps
        str = noisecal.GetStepDescription( i )
        messagebox str
        noisecal.AcquireStep i
    next
    guid = noisecal.GenerateErrorTerms
    wscript.echo "Calibration created calset guid: ",guid
end sub

**Bonus: Accessing the NoiseFigure object and NoiseCal objectusing C#**

Replace `<hostname>` with the full computer name of your PNA
Type pna = Type.GetTypeFromProgID("AgilentPNA835x.Application", "<hostname>");
    AgilentPNA835x.Application app =
    (AgilentPNA835x.Application)Activator.CreateInstance(pna);
    app.Reset();
    app.CreateCustomMeasurementEx(1, "NoiseFigure", "NF", 1);
    AgilentPNA835x.ICalManager5 calManager =
    (AgilentPNA835x.ICalManager5)app.GetCalManager();
    AgilentPNA835x.IGuidedCalibration4 guidedCal4 =
    (AgilentPNA835x.IGuidedCalibration4)
    calManager.CreateCustomCalEx(1);
    AgilentPNA835x.INoiseCal noiseCal =
    (AgilentPNA835x.INoiseCal)guidedCal4.CustomCalConfiguration;

---

Last Modified:

9-Jun-2011    Edited for Rcvr Char method (A.09.41)
14-May-2010    Fixed hostname
9-Nov-2007     MX New topic
Create and Cal an NFX Measurement

This program does the following:

- Setup a Noise Figure SC21 Measurement
- Calibrate Noise Figure channel
- Optional - Configure for an Embedded LO

To run this program, make the following edits, highlighted in **yellow**:

- Set `host` to your PNA computer name
- Set `tunerECal` and `pullECal` to your ECal model and info
- Set `ENR` to correct file name and location
- Set `connector types` for ECal, power sensor, and noise source

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as NFX.vbs. [Learn how to setup and run the macro.](#)

**See Also**

- CreateCustomMeasEX command
- NoiseFigure Object
- NoiseCal Object
- Converter Object
- GuidedCal Object
- EmbeddedLO Object

**Learn About...**

- Noise Figure on Converters
- Noise Cal

**See other COM Examples**

```vbs
option explicit
'Nfx sweep type
' naLinearSweep = 0
' naCWTImesSweep = 2
'Converter sweep mode
```
' naSwept = 0
' naFixed = 1
' Embedded LO tuning mode
' Broadband and precise = 0
' Precise only = 1
' None = 2
dim app
dim chan
dim nfx
dim host
host = "MyPNA"
set app = CreateObject("Agilentpna835x.application", host)
app.reset

call SetupNFX
' optional if not doing embedded LO
' call SetupEmbeddedLO
call CalNFX

sub SetupNFX
dim tunerEcal
tunerEcal = "N4691-60003 ECal 00591"
' create NFX traces
app.CreateCustomMeasurementEx 1, "Noise Figure Converters", "NF", 1
app.CreateCustomMeasurementEx 1, "Noise Figure Converters", "SC21", 1

' set channel and application objects
set chan = app.ActiveChannel
set nfx = chan.CustomChannelConfiguration
dim converter
set converter = chan.GetConverter()

' Set channel properties
chan.single 1
chan.sweeptype = 0 "naLinearSweep"
chan.numberofpoints = 201
chan.IFBandwidth = 1.e3
' Set nfx properties
nfx.noiseaveragestate = true
nfx.noiseaveragefactor = 10
nfx.noisetuner = tunerECal
nfx.NoiseTunerIn = "B"
nfx.NoiseTunerOut = "A"
nfx.NoiseBandwidth = 8e6
nfx.noisegain = 0 'low

' converter properties
converter.InputRangeMode = 0 ' swept
converter.LORangeMode(1) = 1 'fixed
converter.OutputRangeMode = 0 'swept
converter.InputStartFrequency = 8.0e8
converter.InputStopFrequency = 3.0e8
converter.LOFixedFrequency(1) = 1.5825e10
converter.LOPower(1) = -10
converter.Calculate 2 'calculateOutput
chan.Single 1
end sub

sub CalNFX
' Set ecal and noise tuner
dim SparamECal
SparamECal = "N4693-60001 User 2 ECal 00012"
chan.single 1

dim calMgr
set calMgr = app.GetCalManager
dim nfxCal
dim nfxCalExt
dim nfxCalExt = nfxCal.CustomCalConfiguration
nfxCalExt.ENRFile = "C:/Program Files/Agilent/Network Analyzer/Noise/346C_44420601.enr"
'setup calibration
nfxCal.Initialize 1, true

'dut connector
nfxCal.ConnectorType(1) = "APC 3.5 female"
nfxCal.ConnectorType(2) = "APC 3.5 female"
nfxCal.CalKitType(1) = SparamECal
nfxCal.CalKitType(2) = SparamECal

'power sensor connector
nfxCal.PowerCalibrationPowerLevel(1) = -20
nfxCal.PowerSensorConnectorType(2) = "APC 3.5 male"
nfxCal.PowerSensorCalKitType(2) = SparamECal

'noise source connector
nfxCalExt.NoiseSourceConnectorType = "APC 3.5 male"
nfxCalExt.NoiseSourceCalKitType = SparamECal
nfxCalExt.CalMethod = "Vector"
nfxCalExt.EnableLOPowerCal(1) = False
nfxCalExt.ForceDeEmbedENRAdapter = False
nfxCalExt. ForceDeEmbedSensorAdapter = False

'step through calsteps
dim steps
steps = nfxcal.GenerateSteps
dim i , str
for i = 1 to steps
str = nfxcal.GetStepDescription(i)
msgbox str
nfxcal.AcquireStep i
next
dim guid
guid = nfxcal.generateerrorterms
wscript.echo "Calibration created calset guid: ", guid
chan.continuous
end sub

sub SetupEmbeddedLO
' Set embedded LO properties
dim ELO
set ELO = converter.ConverterEmbeddedLO
ELO.NormalizePoint = 101
ELO.TuningMode = 0 ' Broadband and precise
ELO. TuningIFBW = 3.0e4
ELO.MaxPreciseTuningIterations = 5
ELO.PreciseTuningTolerance = 1
ELO.TuningSweepInterval = 1
ELO.IsOn = true
chan.Single 1
end sub
Create and Cal a Swept IMD Measurement

This VBScript example creates IMD power and IM3 measurements, sets sweep mode to Center Frequency Sweep, and performs an IMD cal.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as IMD.vbs.

You can see the VB Script program on the PNA that produces an IM spectrum channel from the Marker function at: C:/"/Program Files/Agilent/Network Analyzer/Applications/IMD/IMD.VBS”.

Learn how to setup and run the macro.

See SweptIMD Object.

```
option explicit
'declare variables
dim SweepMode, StartDeltaFreq, StopDeltaFreq, NumFreqs, TonePower, CWFreq
dim app, hostname

' Sweep type:
' naIMDToneCWSweep = 0
' naIMDTonePowerSweep = 1
' naIMDToneCenterFreqSweep = 2
' naIMDDeltaFrequencySweep = 3
' naIMDToneSegmentSweep = 4

'init variables
SweepMode = 3      ' Sweep DeltaF
StartDeltaFreq = 100e3
StopDeltaFreq = 1e9
NumFreqs = 201
TonePower = -7
CWFreq = 5e9

' get host name from commandline
dim objargs
set objargs = wscript.arguments
if(objargs.Count = 1) then hostname = objargs(0)
set app = CreateObject("Agilentpna835x.application", hostname)
call SetupIMD
call CalIMD
```
' Create and Configure IMD channel

sub SetupIMD
    dim chan, imd
    app.reset
    ' Create IMD measurements
    app.CreateCustomMeasurementEx 1, "Swept IMD", "PwrMain", 1
    app.CreateCustomMeasurementEx 1, "Swept IMD", "IM3", 1
    set chan = app.channels(1)
    chan.hold 1

    set imd = chan.CustomChannelConfiguration
    imd.SweepType = SweepMode
    imd.FrequencyCenter = CWFreq
    imd.DeltaFrequencyStart = StartDeltaFreq
    imd.DeltaFrequencyStop = StopDeltaFreq
    imd.TonePower(0) = TonePower 'F1 power
    imd.TonePower(1) = TonePower 'F2 power

    chan.NumberOfPoints = NumFreqs
    chan.single 1
end sub

sub CalIMD
    dim chan, CalMgr, IMDCal, IMDCustomCal, CalSteps, I, CalSet
    set chan = app.ActiveChannel
    set CalMgr = app.GetCalManager
    set IMDCal = CalMgr.CreateCustomCalEx(1)
    'Configure IMD GuidedCal for the connector types and ECal module that will be used
    ' Substitute appropriate connector type and ECal identification strings here
    IMDCal.Initialize 1, true 'channel number is 1
    IMDCal.ConnectorType(1) = "APC 3.5 female"
    IMDCal.ConnectorType(2) = "APC 3.5 male"
    IMDCal.CalKitType(1) = "N4693-60001 User 2 ECal 00012"
IMDCal.CalKitType(2) = "N4693-60001 User 2 ECal 00012"

' IMD Custom settings
set IMDCustomCal = IMDCal.CustomCalConfiguration

' Set the Power Level at the power sensor to be used in calibration
IMDCustomCal.PowerLevel = 0

' Specify the connector type of the power sensor. If there is an adapter between
' the input port and the power sensor, specify the connector type here, and set
' the appropriate cal kit type for the connector so that extra calibration can be
' performed. To skip the calibration for the adapter, set
PowerSensorConnectorType to "Ignored"
' i.e.: IMDCustomCal.PowerSensorConnectorType = "Ignored"
IMDCustomCal.PowerSensorConnectorType = "APC 3.5 female"
IMDCustomCal.PowerSensorCalKitType = "N4693-60001 User 2 ECal 00012"

' Set the Max product to calibrate, valid values are 3, 5, 7, and 9
IMDCustomCal.MaxProduct = 3

' Set the calibration Frequencies, can choose between calibrate only at
center Frequencies (0)
' or calibrate at all frequencies (1).
IMDCustomCal.CalibrationFrequencies = 1

' Include 2nd order product in calibration
IMDCustomCal.Include2ndOrderProduct = true

CalSteps = IMDCal.GenerateSteps
for I = 1 to CalSteps
    msgBox IMDCal.GetStepDescription(I)
    IMDCal.AcquireStep(I)
next
CalSet = IMDCal.GenerateErrorTerms
msgBox "IMD Cal Done"
ENR File Management Example

This VB Script program illustrates ENR file management using COM commands.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as Calibrate.vbs. Learn how to setup and run the macro.

See Other COM Example Programs

' Sample VBS program illustrating COM commands for ENR file management.

option explicit

dim pna ' application
dim enr ' ENRFile object
dim scpi, hostname

set pna=CreateObject("agilentpna835x.application")
set scpi = pna.ScpiStringParser
set enr = pna.ENRFile

' Generate data to put in ENR file
Dim vdata(3)
vdata(0) = 100E6 ' first frequency point
vdata(1) = 14.532 ' first ENR value
vdata(2) = 20E9 ' second frequency point
vdata(3) = 15.731 ' second ENR value

' send data to ENRFile object
enr.PutENRData(vdata)

' Set noise source serial number
enr.ENRSN = "ABCD1234"

' Write ENR file to disk
enr.SaveENRFile("C:/Program Files/Agilent/Network Analyzer/Documents/sample.enr")
The contents of the file created by this program are shown below.

[Filetype ENR]
[Version 1.0]
[Serialnumber ABCD1234]

<table>
<thead>
<tr>
<th>Frequency</th>
<th>ENR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hz</td>
<td>dB</td>
</tr>
<tr>
<td>100000000</td>
<td>14.53200</td>
</tr>
<tr>
<td>2e+010</td>
<td>15.73100</td>
</tr>
</tbody>
</table>

Last Modified:

- 14-May-2010  Fixed hostname
- 2-Aug-2007  MX New topic
Events with C++

The following code, along with the Header file, shows how to use the PNA Events.

Download the Header file 'preventcatcher.h'

```cpp
#include <atlbase.h>
#include <atcom.h>
#include <iostream>
#import "835x.tlb" no_namespace,raw_interfaces_only,named_guids
#include "pnaeventcatcher.h"

inline void HR(HRESULT hr)
{
    if (FAILED(hr))
        throw hr;
}

class MyEventCatcher : public CPNAEventCatcher
{

public:
    MyEventCatcher()
    {
        CoInitialize(NULL);
        CComPtr<IApplication> app;
        HR(app.CoCreateInstance(CLSID_Application));
        CPNAEventCatcher::SubscribeCatcher(app);
        HR(app->AllowAllEvents());
    }

    ~MyEventCatcher()
    {
        CPNAEventCatcher::Release();
        CoUninitialize();
    }
};
```
virtual void OnMeasurementEvent(long eventID, long measurementNumber) {}
virtual void OnChannelEvent(long eventID, long ch)
{
    if (eventID == 0x68070709L) // MSG_ALL_SWEEPS_COMPLETED_AND_PROCESSED
    {
        static int i = 0;
        ++i;
        std::cout << "Sweep:" << i << std::endl;
    }
}

In a .cpp file, (just like most ATL projects) you must have a declared an instance of CComModule. This will work:
CComModule _Module;

Remember that you are now the "Server" and the PNA is the Client. That makes DCOM a bit complicated.
This code was tested in VS2005 using a wizard generated MFC MDI project.

Last Modified:
13-Nov-2007 MX New topic
FOM Examples

All three VBScript examples in this topic create a FOM measurement with the following attributes:

- Sweep the Source (input) from 1 GHz to 2 GHz
- Sweep the Receivers (output) from 2 GHz to 3 GHz
- You provide an LO at 1 GHz

Learn more about Frequency Offset Mode

These programs can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as FOM.vbs. Learn how to setup and run the macro.

The following example will run on any PNA model with FOM (opt 080). However, these commands have no provisions for internal second source. It uses commands introduced before 'enhanced FOM' was released for the A.07.10 release.

```
set app = CreateObject("Agilentpna835x.application")
set chan = app.ActiveChannel
chan.startFrequency = 1e9
chan.StopFrequency = 2e9
' set the receiver frequencies to be 2e9->3e9
chan.FrequencyOffsetFrequency = 1e9
chan.FrequencyOffsetState = 1
```

The following example can be run ONLY on a PNA with revision A.07.10 or later and has FOM (opt 080). It uses new FOM commands. See FOMRange object.

```
set app = CreateObject("Agilentpna835x.application")
set chan = app.ActiveChannel
chan.startFrequency = 1e9
chan.StopFrequency = 2e9
' set the receiver frequencies to be 2e9->3e9
chan.fom("Receivers").Offset = 1e9
chan.fom.State = 1
```

The following example can be run ONLY on a PNA with a second internal source, has revision A.07.10 or later, and has FOM (opt 080). It uses the internal 2nd source for the fixed LO frequency.
set app = CreateObject("Agilentpna835x.application")
set chan = app.ActiveChannel
chan.startFrequency = 1e9
chan.StopFrequency = 2e9
' set the receiver frequencies to be 2e9->3e9
chan.fom("Receivers").Offset = 1e9
chan.fom("Source2").Coupled = 0
chan.fom("Source2").StartFrequency = 1e9
chan.fom("Source2").StopFrequency = 1e9
' turn off port coupling
chan.coupleports = 0
' set LO to 10 dBm
chan.TestPortPower(3) = 10
'Turn ON port 3, our LO signal on our 2 source PNA
chan.SourcePortMode(3) = 1
chan.fom.State = 1
Limit Line Testing with COM

This Visual Basic program:

- Turns off existing Limit Lines
- Establishes Limit Lines with the following settings:
  - Frequency range - 4 GHz to 8 GHz
  - Maximum value - (10dB)
  - Minimum value - (-30dB)
- Turns on Lines, Testing, and Sound

If using Global Pass/Fail to report limit results, trigger the PNA after configuring and enabling Limit lines.

```vbnet
Public limts As LimitTest
Set limts = meas.LimitTest
'All Off
For i = 1 To 20
    limts(i).Type = naLimitSegmentType_OFF
Next i

'Set up Limit Lines
limts(1).Type = naLimitSegmentType_Maximum
limts(1).BeginResponse = 10
limts(1).EndResponse = 10
limts(1).BeginStimulus = 4000000000#
limts(1).EndStimulus = 8000000000#
limts(2).Type = naLimitSegmentType_Minimum
limts(2).BeginResponse = -30
limts(2).EndResponse = -30
limts(2).BeginStimulus = 4000000000#
limts(2).EndStimulus = 8000000000#

'Turn on Lines, Testing, and Sound
limts.LineDisplay = 1
limts.State = 1
limts.SoundOnFail = 1
```
Modify Display Colors

This VBScript example modifies display colors, modifies trace1 colors, then saves and recalls the theme.

These programs can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as FOM.vbs. Learn how to setup and run the macro.

```vbnet
function RGB(R, G, B)
    RGB = R + G*(2^8) + B*(2^16)
end Function

shell.AppActivate "PNA Series Network Analyzer"
Set app = CreateObject("AgilentPNA835X.Application")
app.preset
Set colors = app.Preferences.DisplayColors
' Uncomment the following line to modify Print colors
'    Set colors = app.Preferences.PrintColors
colors.ResetTheme()
    colors.background = RGB(64,0,64)  ' purple
displaycolors.grid = RGB(0,255,128)  ' greenish
colors.activeLabels = RGB(0,0,255)  ' blue
colors.inactiveLabels = RGB(255,0,0)  ' red
colors.failedTraces = RGB(255,128,64) ' orange
dim Trace1
Set Trace1 = colors.Trace(1)
    Trace1.DataAndLimits = RGB(1,251,1)  ' green
    Trace1.Memory = RGB(251,1,1)  ' red
    Trace1.Markers = RGB(251,251,251)  ' white
    Trace1.MemoryMarkers = RGB(1,251,251)  ' green + blue
colors.StoreTheme("c:/Program Files/Agilent/Network Analyzer/Colors/Theme1.colors")
colors.LoadTheme("c:/Program Files/Agilent/Network Analyzer/Colors/Theme1.colors")
```
E5091 Testset Control

The following VB Script example exercises the COM commands used to control the E5091A testset.

For a description of each command, see E5091 Testsets collection.

```
Sub Main()
    Set pna = CreateObject("AgilentPNA835x.Application")
    Dim testsets As E5091Testsets
    Set testsets = pna.E5091Testsets
    Dim tset1 As E5091Testset
    Set tset1 = testsets(1)
    tset1.OutputPort(1, 3) = naE5091PortR2
    tset1.ControlLines(1) = 5
    tset1.ShowProperties = True
    tset1.Enabled = True
    MsgBox tset1.ID
    MsgBox tset1.Enabled
    MsgBox tset1.ShowProperties
    ' NumberOfPorts property returns 0 when testset not connected
    MsgBox tset1.NumberOfPorts
    MsgBox tset1.OutputPort(1, 3)
    MsgBox tset1.ControlLines(1)

    Dim tset2 As E5091Testset
    Set tset2 = testsets(2)
    tset2.Enabled = True
    tset2.ShowProperties = True
    MsgBox tset2.Enabled
    MsgBox tset2.ShowProperties
End Sub
```
Errors and the SCPIStringParser Object

This C++ program uses the SCPIStringParser.Parse command to detect the failed HRESULT and interrogate the errorInfo object for more details.

// scpierrors.cpp : Defines the entry point for the console application.
//

#include <iostream>
#include "afx.h"
#include "atlib.h"
# import "C:/program files/common files/agilent/pna/835x.tlb" raw_interfaces_only, no_namespace, named_guids
using namespace std;
HRESULT SendScpiCommand( IScpiStringParser* parser, CComBSTR& cmd, CComBSTR& response)
{
    CComBSTR bstr;
    HRESULT hr = parser->Parse(CComBSTR(cmd), &response);
    if (FAILED(hr))
    {
        // see if this interface supports ErrInfo
        CComPtr<ISupportErrorInfo> spSupportsErrInfo;
        if (SUCCEEDED(parser->QueryInterface(&spSupportsErrInfo)))
        {
            // it does, so let's get the errorinfo object
            CComPtr<IErrorInfo> spErrorInfo;
            if (SUCCEEDED(GetErrorInfo(0, &spErrorInfo)))
            {
                CComBSTR errStr;
                spErrorInfo->GetDescription(&errStr);
                std::cout << "ERROR: " << CString(errStr) << std::endl;
            }
        }
    }
    return hr;
}
int main()
{
    CoInitialize(NULL);
    {
        CComBSTR response;
        CComPtr<IApplication> spPNA;
        CComPtr<IScpiStringParser> spSCPI;
        if (SUCCEEDED(spPNA.CoCreateInstance(CLSID_Application)))
        {
            spPNA->get_ScpiStringParser(&spSCPI);
            SendScpiCommand(spSCPI, CComBSTR("SYSTEM:PRESET"), response);
            SendScpiCommand(spSCPI, CComBSTR("CALC:PAR:CAT?"), response);
            std::cout << CString(response) << std::endl;
            SendScpiCommand(spSCPI, CComBSTR("THIS:IS:A:SYNTAX:ERROR"), response);
        }
    }
    CoUninitialize();
    return 0;
}
**External Testset Control**

The following VB Script example exercises the COM commands used to control the Z5623AK64 testset. For a description of each command, see TestsetControl Object.

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as Testset.vbs. Learn how to setup and run the macro.

```vbnet
' Demonstrate some COM commands for external testsets.
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Sub DemoTestset(na)
    Dim testsets, tset1
    Dim portNum
    Dim chNum, address
    Set testsets = na.ExternalTestsets
    chNum = 1

    ' Load a configuration file.
    ' NOTE: the K64 testset is only compatible with 4-port analyzers.
    address = 0
testsets.Add "Z5623AK64", address

    ' Get the testset object
    ' in the testsets collection.
    Set tset1 = testsets(1)

    ' Show the selections available for each port.
    For portNum = 1 To 4
        MsgBox("Port " & CStr(portNum) & " catalog: " & tset1.PortCatalog(portNum))
    Next

    ' Set port mappings on channel 1.
tset1.OutputPorts(chNum) = "5 ext R,2 int R,3 int R,6 int R"
End Sub
```
' Set control lines.
tset1.ControlLines(chNum) = 85

' Set label.
tset1.Label(chNum) = "Some label"

' Enable external testset control. This automatically enables status bar display as well.
tset1.Enabled = True
End Sub

' The testset used in this demo is only usable on 4-port analyzers
If (pna.NumberOfPorts <> 4) Then
MsgBox("This program only runs on 4-port analyzers.")
Else
DemoTestset(pna)
End If
PathConfiguration Example

**Note:** These commands are accessible only for PNA-X models.

These Visual Basic and C# examples exercise various commands on the:

- PathConfigurationManager Object
- PathConfiguration Object
- PathElement Object

**VB Example**

```vbnet
' Create / Get the PNA application
Dim app
Set app = CreateObject("AgilentPNA835x.Application")

' Preset the instrument
app.Preset

' Get a channel interface on which to operate
Dim chan
Set chan = app.ActiveChannel

' Modify the Default configuration, and save it as "My Config"
chan.PathConfiguration = "Default"

' Set the "Combiner" element to value "Reversed"
chan.PathConfiguration.Element("Combiner").Value = "Reversed"

' Set the "Src1" element to value "High Power"
chan.PathConfiguration.Element("Src1").Value = "High Power"

' Change the description text
chan.PathConfiguration.DescriptionText = "Connect J8 to J9."

' Store the modified configuration
chan.PathConfiguration.Store ("My Config")

' Set the instrument’s path config back to the default (req. 8)
chan.PathConfiguration = "Default"

' Load a previously saved configuration onto channel 1
app.PathConfigurationManager.Load 1, "My Config"
```

**C# Example**

```csharp
// Create / Get the PNA application
Dim app
Set app = CreateObject("AgilentPNA835x.Application")

// Preset the instrument
app.Preset

// Get a channel interface on which to operate
Dim chan
Set chan = app.ActiveChannel

// Modify the Default configuration, and save it as "My Config"
chan.PathConfiguration = "Default"

// Set the "Combiner" element to value "Reversed"
chan.PathConfiguration.Element("Combiner").Value = "Reversed"

// Set the "Src1" element to value "High Power"
chan.PathConfiguration.Element("Src1").Value = "High Power"

// Change the description text
chan.PathConfiguration.DescriptionText = "Connect J8 to J9."

// Store the modified configuration
chan.PathConfiguration.Store ("My Config")

// Set the instrument’s path config back to the default (req. 8)
chan.PathConfiguration = "Default"

// Load a previously saved configuration onto channel 1
app.PathConfigurationManager.Load 1, "My Config"
```
Type pnaType = Type.GetTypeFromProgID("AgilentPNA835x.Application", "PNA-NAME-HERE");

AgilentPNA835x.Application pna = (AgilentPNA835x.Application)Activator.CreateInstance(pnaType);

AgilentPNA835x.Channel chan = (AgilentPNA835x.Channel)pna.ActiveChannel;

// Preset the Instrument
pna.Preset();

// Modify the Default configuration, and save it as "My Config"
chan.set_PathConfiguration("Default");

// Set the "Combiner" element to value "Reversed"
chan.get_PathConfiguration().get_Element("Combiner").Value = "Reversed";

// Change the description text
chan.get_PathConfiguration().DescriptionText = "Connect J8 to J9."

// Store the modified configuration
chan.get_PathConfiguration().Store("My Config");

// Set the instrument’s path config back to the default (req. 8)
chan.set_PathConfiguration("Default");

// Load a previously saved configuration onto channel 2
pna.PathConfigurationManager.LoadConfiguration(1, "My Config");
Create a Narrowband Pulsed Measurement using the PNA-X and COM

The following COM example demonstrates how to create a narrowband pulsed measurement using the Pulsed Application DLL on the PNA-X.

See the example program for the E836x PNA models.

See the example program for wideband pulsed measurements on the PNA-X.

It first gets valid configuration settings and then uses those settings to configure the PNA and internal pulsed generators.

To run this program, you need:

- PNA-X
- Pulsed Application (Option H08)

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as BalancedCOM.vbs. Learn how to setup and run the macro.

See Also

- Learn how to install and register the pulsed .dll on your PC
- See the ConfigEnhancedNB2 method for sending and returning parameters to the .dll.
- See the ConfigEnhancedNBIFAtten method for setting the receiver IF gain.
- See the COM IF Configuration commands used in the program.
- See the equivalent SCPI IF Configuration commands.

Interfaces

```vbs
Dim OApp As AgilentPNA835x.Application
Dim OIntPG As AgilentPNA835x.PulseGenerator
Dim OPathConf As AgilentPNA835x.PathConfiguration
Dim OFilter As AgilentPNA835x.SignalProcessingModuleFour
Dim OIF As AgilentPNA835x.IFConfiguration

'Pulsed parameters
Dim DPRF As Double
Dim DBW As Double
Dim DPhysicalIF As Double
Dim DNCO As Double
Dim DCF As Double
```
Dim DGD As Double
Dim DGW As Double
Dim DSWG As Double
Dim DSWG As Double
Dim DSWGD As Double
Dim DSWGW As Double
Dim DSWGR As Long
Dim LStage1TapArray() As Long
Dim LStage2TapArray() As Long
Dim LStage3TapArray() As Long
Dim BFixedPRF As Boolean
Dim IIFAtten As Integer
'pulsed DLL interface
Dim OPulsed As New AgilentPNAPulsed.application

'Pulsed settings
DPRF = 5000 'Hz
DBW = 500 'Hz
BFixedPRF = True
DNCO = 0#
DCF = 0#
DGD = 0#
DGW = 0.000001
DSWGR = 0#

'Send desired pulsed parameters to the pulsed configuration DLL. The DLL will return a new set of pulse parameters to send to the PNA.
OPulsed.ConfigEnhancedNB2 DPRF, DBW, DPhysicalIF, DNCO, DCF, LStage1TapArray, LStage2TapArray, LStage3TapArray, BFixedPRF, DGD, DGW, DSWG, DSWG, DSWG

'Send configuration to PNA

'Connect to the PNA application
Set OApp = CreateObject("AgilentPNA835x.Application")

'Create instance of pulse generators on active channel
Set OIntPG = OApp.ActiveChannel.PulseGenerator

'Create instance of path configuration on active channel
Set OPathConf = OApp.ActiveChannel.PathConfiguration

'Create instance of digital filter on active channel
Set OIF = OApp.ActiveChannel.IFConfiguration

'Create instance of Hana digital filter on active channel
Set OFilter = OApp.ActiveChannel.SignalProcessingModuleFour

' Set up master pulse period for internal pulse generators
OIntPG.Period = 1 / DPRF

' Set up internal pulse generator output #1 to drive internal source modulation
OIntPG.Width(1) = 0.0001 '100us
OIntPG.Delay(1) = 0.00001 '10us
OIntPG.State(1) = True

OPathConf.Element("PulseModDrive").Value = "Pulse1"

' Set up internal pulse generator output #2 to drive internal receiver gates for a 2 port PNA-X
OIntPG.Width(2) = 0.000001 '1us
OIntPG.Delay(2) = 0.00005 '50us
OIntPG.State(2) = True

OPathConf.Element("IFGateA").Value = "Pulse2"
OPathConf.Element("IFGateB").Value = "Pulse2"
OPathConf.Element("IFGateR1").Value = "Pulse2"
OPathConf.Element("IFGateR2").Value = "Pulse2"

' Configure PNA in pulsed mode operation

' Turn off ALC and turn on modulator control
OApp.ActiveChannel.ALCLevelingMode(1) = naALCOpenLoop 'Source 1 output #1 ALC off
OPathConf.Element("Src1Out1PulseModEnable").Value = "Enable" 'Enable Source 1 pulse modulator

' Set path and enable IF gates
OApp.ActiveChannel.IFBandwidth = DBW
OPathConf.Element("IFSigPathAll").Value = "NBF"

' Set filter stages based on pulse parameters
OIF.IFFrequency = DPhysicalIF
OIF.IFFrequencyMode = naMANUAL
OFilter.Stage1Frequency = DNCO
OFilter.Stage1Coefficients = LStage1TapArray
OFilter.Stage2Coefficients = LStage2TapArray
OFilter.Stage3FilterType = "RECT"
OFilter.Stage3Parameter("C") = LStage3TapArray(0)
OFilter.FilterMode = naMANUAL

' Set receivers to auto gain setting
OPulsed.ConfigEnhancedNBIFAtten DPRF, DGW, IIFAtten '1us pulse width
OPathConf.Element("NBFATNA").Value = IIFAtten
OPathConf.Element("NBFATNB").Value = IIFAtten
OPathConf.Element("NBFATNR1").Value = IIFAtten
OPathConf.Element("NBFATNR2").Value = IIFAtten
MsgBox "Done"

Last Modified:

11-Jun-2007   Rev 2- some edits
16-Feb-2007   MX New topic
Setup Basic Measurements

This VBScript program sets up four basic s-parameter measurements in four windows, all in a single channel. Handles are created to the measurement, channel, and window objects so that subsequent settings can be made for each.

**Note:** This is only an example. This is not necessarily the most efficient way to make basic S-parameter measurements.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Basic.vbs. Learn how to setup and run the macro.

See PNA Object Model

See CreateSParameterEx

See Other COM Example Programs

```vbscript
Set pna = CreateObject("AgilentPNA835x.Application")
pna.Preset
' Get a handle to the preset channel 1, S11 meas, and window(1)
set meas1=pna.ActiveMeasurement
set chan1=pna.ActiveChannel
set win1=pna.ActiveNAWindow
' Creates a new S21 measurement in New window(2)
pna.CreateSParameterEx 1,2,1,2,2
set meas2=pna.ActiveMeasurement
set win2=pna.ActiveNAWindow
' Creates a new S12 measurement in New window(3)
pna.CreateSParameterEx 1,1,2,1,3
set meas3=pna.ActiveMeasurement
set win3=pna.ActiveNAWindow
' Creates a new S22 measurement in New window(4)
pna.CreateSParameterEx 1,2,2,2,4
set meas4=pna.ActiveMeasurement
set win4=pna.ActiveNAWindow
'Make settings
'set Stop Frequency for channel
chan1.StopFrequency=1e9
' set Display formats
meas1.format=1 'Lin Mag
```
meas2.format=2 'Log Mag
meas3.format=3 'Phase
meas4.format=4 'Smith
'Show title in all windows
win1.title="Win #1"
win2.title="Win #2"
win3.title="Win #3"
win4.title="Win #4"

Last Modified:
19-Apr-2010  Modified example for same channel
**Setup Compression Marker**

This example program does the following:

- Creates a compression marker
- Queries the Power Out and Power In values

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as CompMkr.vbs. Learn how to setup and run the macro.

See the FIFO object.

---

**See Other COM Example Programs**

```vbscript
Set app = CreateObject("AgilentPNA835X.Application")
set meas = app.activemeasurement

'get the COM marker object
'and create marker1
set mark = meas.marker(1)

'set the compression level
mark.compressionlevel = 1.5

'make it a compression marker
'and find the compression point
mark.searchcompressionpoint

'return power out and power in
'power in
dim answer
answer = mark.compressionpin
wscript.echo("pin: " & answer)

'power out
answer = mark.compressionpout
wscript.echo("pout: " & answer)
```

---

Last Modified:

12-Feb-2009   MX New topic
Set Up an Embedded LO Measurement

This VBScript example creates a Converter measurement for a converter with an Embedded LO.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Noise.vbs. Learn how to setup and run the macro.

```vbscript
option explicit
dim app
dim chan
dim host
host = <your PNA computer name here>
set app = CreateObject("Agilentpna835x.application", host)
app.reset
' create NFX traces
app.CreateCustomMeasurementEx 1, "Noise Figure Converters", "NF", 1
app.CreateCustomMeasurementEx 1, "Noise Figure Converters", "SC21", 1
'set channel and application objects
set chan = app.ActiveChannel
set nfx = chan.CustomChannelConfiguration
dim converter
set converter = chan.GetConverter()
dim calMgr
set calMgr = app.GetCalManager
dim nfxCal
set nfxCal = CalMgr.CreateCustomCalEx(1)
dim nfxCalExt
set nfxCalExt = nfxCal.CustomCalConfiguration
'dim ELO
'set ELO = converter.ConverterEmbeddedLO
' Set embedded LO properties
ELO.NormalizePoint = 101
ELO.TuningMode = 0 ' Broadband and precise
ELO. TuningIFBW = 3.0e4
ELO.MaxPreciseTuningIterations = 5
```
ELO.PreciseTuningTolerance = 1
ELO.TuningSweepInterval = 1
ELO.IsOn = true
chan.Single 1

Last Modified:
27-Oct-2009    MX New topic
Setup FastCW and FIFO

This example program does the following:

- Setup an A/R and B/R measurement
- Turn ON point averaging
- Set external edge triggering (commented out)
- Set FIFO and Fast CW
- Write data into FIFO data buffer
- Read FIFO data buffer

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as FIFO.vbs. Learn how to setup and run the macro. See the FIFO object.

See Other COM Example Programs

```vbs
dim app
set app = createobject("Agilentpna835x.application")
' Setup and measure A/R and B/R
app.Reset
app.CreateMeasurement 1,"A/R1",0
app.CreateMeasurement 1,"B/R1",0
' Set IFBW to 600khz (400thousand pts/second)
app.activeChannel.IFBandwidth = 600e3
' Point Averaging Count = 10
app.activeChannel.AverageMode = 0 ' point
app.activeChannel.averagingFactor = 10
app.ActiveChannel.averaging = 1 ' turn on
' Edge triggering - positive edge
'app.TriggerSetup.ExternalTriggerConnectionBehavior(1) = 2 ' BNC1 = trigger positive edge
'app.TriggerSetup.Source = 2 ' external
'app.ActiveChannel.TriggerMode = 0 'point
' Setup FIFO and Fast CW count
```
app.ActiveChannel.Hold 1 ' hold - synchronous
app.FIFO.State = 1' turn on FIFO
app.FIFO.Clear
app.activechannel.sweeptype = 3 ' CW sweep
app.activechannel.fastcwpointcount = 1000000' set the point count to 1million
app.activechannel.single 1 ' synchronous single
' the single will wait until the end of sweep.
'You do not have to wait until end of sweep to start emptying FIFO.
points = app.fifo.datacount
msgbox points
' points == 2000000 ' points = 2million. Took 5 seconds to acquire
For I = 0 to 1 ' 2 iterations (2 parameters * 2 sets of 1 million)
Dim data
Data = app.fifo.data(1000000)
Next
msgbox data(0)
Setup Noise Figure Port Mapping

This program demonstrates how to change source and receive ports when measuring noise figure. It assumes that option 029 ("Fully Corrected Noise Figure") is installed.

If only option 028 ("Noise figure measurements using standard receivers") is installed, switching ports is simpler, since only one noise receiver selection is available.

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as NF.vbs. Learn how to setup and run the macro.

See Also
Noise Figure Object
Create and Cal a NoiseFigure Measurement

See Other COM Example Programs

```vbnet
option explicit
' Noise receiver enumerations
dim naStandardReceiver, naNoiseReceiver
' standard PNA receiver
naStandardReceiver = 0
' dedicated noise receiver (option 029 only)
aNoiseReceiver = 1
dim pna, windowNum, channelNum
set pna = CreateObject("Agilentpna835x.application")
windowNum = 1
channelNum = 1
pna.Reset
' Create Noise Figure measurement
dim noise,noiseChan,noiseConfig
set noise = pna.CreateCustomMeasurementEx(channelNum, "Noise Figure Cold Source", "NF", windowNum)
set noiseChan = pna.ActiveChannel
' provides access to noise-specific channel properties
set noiseConfig = noiseChan.CustomChannelConfiguration
'To change from the default input/output port settings of
' source port = PNA1, receive port = PNA2,
' you must first change the noise receiver,
```
' then select the desired ports.
dim srcPort, rcvPort
' set port mapping to source port = PNA3, receive port = PNA4
srcPort = 3
rcvPort = 4
' use PNA receiver for noise measurements
noiseConfig.NoiseReceiver = naStandardReceiver
noiseConfig.SetPortMap srcPort, rcvPort
' To revert back to using the noise receiver, the source
' and receive ports must be set to their default values
' BEFORE switching to the noise receiver.
' Otherwise, a COM exception will be thrown.
' restore defaults: source=PNA1, receiver=PNA2
noiseConfig.SetPortMap 1,2
' use dedicated noise receiver for noise measurements
noiseConfig.NoiseReceiver = naNoiseReceiver
Setup Phase Control

The following VB Script example exercises the COM commands used to setup and display Phase Sweep measurements.

See Also
About Phase Control
PhaseControl Object

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as RxLevel.vbs. Learn how to setup and run the macro.

```vbs
'Assume port 1 is connected to port 3
Set pna = CreateObject("AgilentPNA835x.Application")
pna.Preset
Set chan = pna.ActiveChannel
chanNum = chan.ChannelNumber
'Create 3 traces: S33, R3/C(amp), R3/C(phase)
pna.CreateMeasurement 1,"S33",3
Set meas1 = pna.ActiveMeasurement
meas1.Format = 4 'Smithchart format
pna.CreateMeasurement 1,"R3/C",3 'Log format
Set meas2 = pna.ActiveMeasurement
meas2.Format = 1 'Phase format
Set meas = pna.ActiveMeasurement
meas.Format = 2 'Phase format
'turn on 3 and 1
chan.SourcePortMode(1) = 1
chan.SourcePortMode(3) = 1
chan.SweepType = 5 'Phase sweep
Set phase = chan.PhaseControl
'set port3's control parameter to R3/C
phase.PhaseParameter(3) = "R3/C"
'notice the reference port should not included in the parameter
```
phase.PhaseReferencePort(3) = 1

'Set port3 to PAR mode

phase.PhaseControlMode(3) = 1 'PhaseControlParameter mode

phase.FixedRatioedPower(3) = 3

phase.StartPhase(3) = 0

phase.StopPhase(3) = 180
Setup PNOP and PSAT Marker Search

This example program does the following:

- Sets up measurement for either PNOP or PSAT marker search
- Sets parameters for search
- Reads a parameter for each

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as SearchMkr.vbs. Learn how to setup and run the macro.

```
Set app = CreateObject("AgilentPNA835X.Application")
app.Preset
set meas = app.activemeasurement
'View Power Out vs Power In
meas.ChangeParameter "B", 1
'perform power sweep
set chan = app.ActiveChannel
chan.SweepType = 2
chan.StartPower = -5
chan.StopPower = 0
'-------------------
'Choose marker search
resp=Msgbox ("PNOP (yes) or PSAT (no)", 4, "PNA Marker Search Demo")
if resp=6 then
  PNOP1()
Else
  PSAT1()
End If
'-------------------
'PSAT marker search
Sub PSAT1()
set psat = meas.PSaturation
```
psat.PMaxBackOff = .3
psat.SearchPowerSaturation

' Read PSAT Parameter
dim answer
answer=psat.GainSaturation
wscript.echo("Gain Sat: ", answer)
End Sub

'--------------------
' PNOP marker search
Sub PNOP1()
set pnon = meas.PNOP
pnon.BackOff = 2
pnon.PinOffset = 1
pnon.SearchPowerNormalOperatingPoint

' Read PNOP Parameter
dim answer
answer=pnon.Gain
wscript.echo("PNOP Gain: ", answer)
End Sub
Setup Receiver Leveling

The following VB Script example exercises the COM commands used to setup Receiver Leveling.

See Also
About Receiver Leveling
RxLevelingConfiguration Object

See Other COM Example Programs

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save on the PNA hard drive as RxLevel.vbs. Learn how to setup and run the macro.

```
' Demonstrate some COM commands for Receiver Leveling.
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Dim chan
Set chan = pna.ActiveChannel
Dim RxLevel
Set RxLevel = chan.GetRxLevelingConfiguration
Dim srcPort
srcPort = 1
pna.Preset
RxLevel.ReferenceReceiver(srcPort) = "R1"
RxLevel.Tolerance(srcPort)= 0.02
RxLevel.IterationNumber(srcPort)= 10
RxLevel.FastMode(srcPort)=True
RxLevel.LevelingIFBW(srcPort)= 100
RxLevel.PowerOffset(srcPort)= 0
RxLevel.PowerMax(srcPort)= 20
RxLevel.PowerMin(srcPort)= -50
RxLevel.SafeMode(srcPort)= True
RxLevel.State(srcPort)= True
```

Last Modified:
Show Custom Cal Windows during a Guided Calibration

This VBScript program shows how to send commands that allow you to view specific 'custom' windows, and sweep specific channels, during a UI (Cal Wizard) or remote calibration.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as CalWindow.vbs. Learn how to setup and run the macro.

These commands are used to show and sweep specific windows and channels:

- AllowChannelToSweepDuringCalAcquisition Method
- DisplayNAWindowDuringCalAcquisition Method
- DisplayOnlyCalWindowDuringCalAcquisition Method
- SweepOnlyCalChannelDuringCalAcquisition Method

The following command sweeps the Cal Windows before remote acquisition:

- SetupMeasurementsForStep Method

See Other COM Example Programs

```vbs
Set pna = CreateObject("AgilentPNA835x.Application")
pna.Preset
' get a handle to the preset channel 1 so that we can later cal it
set meas=pna.ActiveMeasurement
' Creates a new S21 measurement in channel 2 and New window(2)
' this will be the channel and window to show during cal
pna.CreateSParameterEx 2,2,1,2,2
Set calMgr = pna.GetCalManager
Set guidedCal = calMgr.GuidedCalibration
' show window 2 during cal
calMgr.DisplayNAWindowDuringCalAcquisition 2,True
' sweep channel 2 during calibration of chan 1
calMgr.AllowChannelToSweepDuringCalAcquisition 1,2,True
' make Channel1 the active channel
' activating the measurement also activates the channel
meas.Activate
```
```vba
guidedCal.Initialize 1, True
' Do 2-port cal
' Select the connectors
guidedCal.ConnectorType(1) = "APC 3.5 female"
guidedCal.ConnectorType(2) = "APC 3.5 male"
' Select the Cal Kit for each port being calibrated.
guidedCal.CalKitType(1) = "85052D"
guidedCal.CalKitType(2) = "85052D"
' Initiate the calibration and query the number of steps
numSteps = guidedCal.GenerateSteps
' Measure the standards, compute and apply the cal
value = MsgBox("Number of steps is " + CStr(numSteps))
' Measure the standards
For i = 1 to NumSteps
    step = "Step " + CStr(i) + " of " + CStr(numSteps)
    strPrompt = guidedCal.GetStepDescription(i)
    ' Sweep the Cal window prior to standard acquisition
    guidedCal.SetupMeasurementsForStep i
    ' prompt to connect standard
    value = MsgBox(strPrompt, vbOKOnly, step)
    ' measure standard
    guidedCal.AcquireStep i
Next
' Conclude the calibration
guidedCal.GenerateErrorTerms
MsgBox ("Cal is complete!")
' clear the Cal window and channel flags
calMgr.DisplayOnlyCalWindowDuringCalAcquisition
calMgr.SweepOnlyCalChannelDuringCalAcquisition
```
COM Events Example

This Visual Basic program shows how to monitor the end of sweep. The program will set sweep time to various amounts and BEEPs when sweep is completed. This method allows other processes to continue while waiting for end-of-sweep. This program stops after 10 loops.

Note: To avoid Permission Denied problems, this should be run on the PNA and not a PC. To run it from a PC both units must be "trusted" and on the same domain/workgroup.

```vbnet
Option Explicit
Dim na As AgilentPNA835x.Application
Dim WithEvents naEvnt As AgilentPNA835x.Application
Dim ch As AgilentPNA835x.Channel
Dim sweepComplete As Boolean

Private Sub Form_Load()

Dim N As Integer
Set na = CreateObject("AgilentPNA835x.application")
n.a.preset
Set ch = na.ActiveChannel
na.DisallowAllEvents ' Turn off all events
Set naEvnt = na ' Enable event interrupts
Do
N = N + 1 ' Loop counter
ch.sweepTime = 1 + (Rnd * 9) ' Set random sweep-time from 1-10 sec
sweepComplete = False
ch.Single False ' Trigger sweep
naEvnt.AllowEventCategory naEventCategory_CHANNEL, True ' Enable Channel event
Do
DoEvents ' Allows other processes to continue
Loop Until sweepComplete = True
naEvnt.AllowEventCategory naEventCategory_CHANNEL, False ' Disable event until ready for next one
Beep ' Do end-of-sweep processing here;
Loop Until N > 10
End

End Sub

Private Sub naEvnt_OnChannelEvent(ByVal eventID As Variant, ByVal chNumber As Variant)
' In this example we don't care about the channel info
If eventID = naEventID_CHANNEL_TRIGGER_COMPLETE Then sweepComplete = True
End Sub
```
Upload a Segment Table using C++

This example program shows the Variant method for uploading a segment sweep to the PNA using the `SetAllSegments` method.

```cpp
#include "stdafx.h"
#include <stdio.h>
#include "atlbase.h"
#include "objbase.h"

// import the PNA type library
#include "C:/Program Files/Common Files/Agilent/PNA/835x.tlb" no_namespace, named_guids

int _tmain(int argc, _TCHAR* argv[])
{
    // interface pointers to retrieve COM interfaces
    HRESULT hr;
    IUnknown* pUnk = 0;
    IApplication* pNA = 0;
    IChannel* pChan = 0;
    IMeasurement* pMeas = 0;
    IMeasurement5* pMeas5 = 0;
    IArrayTransfer* pTrans = 0;
    ISegments* pSeg = 0;
    ISegments2* pSeg2 = 0;

    //Variables for X and Y data read portion
    SAFEARRAY* sArray;
    _variant_t vXVals;
    double HUGEP* xVals;
    float* pScalarData;

    //Variables for Segment portion
    double Fstart, Fstop, SegWidth;
    long i[2];
```
int num_points = 11;
SAFEARRAY* pSA;
VARIANT vSeg;
VARIANT v;

int NUM_SEGS = 10;
int SEG_SIZE = 7;

//Create SafeArray to hold the segment data
SAFEARRAYBOUND aDim[2]; //This must be 2, the PNA expects to see a 2-dimensional array
aDim[0].lLbound = 0;
aDim[0].cElements = SEG_SIZE; //This will be set to 7 unless port power is uncoupled
aDim[1].lLbound = 0;
aDim[1].cElements = NUM_SEGS;
pSA = SafeArrayCreate(VT_VARIANT, 2, aDim); //The cDim parameter must be set to 2 as the PNA expects a 2D array

//Init Variant to set values in Safearray
VariantInit(&vSeg);

Fstart = 10e6;
Fstop = 3e9;
SegWidth = (Fstop - Fstart) / NUM_SEGS;

//Loop to write segment data
for(int j = 0; j < NUM_SEGS; ++j)
{
    i[1] = j; //Set Segment #

    //Segment Definition
    i[0] = 0;
    vSeg.vt = VT_BOOL; //First parameter is Boolean
    vSeg.boolVal = VARIANT_TRUE; //Segment State
    SafeArrayPutElement(pSA, i, &vSeg);
    i[0] += 1;
}
vSeg.vt = VT_I4; //Second parameter is an integer
vSeg.intVal = num_points; //Number of Points
SafeArrayPutElement(pSA, i, &vSeg);
i[0] += 1;

vSeg.vt = VT_R8; //Remaining parameters are of type double
vSeg.dblVal = Fstart+j*SegWidth; //Start Frequency
SafeArrayPutElement(pSA, i, &vSeg);
i[0] += 1;
vSeg.dblVal = vSeg.dblVal+SegWidth; //Stop Frequency
SafeArrayPutElement(pSA, i, &vSeg);
i[0] += 1;
vSeg.dblVal = 1.0e3; //IF Bandwidth
SafeArrayPutElement(pSA, i, &vSeg);
i[0] += 1;
vSeg.dblVal = 0.0; //Dwell time
SafeArrayPutElement(pSA, i, &vSeg);
i[0] += 1;
vSeg.dblVal = -5.0; //Power
SafeArrayPutElement(pSA, i, &vSeg);
}

//vSeg no longer needed, clean up
VariantClear(&vSeg);

//Declare Variant to use with Segment data
VariantInit(&v);
v.vt = VT_ARRAY|VT_VARIANT;
v.parray = pSA; //write safearray to variant

// Initialize the COM subsystem
CoInitialize(NULL);
CoInitializeSecurity(NULL, //security descriptor
-1, // authn svc entries
NULL, // authn svcs
NULL, // reserved
RPC_C_AUTHN_LEVEL_NONE,
RPC_C_IMP_LEVEL_IMPERSONATE,
NULL,        // authn info
0,          // capabilities
NULL);      // reserved

// Create an instance of the network analyzer
// Request the NA's IUnknown interface
hr = CoCreateInstance(CLSID_Application, 0, CLSCTX_ALL, IID_IUnknown, (void**) &pUnk);
if (!FAILED(hr))
{
    // QueryInterface for the INetworkAnalyzer interface of the NetworkAnalyzer object
    hr = pUnk->QueryInterface(IID_IApplication, (void**) &pNA);
    if (!FAILED(hr))
    {
        // Reset the analyzer to instrument preset
        pNA->Reset();

        // Create S11 measurement
        pNA->CreateSParameter(1, 1, 1, 1);

        // Set pChan variable to point to the active channel
        pNA->get_ActiveChannel(&pChan);

        // Show Segment table
        pNA->NAWindows->Item(1)->ShowTable((NATableType)2);

        // Get handle to ISegments Interface
        pChan->get_Segments(&pSeg);

        // Get handle to ISegments2 Interface
        hr = pSeg->QueryInterface(IID_ISegments2, (void**) &pSeg2);

        // Set Segment Sweep Options
pSeg2->IFBandwidthOption = VARIANT_TRUE;
pSeg2->SourcePowerOption = VARIANT_TRUE;

// Push segments to PNA
pSeg2->SetAllSegments(v);

// Set Sweep Type to Segment Sweep
pChan->SweepType = naSegmentSweep;

if (pChan)
{
    // Set pMeas variable to point to the active measurement
    pNA->get_ActiveMeasurement(&pMeas);

    if(pMeas)
    {
        // Setup the channel for a single trigger
        pChan->Hold(true);
        pNA->TriggerSignal = naTriggerManual;
        pChan->TriggerMode = naTriggerModeMeasurement;

        // Make the PNA application visible
        pNA->put_Visible(true);

        // Send a manual trigger to initiate a single sweep
        pChan->Single(true);

        // QueryInterface for the IArrayTransfer interface of the
        NetworkAnalyzer object
        hr = pMeas->QueryInterface(IID_IArrayTransfer,(void**)&pTrans);

        // Get handle for IMeasurement5 interface
        hr = pMeas->QueryInterface(IID_IMeasurement5, (void**)&pMeas5);

        if (!FAILED(hr))
        {
            int val = num_points*NUM_SEGS;
        }
    }
}
// Store the data in the "result" variable
pScalarData = new float[val];
xVals = new double[val*2];

// Get X axis values
vXVals = pMeas5->GetXAxisValues();

// Convert _variant_t array to a SAFEARRAY
sArray = vXVals.parray;

// Convert data from SAFEARRAY to double array. Each SAFEARRAY value is 16 bytes so it takes up 2 floats so the xVals size is double the number of points. This also means that every other data point in the resulting array can be discarded.
hr = SafeArrayAccessData(sArray, (void HUGEP**)&xVals);

// Get Measurement Values
pTrans->getScalar(naRawData, naDataFormat_LogMag, (long *)&val, pScalarData);

// Display the result
printf("S11(dB) - Visual C++ COM Example for PNA operating in segment sweep mode/n/n");
for (int j = 0; j < val; j++)
{
    // Write value... the xVals array is offset by 1 in each data point since the return data is 16 bytes and each double is 8.
    printf("%.3lf GHz, %.4f/n", xVals[2*j+1]/1e9, pScalarData[j]);
}
}
else
{
    printf("Programmed failed to connect to the PNA.");
}
}
CoUninitialize();

system("PAUSE");
return 0;
Using C#

The following are common C# examples:

**Connecting to a specific PNA via DCOM:**

```csharp
public AgilentPNA835x.Application Connect(string hostname)
{
    AgilentPNA835x.Application pna = null;
    try
    {
        Type t = Type.GetTypeFromProgID("AgilentPNA835x.Application", hostname, true);
        pna = (AgilentPNA835x.Application) Activator.CreateInstance(t);
    }
    catch (Exception e)
    {
        HandleExceptions(e);
    }
    return pna;
}
```

**Using the GetData Interface**

```csharp
AgilentPNA835x.IMeasurement meas = app.ActiveMeasurement;
    object[] dataArrayAsObj;
    dataArrayAsObj = (object[])meas.getData(AgilentPNA835x.NADataStore.naMeasResult,
    AgilentPNA835x.NADataFormat.naDataFormat_LogMag);
    float[] dataArray = new float[dataArrayAsObj.Length];
    for (int j = 0; j < dataArrayAsObj.Length; j++)
    {
        dataArray[j] = (float)dataArrayAsObj[j];
    }
```

**2-dimensional GetData**

2201
AgilentPNA835x.IMeasurement meas = app.ActiveMeasurement;
    app.ActiveChannel.Single(true);
    object[,] dataArrayAsObj;
    dataArrayAsObj = (object[,])meas.getData(AgilentPNA835x.NADataStore.naRawData,
    AgilentPNA835x.NADataFormat.naDataFormat_Smith);
    float[,] dataArray = new float[dataArrayAsObj.Length,2];
    for (int j = 0; j < dataArrayAsObj.Length; j++)
    {
        dataArray[j,0] = (float)dataArrayAsObj[j,0];
        dataArray[j,1] = (float)dataArrayAsObj[j,1];
    }

Other C# / .NET Topics

Perform a Guided Cal with CSharp
Getting a handle to the Noise Figure Cal object.
Using .NET

Last Modified:

15-Sep-2009 Modified GetData example
Configure for COM-DCOM Programming

Before developing or running a COM program, you should first establish communication between your PC and the analyzer. This process is referred to as gaining Access to the analyzer. You should then register the PNA type library on your PC.

DCOM (Distributed Component Object Model) refers to accessing the PNA from a remote PC. COM refers to accessing the PNA application from the analyzer PC.

- **Access Concepts**
- **Access Procedures**
- **Register the PNA Type Library on Your PC**
- **Problems?**

**Note:** After performing a Firmware Upgrade you must copy the new type library to your development PC to get access to new COM commands. See Register the analyzer on your PC.

**Other Topics about COM Concepts**

For detailed information on this subject, see [http://na.tm.agilent.com/pna/DCOMSecurity.html](http://na.tm.agilent.com/pna/DCOMSecurity.html)

**Note:** ONLY 32-bit compiler option is supported (64-bit is NOT supported).

**Access Concepts**

PNAs are shipped from the factory such that Everyone has permission to launch and access the PNA application via COM/DCOM. The term **Everyone** refers to a different range of users depending on whether the PNA is a member of a **Domain** or **Workgroup** (it must be one or the other; not both). By default, the PNA is configured as members of a workgroup. Therefore, **Everyone** includes only those users who have been given logon accounts on the PNA.

**Workgroup**

A workgroup is established by the **PNA administrator** declaring the workgroup name and declaring the PNA as a member of the workgroup. A workgroup does not require a network administrator to create it or control membership.

**Everyone** includes only those users who have been given logon accounts on the PNA.

By default, the PNA is configured as members of a workgroup named WORKGROUP.

**Note:** To setup a logon account for a new user, see Additional Users.

The easiest method of gaining DCOM access, is to make the user's account name and password on the PNA to EXACTLY match their PC logon account name and password.

**Domain**

A domain is typically a large organizational group of computers. Network administrators maintain the domain and
control which machines have membership in it. **Everyone** includes those people who have membership in the domain. In addition, those with logon accounts can also access the analyzer.

**Summary**

- A **Workgroup** requires no maintenance, but allows DCOM access to only those users with a log-on account for the PNA.

- A **Domain** requires an administrator, but all members of the domain and those with logons to the analyzer are allowed DCOM access to the PNA.

The following section "Access Procedures" provides a tighter level of security allowing only selected (not **Everyone**) domain and workgroup users DCOM **Access** and **Launch** capability of the PNA.

**Access Procedures**

Perform this procedure for the following reasons:

- To allow only selected users (not everyone) remote Access and remote Launch capability to the PNA. Launch capability is starting the PNA application if it is not already open.
- To verify that you have DCOM access to the analyzer.

**Note:** Before doing this procedure, you must first have a logon account on the PNA. See [PNA User Accounts](#)

The following procedure grants specific users DCOM access and launch capability of the PNA application: To perform this procedure, you must first minimize the PNA application.

**How do I know which Operating System I have?**

<table>
<thead>
<tr>
<th>Windows 2000</th>
<th>Windows XP</th>
</tr>
</thead>
<tbody>
<tr>
<td>On the PNA, click the Windows <strong>Start</strong> button</td>
<td>On the PNA, click the Windows <strong>Start</strong> button</td>
</tr>
<tr>
<td>Click <strong>Run</strong></td>
<td>Click <strong>Run</strong></td>
</tr>
<tr>
<td>In the <strong>Open:</strong> box, type <strong>dcomcnfg</strong></td>
<td>In the <strong>Open:</strong> box, type <strong>dcomcnfg</strong></td>
</tr>
<tr>
<td>Click <strong>OK</strong></td>
<td>Click <strong>OK</strong></td>
</tr>
</tbody>
</table>
| In the Distributed COM Configuration Properties window, Click on **Agilent PNA Series** in the Applications list. Then click **Properties**... | Open the following folder sequence:  
Component Services Window  
Component Services  
Computers  
My Computer  
DCOM Config |
Right click **Agilent PNA Series**

| Click the **Security** tab | Click the **Security** tab |
| Click **Use custom access permissions** then click **Edit** | Under **Access Permissions**, click **Customize**, then **click Edit** |
| **In Registry Value Permissions**, select **Everyone**, then click **Remove** | Select **Everyone**, then click **Remove** |
| **Click Add** | **Click Add** |
| You could either select one or more of these groups to have access to the PNA, or specific users. To give specific users access, click **Show users** or **Members**, then select the name from the list. | Type a group name or user account name |
| **Click Add**, then **click OK** | **Click OK** |

**Launch Permission**

| Click **Use custom launch permissions**, then click **Edit** | Under **Launch Permissions**, click **Customize**, then **click Edit** |
| **In Registry Value Permissions**, select **Everyone** then click **Remove** | Select **Everyone**, then click **Remove** |
| **Click Add** | **Click Add** |
| You could either select one or more of these groups to have launch permission of the PNA, or specific users. |
|  - To give groups launch permission, select the group from the list. |
|  - To give specific users launch permission, click **Show users** or **Members**, then select the name from the list. | Type a group name or user account name |
| **Click Add**, then **click OK** | **Click OK** |

**Register the PNA Type Library on Your PC**

The type library contains the PNA object model. On your PC, there is a Registry file that keeps track of where object models are located. Therefore, you must register the type library on the PC that will be used to develop code and run the program. It is much more efficient to have the type library registered at design time (BEFORE running your COM program).

Do the following two items before proceeding:
1. Connect your PC and the PNA to LAN.

2. Either map a drive to the analyzer or copy the type library files on a floppy disk or other media. See Drive Mapping.

**Note:** To register the type library on your PC, you must be logged on as an administrator of your PC. Learn about User Accounts.

This procedure will do the following:

- Register the Network Analyzer application on your PC.
- Copy and register the proxystub (835xps.DLL) onto the PC.
- Copy and register the PNA type library (835x.tlb) onto the PC.
- Copy and register the FCA type library (fca.tlb) onto the PC.

1. Using Windows Explorer on your PC, find the Analyzer's C: drive. The drive will not be named "C:" on your PC, but a letter you assigned when mapping the drive.

2. Navigate to Program Files / Agilent / Network Analyzer / Automation

3. Double-click pnaproxy.exe and follow the prompts to Install PNA Proxy. If the installation offers a choice of Modify, Repair, or Remove, then select Remove. Then double-click on pnaproxy.exe again.

4. When prompted, type the Computer name of the PNA (Learn how to find this).

5. After the install program runs, the PNA and FCA type library should be registered on your PC.

6. Your programming environment may require you to set a reference to the PNA type library now located on your PC. In Visual Basic, click Project, References. Then browse to C:/Program Files/Common Files/Agilent/PNA Select 835x.tlb

**Problems?**

- These procedures will fail if there are any programs using the PNA type library (for example: Visual basic, VEE, Visual Studio, or any other application program that may communicate with the PNA).

- Perform the following procedure if the previous procedure did not return an error, but you cannot connect to the PNA.

- If you received an error, check that both the account name and password used on both the PNA and PC match EXACTLY.

- If you still get errors, see http://na.tm.agilent.com/pna/DCOMSecurity.html.

1. Map a drive from your remote PC to the PNA. Note the drive letter your PC assigns to the PNA. Substitute this drive letter for PNA in the following procedure.
2. On your PC, go to a DOS prompt c:>

3. Type **PNA:** (for example o:)

4. Type `cd program files/agilent/network analyzer/automation`

5. Type `copy 835xps.dll c:/program files/common files/agilent/pna`

6. Type `copy 835x.tlb c:/program files/common files/agilent/pna`

   If you will NOT be using **FCA commands**, skip steps 7.,8, and 9.

7. Type `cd..`

8. Type `cd extensions/fca`

9. Type `copy fca.tlb c:/program files/common files/agilent/pna`

10. If it is not already there, copy `reglib.exe` from PNA:/WINNT to your C:/<windows>/system32 directory
    (<windows> is OS-dependent- it is either windows or WINNT)

11. Type `reglib C:/program files/common files/agilent/pna/835x.tlb`

12. Type `regsvr32 C:/program files/common files/agilent/pna/835xps.dll`

13. Type `reglib C:/program files/common files/agilent/pna/fca.tlb`

   Perform the **Access Procedure** after doing these steps.
COM Fundamentals

The following terms are discussed in this topic:

- Objects
- Interfaces
- Collections
- Methods
- Properties
- Events
- Visual Basic Syntax

Note: The information contained in this topic is intended to help an experienced SCPI programmer transition to COM programming. This is NOT a comprehensive tutorial on COM programming.

Other Topics about COM Concepts

Visual Basic Syntax

The examples in PNA Help use Visual Basic as the programming environment for COM, which uses 'dot' notation.

To set a property, follow the object reference with:

- a period (.)
- property or method
- an equal sign (=)
- the new value

For example:

```
object.property = value
```

To read a property, a variable to contain the returned value is followed with:

- an equal sign (=)
- an object, or reference to an object
- a period (.)
- property

For example:
variable = object.property

To execute a method, an object, or reference to an object is followed with:

- a period (.)
- the method
- a blank space
- any required parameters

For example:
object.method parameters

Some methods return values, such as methods that return data. To return data from a method, a variable to contain the returned data is followed with:

- an equal sign (=)
- an object, or reference to an object
- a period (.)
- the method
- any required parameters enclosed in parenthesis

variable = object.method (parameters)

Objects
The objects of the Network Analyzer (Application) are arranged in a hierarchical order. The PNA object model lists the objects and their relationship to one another.

In SCPI programming, you must first select a measurement before making settings. With COM, you first get a handle to the object (or collection) and refer to that object in order to change or read settings (properties).

For more information on working with objects, see Getting a Handle to an Object.

Interfaces
A COM Interface is the connection to an object. When you get a handle to an object, you are actually using an interface to an object. This is important if you are developing PNA code that will run on multiple code versions. For more information, see PNA Interfaces.

Collections
A collection is an object that contains several other objects of the same type. For example, the Channels collection contains all of the channel objects.

Note: In the following examples, the collections are referred to as a variable. Before using a collection object, you must first get an instance of that object. For more information, see Getting a Handle to an Object

Generally, items in a collection can be identified by number or by name. The order for objects in a collection cannot be assumed. They are always unordered and begin with 1. For example, in the following procedure, chans(1) is used to set averaging on the first channel in the Channels collection (not necessarily channel 1).
Sub SetAveraging()
    chans(1).AveragingFactor = 10
End Sub

The following procedure uses the measurement string name to set the display format for a measurement in the measurements collection.

meass("CH1_S11_1").Format = 1

You can also manipulate an entire collection of objects if the objects share common methods. For example, the following procedure sets the dwell time on all of the segments in the collection.

Sub setDwell()
    For Each seg In segs
        segs.DwellTime = 0.03
    Next
End Sub

Methods
A method is an action that is performed on an object. For example, CreateSParameter is a method on the Application object. The following procedure uses that method to create a new S21 measurement in channel 1 in a new window.

Sub CreateMeas
    app.CreateSParameter 1,2,1,1
End Sub

Properties
A property is an attribute of an object that defines one of the object's characteristics, such as size, color, or screen location. A property can also change an aspect of the object's behavior, such as whether the object is visible. In either case, to change the characteristics of an object, you change the values of its properties.

For example, the following statement sets the IF Bandwidth of a channel to 1 KHz.

Chan.IFBandwidth = 1e3

You can also read the current value of a property. The following statement reads the current IF Bandwidth of a channel into the variable Ifbw.

Ifbw = Chan.IFBandwidth

Some properties cannot be set and some cannot be read. The Help topic for each property indicates if you can:

- Set and read the property (Write/Read)
- Only read the property (Read-only)
- Only set the property (Write-only)

Events
A COM event is an action recognized by an object, such as clicking the mouse or pressing a key. Using events, your program can respond to a user action, program code, or triggered by the analyzer.

The SCPI equivalent of an event is a Service Request (SRQ).

For example:

OnChannelEvent
For more information, see Working with the Analyzer's Events.
Getting a Handle to an Object

The following are discussed in this topic:

- **What Is a Handle**
- **Declaring an Object Variable**
- **Assigning an Object Variable**
- **Navigating the Object Hierarchy**
- **Getting a Handle to a Collection**

### Other Topics about COM Concepts

### What Is a Handle

In SCPI programming, you must first select a measurement before changing or reading settings. With COM, you first get a handle to the object (or collection) and refer to that object to change or read its settings. The following analogy illustrates this:

A CAR could be called an object. More precisely, CAR is a class of objects. For example, one of the properties of the CAR class is "**Color**". You can read (by looking) or set (by painting) the color property of a car object. In other words, you can only read or set properties of a specific car object; not the entire car class. Therefore, to read or set a property, you need to get "a handle", or an instance of the object.

This process is also called "accessing an object", "getting an instance of an object", "returning an object", or "referring to an object". You can have handles to many instances of an object at the same time.

### Accessing PNA Objects

The PNA Application object is the highest object in the PNA object model hierarchy. Because of that, it is the only object that must be 'created' before it, or any other objects, can be accessed and used. During the creation process, the application object assigned to a variable name, or handle. Throughout your program, that object is used by referring to that variable. All PNA objects can be assigned to a variable, and subsequently referred to, in this same manner.

The following example shows how to create the PNA Application object, as well as illustrate the general steps of get a handle to an object.

There are two steps in the process of getting a handle to analyzer objects:

1. Declaring a Variable
2. Assigning an Object to the Variable

### Declaring a Variable

**Note:** The examples in these topics use the Visual Basic Programming Language. See the short section regarding Visual Basic syntax.

Use the Dim statement or one of the other declaration statements (Public, Private, or Static) to declare a variable.
The type of variable that refers to an object must be a Variant, an Object, or a specific type of object. Some programming languages, such as VBScript and Agilent VEE, do not allow you to specify variable types. The following examples ALL declare the variable `pna`. Each subsequent statement is more specific than the previous:

- `Dim pna 'Variant data type.`
- `Dim pna As Object 'Object data type.`
- `Dim pna As AgilentPNA835x.Application 'Specific Application type`
- `Dim pna As AgilentPNA835x.IApplication 'Interface type`

1. If you use a variable without declaring it first, the data type of the variable is Variant. If you don't care about using automatic type checking, and willing to run code less efficiently, this method is very safe and is useable on all programming environments.

2. If you know the specific object type, and your programming environment allows it, you can declare the variable as an object.

3. Declaring a specific object type provides automatic type checking (Intellisense), faster code, and improved readability.

4. Declaring the interface is the most specific way and is beneficial when developing code for multiple firmware revisions. Learn more about Interfaces.

### 2. Assigning an Object to a Variable

To assign an object instance to a variable, use the `Set` keyword before the object variable that was declared previously. In the following line of code, we SET the current AgilentPNA835x Application to “pna”.

```
Set pna = AgilentPNA835x.Application
```

As mentioned earlier, the AgilentPNA835x object is unique because it is the highest level of object in the PNA object model hierarchy. Therefore, we must use the `CreateObject` keyword with the `(classname,server name)` parameters.

- The `classname` for the analyzer object is always "AgilentPNA835x.Application".
- To find your analyzer's `server name`, see View or change full computer name

The following statements create an instance of the Analyzer object.

```
Dim pna AS AgilentPNA835x.Application
Set pna = CreateObject("AgilentPNA835x.Application", "Analyzer46")
```

**Note:** These statements will start the PNA application if it is not already running on your instrument.

### Navigating the Object Hierarchy

Once an instance of the PNA Application is "created", you access all of the PNA objects by navigating the object hierarchy. Navigating the object model hierarchy can be tricky. In addition, you also need to know how to refer to a specific instance of that object. For example, if you have three measurements present on the PNA, how do you refer to the channel 1 measurement? Each object on the PNA Object Model image is linked to an object page. At
the top of each object page is a Description section and another called "Accessing the ... Object". These sections together explain how to navigate the PNA hierarchy to access a specific instance of that object.

From the previous discussion, you may think that you must always declare and assign variables to an object before setting or reading its properties. While this method is best for objects that you will continue to reuse, such as a measurement, it is not always necessary. You can also refer to an object directly.

The TriggerSetup object, which is a child of the Application object. Because we will only need to refer to this object once to set a couple of properties, and it is easy to access, we will refer to it directly. From the previous example, we already have a handle to the Application object in the variable pna. The following example uses Visual basic 'dot' notation to refer to the TriggerSetup object, and then the Scope property.

```
Dim trig As Object
Set trig = pna.TriggerSetup
```

By referring to the TriggerSetup object directly, we must type the same path whenever we refer to properties on the TriggerSetup object. The following method assigns the pna.TriggerSetup object to a variable that can be reused.

```
trig.Scope = naChannelTrigger
trig.Source = naTriggerSourceInternal
```

Getting a Handle to a Collection

The analyzer has several collections of objects which provide a convenient way of setting or reading all of the objects in the collection with a single procedure. Also, there are objects (limit lines for example) that can only be accessed through the collection.

To get a handle to an item in a collection, you can refer to the object by item number or sometimes by name. However, you first have to get a handle to the collection. To assign the collection to a variable, use the same two step process (1. declare the variable, 2. assign the variable using ‘Set’).

```
Dim meass As Measurements
Dim meas As Measurement
```

You can then iterate through the entire collection of measurements to read or set properties.

```
Sub setFormat()
    For Each meas In meass
        meas.Format = naDataFormat_LinMag
    Next
End Sub
```

Or you can read or set a property on an individual object in the collection:

```
meass(1).Format = naLinMag
```

Note: Each object and collection has its own unique way of dealing with item names, and numbers. Refer to the Analyzer Object Model for details.
Collections in the Analyzer

Collections are a gathering of similar objects. They are a convenience item used primarily to iterate through the like objects in order to change their settings. Collections generally provide the following generic methods and properties:

<table>
<thead>
<tr>
<th>Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item(n)</td>
</tr>
<tr>
<td>Count</td>
</tr>
<tr>
<td>Add(n)</td>
</tr>
<tr>
<td>Remove(n)</td>
</tr>
</tbody>
</table>

where (n) represents the number of the item in the collection. Some collections may have unique capabilities pertinent to the objects they collect.

Other Topics about COM Concepts

Collections are Dynamic

A collection does not exist until you ask for it. When you request a Channels object (see Getting a Handle to an Object / Collection), handles to each of the channel objects are gathered and placed in an array.

For example, if channels 2 and 4 are the only channels that exist, then the array will contain only 2 items. The command 'channels.Count' will return the number 2, and:

- Channels(1) will contain the channel 2 object.
- Channels(2) will contain the channel 4 object.

The ordering of objects within the collection should not be assumed. If you add a channel to the previous example, as in:

```pna.Channels.Add(3)```

'channels.Count' will now return 3 and:

- Channels(1) will contain the channel 2 object.
- Channels(2) will contain the channel 3 object.
- Channels(3) will contain the channel 4 object.

Primarily, collections are useful for making this type of iteration possible:

```Dim ch as Channel
For each ch in pna.Channels
    Print ch.Number
    Print ch.StartFrequency
    Print ch.StopFrequency
Next ch```

As soon as this for-each block has been executed, the Channels object goes out of scope.
**COM Data Types**

The PNA uses several data types to communicate with the host computer. Before using a variable, it is best to declare the variable as the type of data it will store. It saves memory and is usually faster to access. The following are the most common data types:

- **Long Integer**
- **Single Precision (Real)**
- **Double Precision (Real)**
- **Boolean**
- **String**
- **Object**
- **Enumeration**
- **Variant**

**Other Topics about COM Concepts**

**Long** (long integer) variables are stored as signed 32-bit (4-byte) numbers ranging in value from -2,147,483,648 to 2,147,483,647.

**Double** (double-precision floating-point) variables are stored as IEEE 64-bit (8-byte) floating-point numbers ranging in value from \(-1.79769313486232E308\) to \(-4.94065645841247E-324\) for negative values and from \(4.94065645841247E-324\) to \(1.79769313486232E308\) for positive values.

**Single** (single-precision floating-point) variables are stored as IEEE 32-bit (4-byte) floating-point numbers, ranging in value from \(-3.402823E38\) to \(-1.401298E-45\) for negative values and from \(1.401298E-45\) to \(3.402823E38\) for positive values.

**Boolean** variables are stored as 16-bit (2-byte) numbers, but they can only be True or False. Use the keywords True and False to assign one of the two states to Boolean variables.

When other numeric types are converted to Boolean values, 0 becomes False and all other values become True. When Boolean values are converted to other data types, False becomes 0 and True becomes -1.

In PNA release 5.26, the following properties were changed to return True rather than 1 to conform with this definition. This change may affect the functionality of your COM program:

- **Bandwidth Tracking Property**
- **ErrorCorrection Property**
- **IFGateEnable Property**
String variables hold character information. A String variable can contain approximately 65,535 bytes (64K), is either fixed-length or variable-length, and contains one character per byte. Fixed-length strings are declared to be a specific length. Variable-length strings can be any length up to 64K, less a small amount of storage overhead.

Object variables are stored as 32-bit (4-byte) addresses that refer to objects within the analyzer or within some other application. A variable declared as Object is one that can subsequently be assigned (using the Set statement) to refer to any actual analyzer object.

Enumerations (Enum) are a set of named constant values. They allow the programmer to refer to a constant value by name instead of by number. For example:

```plaintext
Enum DaysOfWeek
    Sunday = 0
    Monday = 1
    Tuesday = 2
    Wednesday = 3
    Thursday = 4
    Friday = 5
    Saturday = 6
End Enum
```

Given this set of enumerations, the programmer can then pass a constant value as follows:

```plaintext
SetTheDay(Monday)
```
rather than

```plaintext
SetTheDay(1)
```
where the reader of the code has no idea what the value 1 refers to.

However, the analyzer RETURNS a long integer, not the text.

```plaintext
Day = DaysofWeek(today) 'Day = 1
```

Variant - If you don't declare a data type ("typed" data) the variable is given the Variant data type. The Variant data type is like a chameleon - it can represent many different data types in different situations.

The PNA provides and receives Variant data because there are programming languages that cannot send or receive "typed" data. Variant data transfers at a slower rate than "typed" data.
PNA Interfaces

A COM interface is the connection to an object. When you get a handle to an object, you are actually using an interface to an object. This subtle distinction is relevant to the COM programmer for the following two reasons:

- **Interface Inheritance (Coding for Multiple PNA Versions)**
- **Custom Interfaces**

### Other Topics about COM Concepts

#### Interface Inheritance (Coding for Multiple PNA Versions)

The PNA continues to evolve and release new firmware/software versions that provide more functionality and features. New commands are added to existing objects, and with them new interfaces are added to support those commands. For example, new commands were added to the Measurement object in PNA release 3.0. These commands are accessible from the new IMeasurement2 interface. This can be important if you develop code using the type library in release 3.0, and run the code on a PNA with an older release, such as 2.0.

When you use a command that was new with release 3.0, and you run that code on a PNA with release 2.0 firmware, errors will occur because that PNA does not recognize the new commands. However, even if you do NOT utilize new commands, errors can still occur. The following example shows how this occurs and how to avoid it.

The following Visual Basic statement dimensions the `meas` variable as an object.

```vbnet
Dim meas As Measurement
```

When the program compiles, Visual Basic figures out what interface to use to access that object. When dimensioning as an object, VB will use the default interface. As new interfaces are added to an object, they become the default interface. If this program was developed and compiled using the PNA 3.0 type library, the default Interface of the Measurement Object was IMeasurement2. However, if this program is run on an instrument with PNA 2.0 firmware, there was no IMeasurement2 Interface, and an E_NOINTERFACE error will occur.

Therefore, the more robust approach would be to specify the interface instead of the object when declaring a variable.

```vbnet
Dim meas As IMeasurement
```

This code will ONLY use the IMeasurement interface; not the default interface.

However, regardless of how you declare a variable, errors will always occur if you use new commands, and run the code on an older instrument.

#### Custom Interfaces

The PNA object model contains three "custom" interfaces use "typed" variables, which is more efficient than using variant type variables. However, these interfaces are only usable from VB6, C, & C++. All other programming languages must use the other standard interfaces.

The custom interfaces are:

- **IArrayTransfer** - Measurement object
- **ICalData** - Calibrator object
- **ISourcePowerCalData** - Channel object
Working with Events

- What are Events?
- Using the Analyzer's Events
- Event ID's
- Filtering Events
- List of Events
- Out of Range Errors
- Troubleshooting Problems with Events

See Also

- Events Example
- Errors and the SCPIStringParser Object

### Other Topics about COM Concepts

### What are Events?

Windows applications work from user-initiated events such as mouse moves and mouse clicks. A mouse-click produces an event that the programmer can either ignore or "handle" by providing an appropriate subroutine like this:

```vbscript
Sub DoThis_onClick
    Perform something
End Sub
```

If this subroutine were in your program and the mouse-click event occurs on your PC, it would generate a "Callback" to the client and interrupt whatever it was doing and handle the event.

A more practical example of an event in the analyzer is Limit test. If limit test is on and the measurement fails, the analyzer produces a "Limit-failed" event. If the measurement passed, the analyzer produces a "Limit-succeeded" event.

The Analyzer has a very sophisticated Event structure. Your program **CAN** be notified when one or more events occur. However, it may not be necessary.

For example, the analyzer has an event that will notify your program when a sweep is complete. A simpler alternative is to use a synchronous command which waits for the sweep to complete.

```vbscript
sync = True
app.ManualTrigger sync
chan.StartFrequency = 4.5E6
```

This would **NOT** work if you want the controller to do other things while waiting, like setup a power meter or sort some data. In this case you would like a "callback" from the analyzer to let your program know that the sweep has completed. For an example of this see [Events Example](#).
Another reason to use events is when you want to be notified of several conditions when they occur, such as errors or source unlock conditions. It would not be practical to routinely poll these conditions while executing your program.

**Using Events**

If you decide to use the COM events to get a callback, your program must do two things:

1. **Subscribe to events:**

   All events in the analyzer are a child of the Application object through the INetworkAnalyzerEvents Interface. You must tell the Application object that you are interested in receiving event callbacks. This process is called subscription.

   In Visual Basic, this is done by including " WithEvents " in the declaration statement. The declaration below dimensions an Application object (myPNA) and subscribes to the events produced by the Application.

   ```vba
   Dim WithEvents myPNA as AgilentPNA835x.Application
   ```

   In C++, this is a bit more involved. You must queryInterface for the IconnectionPointContainer interface, locate the INetworkAnalyzerEvents interface via a call to FindConnectionPoint and call Advise().

2. **Implement the Event Handler**

   When an event occurs, the Application object will "callback" to the client through the INetworkAnalyzerEvents interface.

   In VB, click on the object window (upper left pane). Find the Application object and click it. The event interfaces will appear in the upper right pane. As you click on them, VB supplies the first line of code. You fill in the rest of the handler routine to service the event. The following is an example of a event handler subroutine.

   **Note:** In C++, you must type the callback.

   ```vba
   Private Sub OnChannelEvent( eventID as Variant, channelNumber as Variant)
   Select Case (eventID)
   Case naEventID_CHANNEL_TRIGGER_COMPLETE:
     GetData( channelNumber )
   Case naEventID_CHANNEL_TRIGGER_ABORTED:
     MsgBox( "Hey don't touch the front panel!")
   End Select
   End Sub
   ```

   When the trigger is complete, the application object "fires" the event by making a callback to the event handler Sub OnChannelEvent() .

**Event IDs**

<table>
<thead>
<tr>
<th>Sev</th>
<th>C</th>
<th>R</th>
<th>Facility</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>3</td>
<td>2</td>
<td>2 2 2 2 2 2 2 2 1 1 1 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
<td></td>
</tr>
<tr>
<td>1  0</td>
<td>9</td>
<td>8 7 6 5 4 3 2 1 0 9 8 7 6 5 4 3 2 1 0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Filtering Events**

There are over 140 different events that you subscribe to when you “Dim WithEvents...” (or the equivalent in your programming language). Monitoring all of these conditions slows the speed of the analyzer significantly. The following methods allow you to filter the events so that you only monitor specific conditions.
- **AllowEventMessage** - monitor a specific event
- **AllowAllEvents** - monitor ALL events
- **DisallowAllEvents** - monitor NO events
- **AllowEventCategory** - monitor specific event categories (discussed later)
- **AllowEventSeverity** - monitor events having one or more of the following severity levels associated with them.

### Severity Enumeration

<table>
<thead>
<tr>
<th>Code</th>
<th>Enumeration</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>naEventSeveritySUCCESS</td>
</tr>
<tr>
<td>01</td>
<td>naEventSeverityINFORMATIONAL</td>
</tr>
<tr>
<td>10</td>
<td>naEventSeverityWARNING</td>
</tr>
<tr>
<td>11</td>
<td>naEventSeverityERROR</td>
</tr>
</tbody>
</table>

**Code** | **Severity Enumeration**
---|---
00 | naEventSeveritySUCCESS - the operation completed successfully
01 | naEventSeverityINFORMATIONAL - events that occur without impact on the measurement integrity
10 | naEventSeverityWARNING - events that occur with potential impact on measurement integrity
11 | naEventSeverityERROR - events that occur with serious impact on measurement integrity

### List of Events

The following is a list of categories and the general types of events they include. Click the link view the event details.

<table>
<thead>
<tr>
<th>Category Enumerations</th>
<th>Callback</th>
</tr>
</thead>
<tbody>
<tr>
<td>naEventCategory_PARSER</td>
<td>OnSCPIEvent</td>
</tr>
<tr>
<td>naEventCategory_MEASURE</td>
<td>OnMeasurementEvent</td>
</tr>
<tr>
<td>naEventCategory_CHANNEL</td>
<td>OnChannelEvent</td>
</tr>
<tr>
<td>naEventCategory_HW</td>
<td>OnHardwareEvent</td>
</tr>
<tr>
<td>naEventCategory_CAL</td>
<td>OnCalEvent</td>
</tr>
<tr>
<td>naEventCategory_USER</td>
<td>OnUserEvent</td>
</tr>
<tr>
<td>naEventCategory_DISPLAY</td>
<td>OnDisplayEvent</td>
</tr>
<tr>
<td>naEventCategory_GENERAL</td>
<td>OnSystemEvent</td>
</tr>
</tbody>
</table>

**Note:** Use the [MessageText](#) Method to get a text message describing the event.

### Out of Range Errors

When you attempt to set a value on an active function that is beyond the range (min or max) of the allowable values, the analyzer limits that value to an appropriate value (min or max) and sets the function to the limited value. From the front panel controls this is visually evident by the limited value in the edit box or by the annotation on the display. An example would be attempting to set the start frequency below 300kHz. The edit control doesn't allow the number to fall below 300kHz.

When the automation user programs a setting (such as start frequency below the allowable limits) the same behavior takes place. The analyzer accepts the limited value. However, in order to learn what setting took place, you have to read the HRESULT.
All automation calls return HRESULTs. By default the HRESULT returned when an overlimit occurs is S_NA_LIMIT_OUTOF RANGE. This value is a success code, meaning that bit 31 in this 32 value is 0. Programmers should check the return code from all automation calls to determine success or failure.

Some C++ macros (like SUCCEEDED(hr) or FAILED(hr) ) only check bit 31. So if you are interested in trapping this outOfRange error you will have to check for S_NA_LIMIT_OUTOF RANGE explicitly.

Alternatively, you can configure the analyzer to report outOfRange conditions with an error code. Use the method: App.SetFailOnOverRange (true). With this method set TRUE, any overrange error will return E_NA_LIMIT_OUTOF RANGE_ERROR.

This method is provided for the benefit of VB clients. VB users can't detect specific success codes because the VB runtime strips off the HRESULT and only raises a run time error if bit 31 is set, indicating a fail code.

**Troubleshooting Problems with Callbacks**

When you do callbacks, the client PC becomes the server and the analyzer (server) becomes the client. Callbacks can only take place when both server and client are in the same workgroup or in the same domain. See Configure for COM.
Read and Write Calibration Data using COM

Calibration data in the PNA is stored in Cal Sets. Learn more about Cal Sets.

You can read or write two types of Calibration data:

- **Error Terms** - calculated data using standard measurement data and the algorithms for the specified cal type.

- **Standard Measurement data** - raw data resulting from the measurement of a calibration standard.

Each of these data are available in the PNA in either variant data or typed data. Learn more about variant and typed data.

---

Other Topics about COM Concepts

Calibration / Cal Set Interfaces

There are several interfaces associated with Calibration.

**ICalibrator**

This interface is the original interface provided with the first version of the PNA. It provides remote access to the "Unguided" Calibration wizard. This interface can perform 1 and 2 port calibrations as well as response cals.

This interface can also read and write error terms from/to a Cal Set. However, ICalibrator is NOT recommended for this purpose. The ICalSet2 Interface is better suited for reading and writing error terms.

See a vbscript example of how to perform a 2-port Cal and read the cal data.

**IGuidedCalibration**

This interface provides the methods and properties used by the Guided Calibration wizard. With this interface you can perform multi-port calibrations (1 to 4 port cals), but no response cals.

**ICalSet2 and ICalData3**

These interfaces provide access to the Cal Set contents. You can read and write error terms with both of these interfaces.

- ICalSet2 uses Variant data, which means it is usable from vbscript.

- ICalData3 uses "typed" data, which means it can be used from any automation engine that can read the type library (VEE, VB, C++, etc.). Typed arguments (such as float or single) are more efficient than variants, so use the ICalData3 interface where better performance is needed.

See a vbscript example of how to read Cal Set data.

**ICalSet3**

This interface provides access to the stimulus attributes of the Cal data: frequency, power, number of points. These are the stimulus conditions under which the Cal Set was created.
Programming the PNA with C++

The programming information contained in this Help system is aimed at the Visual Basic programmer. VB does a lot of work for the programmer when it comes to managing and accessing components. Using a lower level language like C++ requires a more thorough understanding of the underlying tenets of COM. It is not the intent of this section to teach COM programming. The following is intended to acquaint you with some of the basic concepts you need to know in order to program against COM.

- Initializing COM
- Importing the Type Library
- Creating the Application Object
- Errors
- Events
- Additional Reading
- Example

**Note:** The information in this section assumes development on a Windows OS using Microsoft tools.

**Other Topics about COM Concepts**

Initializing COM

The first thing you must do before performing any COM transactions is to initialize the COM library. You can do this in a number of ways. The most basic of these is a call to `CoInitialize( )` or `CoInitializeEx( )`. Alternatively you can use the MFC (Microsoft Foundation Classes) `AfxOleInit( )`.

Conversely, before your program exits you must uninitialize COM. You can accomplish this with `CoUninitialize( )` or the MFC routine `AfxOleTerm( )`.

Importing the Type Library

To make a component available to the client, the server exports what is called the type library. For the PNA, this file is 835x.tlb. It is located on the PNA's hard drive at `C:/Program Files/ Agilent/ Network Analyzer/ Automation`. See [Configure for COM-DCOM Programming](#).

The type library can be read and deciphered using another COM interface called ITypeLib. VB uses this interface to present, for example, its object browser. Visual C++ can also read type libraries. This is done by importing the type library into your project with a compiler directive:

```cpp
#import "C:/Program Files/Common Files/Agilent/Pna/835x.tlb", named_guids
```

When you compile your program with this statement in it, the compiler creates two other files: `835x.tlh` and `835x.tli`. The first is a header file that contains the type definitions for the PNA's COM interfaces and their methods. The second file contains inline functions that wrap the PNA's interface methods. The wrappers are beneficial in that they contain error reporting for each of the method calls.

The .tlh file defines a smart pointer which you can use to access the PNA's objects. The smart pointer definition
A smart pointer is a term used for a C++ object that encapsulates a pointer used to refer to a COM object. All COM objects derive from the interface IUnknown. This interface has three methods: QueryInterface(), AddRef(), and Release(). The function of the AddRef and Release methods is to maintain a reference count on the object and thus control the object's lifetime. Anytime you copy or create a reference to a COM object, you are responsible for incrementing its reference count. And likewise, when you are finished using that reference, it is your responsibility to Release it. Smart pointers do this work for you, as shown in the example program. In addition, smart pointers will also perform the QueryInterface call when required. QueryInterface is a method that requests a specific interface from an object. In the example program we gain access to the IArrayTransfer interface of the Measurement object. In the ReadMethod routine, we see this:

PTransferData = pMeas;

The assignment operator is overloaded for the smart pointer and in reality, this simple statement does this:

HRESULT hr = pMeas->QueryInterface( IID_IArrayTransfer, (void**)&pTransferData);

Using the existing interface pointer (pMeas) to the object, this call asks the object if it supports the IArrayTransfer interface, and if so to return a pointer to it in pTransferData. Smart pointer makes life easier for the C++ programmer. Read more about smart pointers in Microsoft Developer's Network Library (MSDN).

Creating the Application Object

The only createable object exported by the PNA is the Application object. Typically this would be done with a call to CoCreateInstance:

STDAP IC oCreateInstance(
    CLSID__IApplication, //Class identifier (CLSID) of the object
    NULL, //Pointer to controlling IUnknown
    CLSCTX_SERVER, //Context for running executable code
    IID_IApplication, //Reference to the IID of the interface
    (void**)&pNA //Address of output variable that receives
    //the interface pointer requested in riid
);

With the smart pointer, this is taken care of with the following call:

IApplicationPtr pNA; // declare the smart pointer
pNA = IApplicationPtr("AgilentPNA835x.Application.1");

Errors

All COM method calls are required to return an HRESULT. This is 32 bit long with a specific format.

- The most significant bit indicates success(0) or failure(1).
- The lower 16 bits indicate the specific failure.

Visual Basic strips off the returned HRESULT and raises an error object for non-successful returns. The C++ programmer must himself be diligent about handling errors. You must check the return value of each COM call to ensure its success.

Events
The Application object sources the INetworkAnalyzerEvents interface. This object is the source for all events. To use events in C++, you must do two things:

1. Implement the INetworkAnalyzerEvents interface - derive an object from INetworkAnalyzerEvents and implement the methods described there.
2. Subscribe to the IConnectionPoint interface of the Application object. - obtain a pointer to the IConnectionPointContainer interface of the Application object and making the following request:

   ```
   FindConnectionPoint( IID_InetworkAnalyzerEvents, &pConnection );
   ```

   A successful call to this interface will return a valid pointer in pConnection. Use this pointer to subscribe to the Application object:

   ```
   pConnect->Advise( IUnknown* punk, DWORD dwCookie);
   ```

   This call provides the server object with a callback address. The IUnknown pointer in this call is the IUnknown pointer of the object that implements the INetworkAnalyzerEvents interface. This is the event sink. The application object needs a pointer to this object in order to call your interface when an event occurs. The `dwCookie` is your subscription key. Use it to unsubscribe (see Unadvise( )).

**Additional Reading**

"MSDN" - Microsoft Developer's Network Library

"Learning DCOM", by Thuan L. Thai, published by O'Reilly(1999)


"Understanding ActiveX and OLE", by David Chappell, also published by Microsoft Press (1996)


**Example**

The example uses the smart pointer created by Microsoft Visual Studio. The calls to CoInitialize and CoUninitialize open and close the COM libraries. In the example, notice that the pointers local to the main routine are explicitly released. When smart pointers go out of scope, they will perform this duty implicitly. However, we are calling CoUninitialize before they have the chance to be destroyed, so we are obliged to release them.

See the example program.
Using COM from .NET

To communicate with the PNA from Microsoft .NET enabled languages such as C# and Visual Basic.NET perform the following steps:

1. Configure your PC and PNA for COM-DCOM Programming.

2. Reference the type library within the development environment (see the following exception for managed C++ projects.) In the process of referencing the type library, a .NET assembly is created that wraps the PNA type library with a .NET friendly interface. This .NET assembly is called an Interop Assembly.

Note: ONLY 32-bit compiler option is supported (64-bit is NOT supported).

Exception for managed C++ projects: To generate the Interop Assembly for managed C++ projects, you must use the tlbimp.exe utility. This utility is described in the MSDN documentation. On your PC, click Start then Run then type: tlbimp.exe 835x.tlb and click OK. After doing this you can use the #using directive to include the Interop Assembly on managed C++ projects.

Example: Creating a .NET object from C#

The following is an example that shows how to create a .NET object that connects to the PNA over DCOM. In this example, machineName is either the DNS name or the IP address of the PNA to connect with.

```csharp
Type pna = Type.GetTypeFromProgID("AgilentPNA835x.Application", machineName);
AgilentPNA835x.IApplication app = (AgilentPNA835x.IApplication)Activator.CreateInstance(pna);
```

See C# Example Programs:

Perform a Guided Cal with C#

Using C#

Registering the PNA Primary Interop Assembly (PIA) (OPTIONAL)

The PIA is NOT necessary to communicate with the PNA. The following procedure is useful only when there are two .NET programs that want to share the same PNA interface definitions. Without the PIA, each .NET application would use its own Interop Assembly.

To register the PIA on a machine, you need to have the common language runtime (CLR) installed. This is included with Visual Studio.NET. Then perform the following steps:

Note: In the following steps, replace <local directory> with the full path name of the specified file on your PC.

1. Run the PNAProxy.exe program as described in Configure for COM-DCOM Programming.

2. On the PNA, copy C:/Program Files/Agilent/Network Analyzer/Automation/AgilentPNA835x.dll to a local directory on your PC. Make a note of this directory.

3. On your PC, click Start, then Run, then type: regasm <local directory>/AgilentPNA835x.dll and click OK to register the dll.

4. Again, click Start, then Run, then type: gacutil /i <local directory>/AgilentPNA835x.dll and click OK to add the assembly to the Global Assembly Cache (GAC).
To Uninstall the PIA, perform the following:

1. On your PC, click Start, then Run, then type: `gacutil /u <local directory> /AgilentPNA835x.exe` and click OK to remove the assembly from the GAC.

2. On your PC, click Start, then Run, then type: `regasm /unregister <local directory> /agilentpna835x.dll` and click OK to unregister the assembly.

3. To uninstall PNA Proxy.exe use the Add/Remove Programs utility in the control panel.

Last Modified:

2-Feb-2009   Updated by JE
SCPI Command Tree

See Also

- Example Programs
- Find commands using a simulated PNA-X UI
- See list of all SCPI Errors.
- See Calibrating the PNA Using SCPI
- Synchronizing the PNA and Controller
- IEEE- 488.2 Common Commands
- Local Lockout

ABORT  Stops all sweeps

+ CALCulate  Click to hide and show CALC branches

:CORRection  Electrical Delay and Phase Offset
:CUStom  Custom measurements
:DATA  Sends and queries data.
:EUQation  Equation Editor
:FIILTer  Time domain gating
:FORMat  Display format
:FSIMulator  Balanced measurements and Fixturing
:FUNCtion  Trace Statistics
:GCDATA  Read Gain compression data
:GCMears  Gain Compression Analysis
:GDElay  Group Delay Aperture setting
:LIMit  Limit lines for pass / fail testing
:MARKer  Marker settings
:MATH  Math / Memory
:MIXer  X-axis display for FCA measurements
:NORMalize  Receiver power cal (Obsolete)
:OFFSet  Mag and Phase offset
:PARameter  Create and delete measurements
:RDATa?  Queries receiver data
:SMOothing  Point-to-point smoothing
:TRANsform  Time domain transform
:X:VAUes  Returns X-Axis values for trace
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONTrol</td>
<td>Interface control, ECAl module state control, and Rear-panel connector control.</td>
</tr>
<tr>
<td>CONTrol:CALPod</td>
<td>Configure and control Calpod devices</td>
</tr>
<tr>
<td>CSET:FIXTure</td>
<td>Create a Cal Set that includes the effects of a fixture.</td>
</tr>
<tr>
<td>DISPLAY</td>
<td>Display settings</td>
</tr>
<tr>
<td>FORMat</td>
<td>Format for data transfer</td>
</tr>
<tr>
<td>HCOPY</td>
<td>Hardcopy printing</td>
</tr>
<tr>
<td>INITiate</td>
<td>Continuous or manual triggering</td>
</tr>
<tr>
<td>LXI</td>
<td>LXI communications</td>
</tr>
<tr>
<td>MMEMory</td>
<td>Saves and recalls instrument states</td>
</tr>
<tr>
<td>OUTPUT</td>
<td>Turns RF power ON and OFF</td>
</tr>
<tr>
<td>ROUTe</td>
<td>Controls internal switch to reference receiver. (Opt 81)</td>
</tr>
</tbody>
</table>

**SENSe**  
Click to hide and show SENSe branches

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>:AVErage</td>
<td>Sweep Averaging</td>
</tr>
<tr>
<td>:BANDwidth</td>
<td>IF Bandwidth</td>
</tr>
<tr>
<td>:CLASSs</td>
<td>Returns measurement class name</td>
</tr>
<tr>
<td>:CORRection</td>
<td>Calibration and other correction settings</td>
</tr>
<tr>
<td>:COUPle</td>
<td>Chopped or Alternate sweep</td>
</tr>
<tr>
<td>:FOM</td>
<td>Frequency Offset (opt 080)</td>
</tr>
<tr>
<td>:FREQuency</td>
<td>Frequency sweep settings</td>
</tr>
<tr>
<td>:GCSetup</td>
<td>Gain Compression App (opt 086)</td>
</tr>
<tr>
<td>:IF (E836xC)</td>
<td>IF Access settings (opt H11)</td>
</tr>
<tr>
<td>:IF (PNA-X)</td>
<td>IF Access settings</td>
</tr>
<tr>
<td>:IMD (PNA-X)</td>
<td>Intermodulation Distortion (opt 087)</td>
</tr>
<tr>
<td>:IMS (PNA-X)</td>
<td>Intermodulation Spectrum (opt 087)</td>
</tr>
<tr>
<td>:MIXer</td>
<td>FCA measurements (opts 082 and 083)</td>
</tr>
<tr>
<td>:MULTiplexer</td>
<td>Controls external test sets.</td>
</tr>
<tr>
<td>:NOISe (PNA-X)</td>
<td>Noise Figure (opt 029)</td>
</tr>
<tr>
<td>:PATH</td>
<td>Provides access to hardware configuration</td>
</tr>
<tr>
<td>:POWer</td>
<td>Receiver attenuation and overpower protection</td>
</tr>
<tr>
<td>:PULSe (PNA-X)</td>
<td>Configure internal pulse generators</td>
</tr>
<tr>
<td>:ROLE</td>
<td>Assign sources to roles.</td>
</tr>
<tr>
<td>:ROSCillator</td>
<td>Returns the source of the reference oscillator.</td>
</tr>
<tr>
<td>:SEGMent</td>
<td>Segment sweep settings.</td>
</tr>
<tr>
<td>:SWEep</td>
<td>Sweep types</td>
</tr>
<tr>
<td>:X:VALues</td>
<td>Returns X-axis values for channel</td>
</tr>
<tr>
<td>Command</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td>SOURce</td>
<td>Source power to the DUT</td>
</tr>
<tr>
<td>SOURce:PHASE</td>
<td>Phase control (Opt 088)</td>
</tr>
<tr>
<td>SOURce:POWER:CORR</td>
<td>Source power Calibration</td>
</tr>
<tr>
<td>STATus</td>
<td>Reads the PNA status registers</td>
</tr>
<tr>
<td>SYSTem</td>
<td>Misc PNA capabilities</td>
</tr>
<tr>
<td>SYSTem:CONF:EDEV</td>
<td>Configure external devices</td>
</tr>
<tr>
<td>TRIGger</td>
<td>Trigger measurements</td>
</tr>
</tbody>
</table>

Last Modified:

- 5-Jan-2011  Added Phase control
- 13-Apr-2010 Added CSET (A.09.20)
- 23-Feb-2010 Added CALC:GDElay and LXI (A.09.20)
- 30-Jul-2009 Added syst:conf:evev and Calc:X?
- 16-Jan-2009 Moved class
IEEE 488.2 Common Commands

*CLS - Clear Status
Clears the instrument status byte by emptying the error queue and clearing all event registers. Also cancels any preceding *OPC command or query. See Status Commands and Reading the Analyzer's Status Registers.

*ESE - Event Status Enable
Sets bits in the standard event status enable register. See Status Commands and Reading the Analyzer's Status Registers.

*ESE? - Event Status Enable Query
Returns the results of the standard event enable register. The register is cleared after reading it. See Status Commands and Reading the Analyzer's Status Registers.

*ESR - Event Status Enable Register
Reads and clears event status enable register. See Status Commands and Reading the Analyzer's Status Registers.

See Also
- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree
**IDN? - Identify**

Returns a string that uniquely identifies the analyzer. The string is of the form "Agilent Technologies",<model number>,<serial "number>,<software revision>".

**Note:** Beginning with Rev 6.01, this command now returns the software revision with 6 digits instead of 4. For example, A.06.01.02.

---

**OPC - Operation complete command**

Generates the OPC message in the standard event status register when all pending overlapped operations have been completed (for example, a sweep, or a Default). See Understanding Command Synchronization.

---

**OPC? - Operation complete query**

Returns an ASCII "+1" when all pending overlapped operations have been completed. See Understanding Command Synchronization

---

**OPT? - Identify Options Query**

Returns a string identifying the analyzer option configuration.

---

**RST - Reset**

Executes a device reset and cancels any pending *OPC command or query, exactly the same as a SYSTem:PRESet with one exception: Syst:Preset does NOT reset Calc:FORMAT to ASCII. The contents of the analyzer’s non-volatile memory are not affected by this command.

---

**SRE - Service Request Enable**

Before reading a status register, bits must be enabled. This command enables bits in the service request register. The current setting is saved in non-volatile memory. See Status Commands and Reading the Analyzer’s Status Registers.

---

**SRE? - Service Request Enable Query**

Reads the current state of the service request enable register. The register is cleared after reading it. The return value can be decoded using the table in Status Commands. See also Reading the Analyzer's Status Registers.

---

**STB? - Status Byte Query**

Reads the value of the instrument status byte. The register is cleared only when the registers feeding it are cleared. See Status Commands and Reading the Analyzer's Status Registers.

---

**TST? - Result of Self-test Query**

Returns the result of a query of the analyzer hardward status. An 0 indicates no failures found. Any other value indicates one or more of the following conditions exist. The value returned is the Weight (or sum of the Weights) of the existing conditions. For example:

- If 4 is returned from *TST?, an Overpower condition exists.
If 6 is returned, both Unleveled and Overpower conditions exists.

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>Phase Unlock</td>
<td>The source has lost phaselock. This could be caused by a reference channel open or a hardware failure.</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>Unleveled</td>
<td>The source power is unleveled. This could be a source is set for more power than it can deliver at the tuned frequency. Or it could be caused by a hardware failure.</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>Not used</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>EE Write Failed</td>
<td>An attempted write to the EEPROM has failed. This is possibly caused by a hardware failure.</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>YIG Cal Failed</td>
<td>The analyzer was unable to calibrate the YIG. Either the phaselock has been lost or there has been a hardware failure.</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>Ramp Cal Failed</td>
<td>The analyzer was unable to calibrate the analog ramp generator due to a possible hardware failure.</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>Not used</td>
<td></td>
</tr>
</tbody>
</table>

**WAI - Wait**

Prohibits the instrument from executing any new commands until all pending overlapped commands have been completed. See Understanding Command Synchronization

Last Modified:

17-Sep-2008 Added *RST vs Syst:Pres note
**About Triggering**

**Abort Command**

**ABORt**

*(Write-only)* Stops all sweeps - then resume per current trigger settings. This command is the same as **INITiate:IMMediate** (restart) except if a channel is performing a single sweep, ABORt will stop the sweep, but not initiate another sweep.

Learn about [Synchronizing the PNA and Controller](#)

| **Examples** | **ABORt**
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>abort</strong></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Query Syntax</strong></th>
<th>Not applicable</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th><strong>Default</strong></th>
<th>Not applicable</th>
</tr>
</thead>
</table>
Calculate:Correction Commands

Controls error correction functions.

```
CALCulate:CORRection
  EDELay
    | DISTance
    | TIME
    | MEDium
    | UNIT
    | WGCutoff
  [STATe]
    | INDicator?
  TYPE
  OFFSet
    | [MAGNitude]
    | PHASe
```

Click on a blue keyword to view the command details.

Red keywords are superseded.

See Also

- Example Programs
- New Calibrating the PNA Using SCPI
- Synchronizing the PNA and Controller
- SCPI Command Tree

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

```
CALCulate<cnum>:CORRection:EDELay:DISTance <num>
```
(Read-Write) Sets the electrical delay in physical length (distance) for the selected measurement.

**See Critical Note**

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<num>** Electrical delay in distance.

First Specify units using CALC:CORR:EDEL:UNIT

Use SENS:CORR:RVEL:COAX <num> to set Velocity factor.

This parameter supports MIN and MAX as arguments. Learn more.

**Examples**

```
CALC1:CORR:EDEL:DIST 5
```
```
calculate2:correction:distance .003
```

**Query Syntax**

CALCulate<cnum>:CORRection:EDELay:DISTance?

**Return Type**

Numeric

**Default**

0

---

**CALCulate<cnum>:CORRection:EDELay:MEDium <char>**

(Read-Write) Sets the media used when calculating the electrical delay.

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1.
- **<num>** Choose from: **COAX** for coaxial medium, **WAVE**guide for waveguide medium.

**Examples**

```
CALC:CORR:EDEL:MED COAX
```
```
calc3:corr:edelay:medium waveguide
```

**Query Syntax**

CALCulate<cnum>:CORRection:EDELay:MEDium?

**Return Type**

Character

**Default**

COAX

---

**CALCulate<cnum>:CORRection:EDELay:UNIT <char>**
(Read-Write) Sets and returns the units for specifying electrical delay in physical length (distance).

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<char>` Units for delay in distance. Choose from:
  - METer
  - FEET
  - INCH

**Examples**

```
CALC:CORR:EDEL:UNIT MET
calc3:corr:edelay:unit inch
```

**Query Syntax**

```
CALCulate<cnum>:CORRection:EDELay:UNIT?
```

**Return Type** Character

**Default** METer

---

**CALCulate<cnum>:CORRection:EDELay[:TIME] <num>**

(Read-Write) Sets the electrical delay for the selected measurement.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` Electrical delay in seconds. Choose any number between:
  - **-10.00** and **10.00**
  - Use **SENS:CORR:RVEL:COAX** `<num>` to set Velocity factor.

This parameter supports MIN and MAX as arguments. Learn more.

**Examples**

```
CALC1:CORR:EDEL:TIME 1NS
calculate2:correction:time 0.5e-12
```

**Query Syntax**

```
CALCulate:CORRection:EDELay[:TIME]?
```

**Return Type** Numeric

**Default** 0 seconds
**CALCulate\(\text{cnum}\):CORRection:EDELay:WGcutoff \(<\text{num}\)**

(Read-Write) Sets the waveguide cutoff frequency used when the electrical delay media is set to WAVEguide. (See **CALCulate:CORRection:EDELay:ME Di um \(<\text{char}\>\).**)

**Parameters**

- \(<\text{cnum}\>\): Any existing channel number. If unspecified, value is set to 1.
- \(<\text{num}\>\): Waveguide cutoff frequency used with the electrical delay calculation.

This parameter supports MIN and MAX as arguments. Learn more.

**Examples**

- **CALC:CORR:EDEL:WGC 18.067 GHz**
- **calculate3:correction:edelay:wgcutoff 14.047 ghz**

**Query Syntax**

**CALCulate\(\text{cnum}\):CORRection:EDELay:WGcutoff?**

**Return Type**

Numeric

**Default**

45 MHz

---

**CALCulate\(\text{cnum}\):CORRection[:STATE] \(<\text{bool}\>)**

(Read-Write) Turns error correction ON or OFF for the selected measurement on the specified channel.

To turn error correction ON or OFF for a channel, use **SENS:CORR:STATe**.

**See Critical Note**

**Parameters**

- \(<\text{cnum}\>\): Any existing channel number. If unspecified, value is set to 1
- \(<\text{bool}\>\): Correction state. Choose from:
  - 0 - Correction OFF
  - 1 - Correction ON

**Examples**

- **CALC:CORR ON**
- **calculate:correction:state off**

**Query Syntax**

**CALCulate\(\text{cnum}\):CORRection:STATe?**

**Return Type**

Boolean

**Default**

Not Applicable
CALCulate<cnum>:CORRection[:STATE]:INDicator?
(Read-only) Returns the error correction state for the selected measurement on the specified channel.
To turn error correction ON or OFF for a channel, use SENS:CORR:STATe.

See Critical Note

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:CORR:IND?</td>
<td></td>
</tr>
<tr>
<td>calculate2:correction:state:indicator?</td>
<td></td>
</tr>
</tbody>
</table>

Return Type Character

- NONE - No error correction
- MAST (Master) - Original error correction terms
- INT - Error terms are interpolated. Learn more.
- DELT - Delta Match calibration terms. Learn more.
- INV - Error terms are not valid

Default NONE

CALCulate<cnum>:CORRection:TYPE <string>
(Read-Write) Sets the Cal Type for the selected measurement on the specified channel. This is used when a Cal Set is applied. Learn more about applying Cal Types.

- Use SENS:CORR:TYPE:CAT? to list the Cal Types in the PNA.
- Use SENS:CORR:CSET:TYPE:CAT? to list the Cal Types contained in the active Cal Set for the channel.
- Use SENS:CORR:COLL:METH to set the Cal type to perform a new calibration,

See Critical Note

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <string> (String) Cal type. Case sensitive. Use one of the following:

  For Full Calibrations:
This command does not distinguish between TRL and SOLT. The same number of error terms is applied for both Cal Types.

"Full n Port(x,y,z...)"

where

\[ n = \text{the number of ports to calibrate} \]
\[ x,y,z = \text{the port numbers to calibrate} \]

For example:

"Full 7 Port(2,3,4,5,6,7,8)"

**For Response Calibrations:**

"Response(param)" OR

"ResponseAndIsolation(param)"

Where param =

- S-parameter. For example"

  "Response(S21)"

  "ResponseAndIsolation(A/R)"

- Single or ratioed receivers using either logical receiver notation or physical receiver notation. For example:

  "Response(A)"

  "ResponseAndIsolation(a3/b4)"

**For FCA Calibrations:**

- "SMC_2P" (Response + Input + Output) All four sweeps required. Most accurate.

- "SMCRsp+IN" No Output match. All four sweeps required.

- "SMCRsp+OUT" No Output match. All four sweeps required.

- "SMCRsp" No Input or Output match. Saves two sweeps.

For VMC, multiple Cal types are not available.
For Gain Compression Cal

where r = receive port; s = source port

- "GCA 2P (r,s)" - full 2-port cal
- “GCA Enh Resp (r,s)” - Enhanced Response Cal

**Examples**

```
CALC:CORR:TYPE "Scalar Mixer Cal"
```

**Query Syntax**

```
CALCulate<cnum>:CORRection:TYPE?
```

**Return Type**

String

**Default**

Not Applicable

---

**CALCulate<cnum>:CORRection:OFFSet[:MAGNitude] <num> Superseded**

*Note:* This command is replaced with `SENS:CORR:RPOWer:OFFSet[:AMPLitude]`.

To set data trace magnitude offset, use `CALC:OFFS:MAGN`.

This command does NOT function for FCA measurements.

See an example of a [Receiver Power Calibration](#).

*Read-Write*

**For Receiver Power Calibration**, specifies the power level to which the selected (unratioed) measurement data is to be adjusted. This command applies only when the selected measurement is of unratiored power.

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` Cal power level in dBm. No limits are enforced on this value, but the PNA receivers themselves have maximum and minimum power specifications (that may differ between PNA models) which this value must comply with for a valid receiver power cal.

**Examples**

```
CALC:CORR:OFFS 10DBM
```

```
CALCulate1:correction:offset:magnitude maximum
```

**Query Syntax**

```
CALCulate<cnum>:CORRection:OFFSet[:MAGNitude]?
```

**Return Type**

Numeric

**Default**

0dBm
CALCulate<cnum>:CORRection:OFFSet:PHASe <num>[<char>]  Superseded

Note: This command is replaced with CALC:OFFS:PHASE

(Read-Write) Sets the phase offset for the selected measurement.

Parameters

<cnum>  Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<num>  Offset phase value. Choose any number between: -360 and 360

<char>  Units for phase. OPTIONAL. Choose either:
  DEG - Degrees (default)
  RAD - Radians

Examples

CALC:CORR:OFFS:PHAS 10  
calculate:correction:offset:phase 20rad

Query Syntax  CALCulate:CORRection:OFFSet:PHASe?

ReturnType  Numeric, returned value always in degrees

Default  0 degrees

Last modified:

19-Jul-2010  Added Calc:Corr:Indicator
22-Sep-2009  Removed VMC from Corr:Type
6-Feb-2009  Added two commands
12-Feb-2008  Fixed typo
9/12/06  MQ Modified Calc:Corr for multiport.
**Calculate:Custom Commands**

Creates and modifies application measurements:

```
CALCulate:CUStom:
    DEFine
    MODify
```

See Also

- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

---

**CALCulate<cnum>:CUStom:DEFine <Mname>, <type> [,param]**

*(Write-only)* Creates a custom measurement. The custom measurement is not automatically displayed. You must also do the following:

- Use `DISP:WIND:STATE` to create a window if it doesn't already exist.
- Use `DISP:WIND:TRAC:FEED` to display the measurement
- Select the measurement (`CALC:PAR:SEL`) before making additional settings.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<Mname>` Name of the measurement. Any non-empty, unique string, enclosed in quotes.
- `<type>` String. The type of custom measurement. Click each to see an example program. Choose from:
  - "Vector Mixer/Converter"
  - "Scalar Mixer/Converter"
  - "Noise Figure Cold Source"
  - "Noise Figure Converters"
  - "Gain Compression"
  - "Gain Compression Converters"
  - "Swept IMD"
  - "IM Spectrum"
- "Swept IMD Converters"
- "IM Spectrum Converters"

[param] String. Optional parameter specifies the measurement to create:

<table>
<thead>
<tr>
<th>Meas Class</th>
<th>Measurement Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vector Mixer/Converter</td>
<td></td>
<td>&quot;S11&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;VC21&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;S22&quot;</td>
</tr>
</tbody>
</table>

| | | Learn about VMC parameters |
| Scalar Mixer/Converter |                     | "S11"                      |
|                       |                   | "SC21"                     |
|                       |                   | "SC12"                     |
|                       |                   | "S22"                      |
|                       |                   | "Ipwr"                     |
|                       |                   | "RevIpwr"                  |
|                       |                   | "Opwr"                     |
|                       |                   | "RevOPwr"                  |

| Gain Compression      |                   | "CompIn21"                 |
|                      |                   | Input power at the compression point. |
|                      |                   | "CompOut21"                |
|                      |                   | Output power at the compression point. |
|                      |                   | "CompGain21"               |
|                      |                   | Gain at the compression point. |
|                      |                   | "CompS11"                  |
|                      |                   | Input Match at the compression point |
|                      |                   | "RefS21"                   |
|                      |                   | Linear Gain                |
|                      |                   | "DeltaGain21"              |
|                      |                   | CompGain21 - Linear Gain   |
|                      |                   | "S11", "S21", "S12", "S22"|
|                      |                   | Standard S-parameters; measured at port 1 and port 2 |

| Gain Compression      |                   | "S11"                      |
|                      |                   | "SC21"                     |

| GCX - All Gain Compression parameters (except S21 and S12) plus the following: |

2247
<table>
<thead>
<tr>
<th>Noise Figure AND NFX:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;NF&quot;</td>
<td>Noise figure</td>
</tr>
<tr>
<td>&quot;ENR&quot;</td>
<td>Validate noise source measurements.</td>
</tr>
<tr>
<td>&quot;T-Eff&quot;</td>
<td>Effective noise temperature.</td>
</tr>
<tr>
<td>&quot;DUTRN&quot; &quot;DUTRNPI&quot;</td>
<td>DUT noise power ratio. (Noise power expressed in Kelvin divided by 290).</td>
</tr>
<tr>
<td>&quot;SYSRN&quot; &quot;SYSRNPI&quot;</td>
<td>System noise power ratio</td>
</tr>
<tr>
<td>&quot;DUTNP&quot; &quot;DUTNPDI&quot;</td>
<td>DUT noise power density. (Noise power expressed in dBm/Hz).</td>
</tr>
<tr>
<td>&quot;SYSNP&quot; &quot;SYSNPDI&quot;</td>
<td>System noise power density.</td>
</tr>
<tr>
<td>&quot;OvrRng&quot;</td>
<td>Indication that the noise receiver is being over powered.</td>
</tr>
<tr>
<td>&quot;T-Rcvr&quot;</td>
<td>Temperature reading (in Kelvin) of the noise receiver board.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Noise Figure ONLY - NOT NFX:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameters</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>&quot;A,1&quot;,&quot;A,2&quot; ...and so forth.</td>
<td>Unratioed parameters; with notation: &quot;receiver, source port&quot;</td>
</tr>
<tr>
<td><strong>NFX ONLY:</strong></td>
<td></td>
</tr>
<tr>
<td>&quot;S11&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;SC21&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;SC12&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;S22&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;Ipwr&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;RevIPwr&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;Opwr&quot;</td>
<td></td>
</tr>
<tr>
<td>&quot;RevOPwr&quot;</td>
<td>Mixer parameters</td>
</tr>
<tr>
<td>&quot;ALO1&quot;,&quot; BLO1&quot; ...and so forth.</td>
<td>Test port receiver at LO1 frequency</td>
</tr>
<tr>
<td>&quot;R1_1&quot;, &quot;B_2&quot; ..and so forth.</td>
<td>Unratioed parameters with notation: &quot;receiver_source port&quot;</td>
</tr>
</tbody>
</table>

Swept IMD

There are over 150 possible Swept IMD parameters, too many to list here.

Build the parameters with the [Swept IMD Parameter](#) dialog, then copy the parameter name to the remote command.

The following are a few example parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;PwrMainLo&quot;</td>
<td>Absolute power of the Low tone at the DUT output.</td>
</tr>
<tr>
<td>&quot;IM3&quot;</td>
<td>Power of the third product relative to the average power of the f1 and f2 tones measured at the DUT output.</td>
</tr>
<tr>
<td>&quot;OIP3&quot;</td>
<td>Theoretical power level at which the third product will be the same power level as the average of the main tones at the output of the DUT.</td>
</tr>
</tbody>
</table>

View signals OUT of the 2249
<table>
<thead>
<tr>
<th>IM Spectrum</th>
<th>&quot;Output&quot;</th>
<th>DUT and into PNA port 2 (B receiver).</th>
</tr>
</thead>
<tbody>
<tr>
<td>IM Spectrum Converters</td>
<td>&quot;Input&quot;</td>
<td>View signals IN to the DUT (R1 receiver).</td>
</tr>
<tr>
<td></td>
<td>&quot;Reflection&quot;</td>
<td>View signals reflected off the DUT input and back into PNA port 1 (A receiver)</td>
</tr>
</tbody>
</table>

**Examples**

```
calc2:cust:define 'MyNF', 'NoiseFigure', 'NF'
calc2:cust:define 'My VC21', 'Vector Mixer/Converter', 'S22'
```

**Query Syntax**

Not applicable

**Overlapped?**

No

**Default**

Not applicable

**CALCulate<cnum>:CUSTom:MODify <param>**

*(Write-only)* Changes the selected custom measurement to a different parameter.

See an example using this command for a VMC and SMC measurement

**Parameters**

- `<cnum>`: Channel of the custom measurement to be changed. First, select the measurement using `CALC:PAR:SEL`.
- `<param>`: Parameter to change the custom measurement to. Select a parameter that is valid for the type of measurement. Choose from the same arguments as `Calc:Cust:Def`.

**Examples**

```
syst:pres
calc2:cust:define 'My VC21', 'Vector Mixer/Converter'
calc:par:sel 'My VC21'
calc2:cust:mod 'S22'
```

**Query Syntax**

Not applicable

**Overlapped?**

No

**Default**

Not applicable
<table>
<thead>
<tr>
<th>Date</th>
<th>Event Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-Mar-2010</td>
<td>Fixed IM spectrum converters</td>
</tr>
<tr>
<td>27-Feb-2009</td>
<td>Added IMD Converters and Noise</td>
</tr>
<tr>
<td>8-Sep-2008</td>
<td>Added IMD arguments</td>
</tr>
<tr>
<td>23-Aug-2007</td>
<td>Added Noise and GC Arguments</td>
</tr>
</tbody>
</table>
Calculate:Data Commands

Controls writing and reading PNA measurement data.

| CALCulate:DATA | CUSTom | CATalog? | SNP? | PORTs? | SAVE |

Click on a blue keyword to view the command details.

Red is a superseded command.

See Also

- Example Programs
- Data Access Map
- Synchronizing the PNA and Controller
- To read receiver data, use CALC:RDATA?
- To read error terms, use SENS:CORR:CSET:DATA
- To read SnP measurement data, use CALC:DATA:SNP?
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

(Write) CALCulate<cnum>:DATA <char>,<data>

(Read) CALCulate<cnum>:DATA? <char>

Reads or writes Measurement data, Memory data, or Normalization Divisor data from the Data Access Map location.

- For Measurement data, use FDATA or SDATA
- For Memory data, use FMEM or SMEM. When querying memory, you must first store a trace into memory
using `CALC:MATH:MEMorize`.

- For Normalization Divisor (Receiver Power Cal error term) data, use `SDIV`.
- For write operations, data type depends on `FORM:DATA` command.

**Equation Editor** Notes:

- When equation editor is active on a trace in a standard S-parameter channel, Calc:Data returns the data from the parameter on the trace that was measured last. For example, for the equation “S22 + S33 + S11”, then S33 is the last measured parameter because it uses source port 3.

- In applications, if equation editor is active and the original parameter for the trace is not requested anywhere in the channel, then zeros are returned. If the original parameter is being measured within the channel, then data for the original parameter is returned.

- In general, if an equation contains no measurement parameters, then data for the original parameter is returned.

**Note:** The Calc:Data SCORR command to read / write error terms is **Superseded** with `SENS:CORR:CSET:DATA`. SCORR commands do NOT accommodate greater than 12 error terms.

See Critical Note

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<char>` **FDATA** Formatted measurement data to or from Data Access Map location `Display` (access point 2).
  - Corrected data is returned when correction is ON.
  - Uncorrected data is returned when correction is OFF.
  - Returns TWO numbers per data point for Polar and Smith Chart format.
  - Returns one number per data point for all other formats.
  - Format of the read data is same as the displayed format.

- **SDATA** Complex measurement data.
  - Writes data to Data Access Map location `Raw Measurement` (access point 0).
    - When writing corrected data, and correction is ON, it will be corrected again, resulting in meaningless data.
  - Reads data from `Apply Error Terms` (access point 1).
- Returns TWO numbers per data point.
- Corrected data is returned when correction is ON.
- Uncorrected data is returned when correction is OFF.

**FMEM** Formatted memory data to or from Data Access Map location Memory result (access point 4).

- Returns TWO numbers per data point for Polar and Smith Chart format.
- Returns one number per data point for all other formats.
- Format of the read data is same as the displayed format.
- Returned data reflects the correction level (On|OFF) when the data was stored into memory.

**SMEM** Complex measurement data to or from Data Access Map location Memory (access point 3).

- Returns TWO numbers per data point.
- Returned data reflects the correction level (On|OFF) when the data was stored into memory.
- Returned data reflects the correction level (On|OFF) when the data was stored into memory.

**SDIV** Complex data from Data Access Map location Normalization (5).

- Returns TWO numbers per data point.
- If normalization interpolation is ON and the number of points changes after the initial normalization, the divisor data will then be interpolated.
- When querying the normalization divisor, you must first store a divisor trace using CALC:NORMALize[:IMMediate].

The following Calc:Data SCORR command to read / write error terms is **Superseded** with SENS:CORR:CSET:DATA. These SCORR commands do NOT accommodate greater than 12 error terms.
For 2-Port SOLT and TRL calibrations

Specify this `<char>` to get or put this Error Term...

<table>
<thead>
<tr>
<th>SCORR1</th>
<th>Forward Directivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCORR2</td>
<td>Forward Source Match</td>
</tr>
<tr>
<td>SCORR3</td>
<td>Forward Reflection Tracking</td>
</tr>
<tr>
<td>SCORR4</td>
<td>Forward Isolation</td>
</tr>
<tr>
<td>SCORR5</td>
<td>Forward Load Match</td>
</tr>
<tr>
<td>SCORR6</td>
<td>Forward Transmission Tracking</td>
</tr>
<tr>
<td>SCORR7</td>
<td>Reverse Directivity</td>
</tr>
<tr>
<td>SCORR8</td>
<td>Reverse Source Match</td>
</tr>
<tr>
<td>SCORR9</td>
<td>Reverse Reflection Tracking</td>
</tr>
<tr>
<td>SCORR10</td>
<td>Reverse Isolation</td>
</tr>
<tr>
<td>SCORR11</td>
<td>Reverse Load Match</td>
</tr>
<tr>
<td>SCORR12</td>
<td>Reverse Transmission Tracking</td>
</tr>
</tbody>
</table>

**EXAMPLE**

```
CALC:DATA, FDATA, Data(x)
calculate2: data sdata, data(r,i)
```

See another example using this command.

Return Type: **Block data**

Default - Not Applicable

**CALCulate<cnum>:DATA:CUStom <name>,<data> Superseded**
**Note:** This command has been replaced by `CALC:DATA:` which can now be used with all PNA applications.

(Read-Write) Reads or writes data from a custom-named measurement buffer.

See Critical Note

<table>
<thead>
<tr>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>&lt;cnum&gt;</strong> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td><strong>&lt;name&gt;</strong> Name of the buffer to be read or written</td>
</tr>
<tr>
<td><strong>&lt;data&gt;</strong> Data to be read or written to the custom buffer. Format as one number per data point.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:DATA:CUST 'VectorResult0',0,1,2,3,4,5 Write</code></td>
<td>Write</td>
</tr>
<tr>
<td><code>CALC:DATA:CUST? 'VectorResult0' Read</code></td>
<td>Read</td>
</tr>
</tbody>
</table>

**Query Syntax**

`CALCulate:DATA:CUSTom? <name>`

**Return Type**

Depends on Form:Data

**Default**

Not Applicable

---

**CALCulate<cnum>:DATA:CUSTom:CATalog?** Superseded

**Note:** This command has been replaced by `CALC:DATA:CAT` which can now be used with all PNA applications.

(Read-only) Reads the list of buffer names (comma separated list of string values) available from the selected parameter. Specify the measurement using `CALCulate:PARameter:SELect`.

See Critical Note

<table>
<thead>
<tr>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>&lt;cnum&gt;</strong> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:DATA:CUST:CAT?</code></td>
<td></td>
</tr>
<tr>
<td><code>calculate:data:custom:catalog?</code></td>
<td></td>
</tr>
</tbody>
</table>

**Return Type**

String

**Default**

Not Applicable
**CALCulate<cnum>:DATA:SNP? <n> Superseded**

**Note:** This command has been replaced by `CALC:DATA:SNP:PORTs?` (Read-only) Reads SnP data from the selected measurement. Learn more about SnP data.

**Note:** This command returns SNP data without header information, and in columns, not in rows as .SnP files. This means that the data returned from this command sends all frequency data, then all Sx1 magnitude or real data, then all Sx1 phase or imaginary data, and so forth.

See Critical Note

### Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<n>** Amount of data to return. If unspecified, <n> is set to 2. The number you specify must be less than or equal to the number of available ports on the PNA.

Choose from:

- **1 (S1P)** returns data for the active measurement.
- **2 (S2P)** returns data for the 2 port parameters associated with the current measurement. Default.
- **3 (S3P)** returns data for the 3 port parameters associated with the current measurement.
- **4 (S4P)** returns data for the 4 port parameters associated with the current measurement.

SnP data can be output using several data formatting options. See `MMEM:STOR<file>.<snp>.

See also `MMEM:STOR <file>.<snp>`

### Examples

```
CALC:PAR:DEF MyMeasurement, S11
CALC:PAR:SEL MyMeasurement
CALC:DATA:SNP? 1
```

### Return Type

- **Default** Not Applicable

---

**CALCulate<cnum>:DATA:SNP:PORTs? <"x,y,z".>**
Note: This command replaces \texttt{CALC\textasciitilde DATA\textasciitilde SNP}\textdollar. This command is more explicit regarding the data to be returned, and works for PNAs with multiport test sets.

(Read-only) Reads SNP data from the selected measurement for the specified ports. Learn more about SNP data.

Note: This command returns SNP data without header information, and in columns, not in rows as .SnP files. This means that the data returned from this command sends all frequency data, then all Sx1 magnitude or real data, then all Sx1 phase or imaginary data, and so forth.

\textbf{See Critical Note}

\textbf{Parameters}

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<"x,y,z">` Comma or space delimited port numbers for which data is requested, enclosed in quotes.

SnP data can be output using several data formatting options. See \texttt{MMEM\textasciitilde STORE\textasciitilde TRACe\textasciitilde FORMat\textasciitilde SNP}.

\textbf{Examples} \texttt{CALC\textasciitilde DATA\textasciitilde SNP\textasciitilde PORTs? "1,2,4,5,7"} 'read data for these ports

\textbf{Return Type} Depends on \texttt{FORMat\textasciitilde DATA}

\textbf{Default} Not Applicable

\texttt{CALCulate<cnum>-DATA\textasciitilde SNP\textasciitilde PORTs\textasciitilde SAVE <"x,y,z">,<filename>}

Note: This command replaces \texttt{MMEM\textasciitilde STORE\textasciitilde sNp}. This command is more explicit regarding the data to be saved, and works for PNAs with multiport test sets.

(Write-only) Saves sNp data from the selected measurement for the specified ports.

Learn more about SNP data.

\textbf{See Critical Note}

\textbf{Parameters}

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<"x,y,z">` \textbf{String} Comma or space delimited port numbers for which data is requested, enclosed in quotes.
<filename> String Path, filename, and suffix of location to store the sNp data. The suffix is not checked for accuracy. If saving 2 ports, specify "filename.s2p"; If saving 4 ports, specify "filename.s4p.", and so forth.

SnP data can be output using several data formatting options. See MMEM:STORe:TRACe:FORMat:SNP.

Examples

CALC:DATA:SNP:PORTs:Save '1,2,4', 'C:/Program Files/Agilent/Network Analyzer/Documents/MyData.s3p'

Return Type Depends on FORMat:DATA

Default Not Applicable

Last modified:

18-May-2009 Superseded Custom commands
24-Mar-2009 Edited SNP note
26-Apr-2007 Added clarification to Calc:Data SDATA
9/18/06 MQ Added two SNP Ports commands for multiport
Calculate: Equation Commands

Controls Equation Editor capabilities.

<table>
<thead>
<tr>
<th>CALCulate:EQUation:LIBRary</th>
</tr>
</thead>
<tbody>
<tr>
<td>FUNCtions</td>
</tr>
<tr>
<td>IMPort?</td>
</tr>
<tr>
<td>REMove</td>
</tr>
<tr>
<td>STATE</td>
</tr>
<tr>
<td>TEXT</td>
</tr>
<tr>
<td>VALid?</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Equation Editor
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using CALC:Par:Select.

CALCulate<cnum>:EQUation:LIBRary:FUNCtions <string>

(Read-only) Returns the functions in the specified DLL.

See Critical Note

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <string> Full path and filename of the *.dll to be read.

Examples

functions = CALC:EQU:LIBR:FUNC "C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll"

Query Syntax

CALCulate<cnum>:EQUation:LIBRary:FUNCtions?
**Return Type**  Comma delimited string of function names.

**Default**  Not Applicable

---

**CALCulate\(\text{cnum}\):EQUation:LIBRary:IMPort \(<\text{string}>\)**

**(Read-Write)** Imports the functions in the specified DLL and returns whether the functions have been imported into the PNA.

See Critical Note

**Parameters**

- **\(<\text{cnum}>\)**  Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, \(<\text{cnum}>\) is set to 1.

- **\(<\text{string}>\)**  Full path and filename of the \*.dll.

**Examples**

<table>
<thead>
<tr>
<th>Write - Imports functions</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:EQU:LIBR:IMPort &quot;C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll&quot;</code></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Read if Imported</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>functions = CALC:EQU:LIBR:IMPort &quot;C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll&quot;</code></td>
</tr>
</tbody>
</table>

**Query Syntax**  CALCulate\(\text{cnum}\):EQUation:LIBRary:IMPort?

Returns the following:

- 1 - Imported
- 0 - NOT imported

**Return Type**  Boolean

**Default**  Not Applicable

---

**CALCulate\(\text{cnum}\):EQUation:LIBRary:REMove \(<\text{string}>\)**
(Write-only) Removes an imported an Equation Editor DLL from the PNA.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<string>` Full path and filename of the *.dll.

Examples

```
CALC:EQU:LIBR:REM "C:/Program Files/Agilent/Network Analyzer/UserFunctions/Expansion.dll"
```

Query Syntax Not Applicable

Default Not Applicable

CALCulate<cnum>:EQUation[:STATe] <bool>

(Read-Write) Turns ON and OFF the equation on selected measurement for the specified channel. If the equation is not valid, then processing is not performed. Use `CALC:EQUation:VALid?` to ensure that the equation is valid.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<bool>`
  - **ON** (or 1) - turns equation ON.
  - **OFF** (or 0) - turns equation OFF.

Examples

```
CALC:EQU 1
calculate2:equation:state 0
```

Query Syntax CALCulate<cnum>:EQUation[:STATe]?

Return Type Boolean

Default OFF (0)

CALCulate<cnum>:EQUation:TEXT <string>
Specifies an equation or expression to be used on the selected measurement for the specified channel.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.

- `<string>` Any valid equation or expression. See Equation Editor.

Examples

```
'Equation (includes '=')
CALC:EQU:TEXT "foo=S11/S21"

'Expression
calculate2:equation:text "S11/S21"
```

Query Syntax

```
CALCulate<cnum>:EQUation:TEXT?
```

Return Type

String

Default Not Applicable

---

**CALCulate<cnum>:EQUation:VALid?**

(Read-Write) Returns a boolean value to indicate if the current equation on the selected measurement for the specified channel is valid. For equation processing to occur, the equation must be valid and ON (CALC:EQU:STAT 1).

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.

Examples

```
CALC:EQU:VAL?
calculate2:equation:valid?
```

Return Type

Boolean

1 - equation is valid
0 - equation is NOT valid

Default Not Applicable
Calculate: Filter Commands

Controls the gating function used in time domain measurements. The gated range is specified with either (start / stop) or (center / span) commands.

```
CALCulate:FILTER[:GATE]
```

COUPLE

TIME

CENTEr   SPAN   STATe   [TYPE]

PARameters   SHApe   START   STOP

Click on a blue keyword to view the command details.

**see Also**

- [Example Programs](#)
- [Learn about Gating](#)
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using `Calc:Par:Select`

```
CALCulate<cnum>:FILTER:GATE1:COUPLE:PARameters <num>
```
CALCulate<cnum>:FILTer[:GATE]:COUPle:PARameters

Specifies the time domain gating parameters to be coupled. The settings for those parameters will be copied from the selected measurement to all other measurements on the channel.

- To enable Trace Coupling, use SENS:COUP:PAR
- To specify Transform parameters to couple, use CALC:TRAN:COUP:PAR

Learn more about Time Domain Trace Coupling

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` (Numeric) Parameters to couple. To specify more than one parameter, add the numbers.
  - 1 - Gating Stimulus (Start, Stop, Center, and Span TIME settings.)
  - 2 - Gating State (ON / OFF)
  - 4 - Gating Shape (Minimum, Normal, Wide, and Maximum)
  - 8 - Gating Type (Bandpass and Notch)

Examples

- 'To couple all parameters:
  
  CALC:FILT:COUP:PAR 15

- 'To couple Stimulus and Type:
  
  calculate2:filter:gate:couple:parameters 9

Query Syntax

CALCulate<cnum>:FILTer:<GATE>:COUPle:PARameters?

Return Type

Numeric

Default

13 (All parameters except 2 - Gating State)
(Read-Write) Sets the gate filter center time.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` Center time in seconds; Choose any number between:
  \[ \pm \left(\frac{\text{number of points}-1}{\text{frequency span}}\right) \]

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
calculate2:filter:time:shape normal
```

**Query Syntax**

```
CALCulate<cnum>:FILTer[:GATE]:TIME:CENTer?
```

**Return Type**

Numeric

**Default**

0

---

**CALCulate<cnum>:FILTer[:GATE]:TIME:SHAPe <char>**

(Read-Write) Sets the gating filter shape when in time domain.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<char>` Choose from
  - MAXimum - the widest gate filter available
  - WIDE -
  - NORMal -
  - MINimum - the narrowest gate filter available

**Examples**

```
CALC:FILT:GATE:TIME:SHAPE MAX
```

**Query Syntax**

```
CALCulate<cnum>:FILTer[:GATE]:TIME:SHAPe?
```

**Return Type**

Character

**Default**

NORMal
CALCulate<cnum>:FILTER[:GATE]:TIME:SPAN <num>

(Read-Write) Sets the gate filter span time.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<num> Time span in seconds; Choose any number between: 0 and 2* [(number of points-1) / frequency span]

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

CALC:FILT:GATE:TIME:SPAN 5 ns
calculate2:filter:time:span maximum

Query Syntax

CALCulate<cnum>:FILTER[:GATE]:TIME:SPAN?

Return Type

Numeric

Default 20 ns

CALCulate<cnum>:FILTER[:GATE]:TIME:STATE <boolean>

(Read-Write) Turns gating state ON or OFF.

See Critical Note

Note: Sweep type must be set to LInear Frequency in order to use Transform Gating.

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<boolean> ON (or 1) - turns gating ON.
OFF (or 0) - turns gating OFF.

Examples

CALC:FILT:TIME:STAT ON
calculate2:filter:gate:time:state off

Query Syntax

CALCulate<cnum>:FILTER[:GATE]:TIME:STATE?

Return Type

Boolean (1 = ON, 0 = OFF)

Default OFF
CALCulate<cnum>:FILTer[:GATE]:TIME:STARt <num>

(Read-Write) Sets the gate filter start time.

See Critical Note

Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<num>** Start time in seconds; any number between:
  \[\pm \frac{(number\ of\ points-1)}{frequency\ span}\]

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

- `CALC:FILT:TIME:STAR 1e-8`
- `calculate2:filter:gate:time:start minimum`

Query Syntax

CALCulate<cnum>:FILTer[:GATE]:TIME:STARt?

Return Type

Numeric

Default

10 ns

CALCulate<cnum>:FILTer[:GATE]:TIME:STOP <num>

(Read-Write) Sets the gate filter stop time.

See Critical Note

Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<num>** Stop time in seconds; any number between:
  \[\pm \frac{(number\ of\ points-1)}{frequency\ span}\]

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

- `CALC:FILT:TIME:STOP -1 ns`
- `calculate2:filter:gate:time:stop maximum`

Query Syntax

CALCulate<cnum>:FILTer[:GATE]:TIME:STOP?

Return Type

Numeric

Default

10 ns
CALCulate<cnum>:FILTer[:GATE]:TIME[:TYPE] <char>

(Read-Write) Sets the type of gate filter used.

See Critical Note

Parameters

- <cnum>: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <char>: Choose from:
  - BPASs - Includes (passes) the range between the start and stop times.
  - NOTCh - Excludes (attenuates) the range between the start and stop times.

Examples

- CALC:FILT:TIME BPAS
- calculate2:filter:gate:time:type notch

Query Syntax

CALCulate<cnum>:FILTer[:GATE]:TIME[:TYPE]?

Return Type

Character

Default

BPAS
Calculate:Format Commands

CALCulate:
  FORMat
  UNIT

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

See Also

- Example using this command.
- Learn About Data Format
- Synchronizing the PNA and Controller

CALCulate<cnum>:FORMat <char>

(Read-Write) Sets the display format for the measurement.

See Critical Note

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <char> Choose from:
  - MLINear
  - MLOGarithmic
  - PHASE
  - UPHase 'Unwrapped phase
  - IMAGinary
  - REAL
  - POLar
  - SMITH
  - SADMittance 'Smith Admittance
  - SWR
  - GDElay 'Group Delay
Examples

```
CALC:FORM MLIN
calculate2:format polar
```

Query Syntax

```
CALCulate<cnum>:FORMat?
```

Return Type

Character

**Default**

MLINear

---

**CALCulate<cnum>:FORMat:UNIT <dataFormat>, <units>**

*(Read-Write)* Sets and returns the units for the specified data format. Measurements with display formats other than those specified are not affected.

See Critical Note

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<dataFormat>` Choose MLOG.
- `<units>` For unratioed measurements, choose from:
  - **DBM** Units are displayed in dBm. 0 dBm = 0.001 watt
  - **DBMV** Units are displayed in dBmV. 0 dBmV = 0.001 volt

Examples

```
CALC:FORM MLOG, DBM
calculate2:format mlog, dbmv
```

Query Syntax

```
CALCulate<cnum>:FORMat:UNIT? <dataFormat>
```

Return Type

Character

**Default**

DBM

---

Last Modified:

- 25-Aug-2008 Added Units
- 1-Oct-2007 Added temperature formats
**CALCulate:FSIMulator**

- **BALun**  More commands
- **EMBed**  More commands
- **SENDed**  More commands
- **SNP:EXTRapolate**
- **STATe**

Click a blue keyword to view the command details.

See Also

- Example Programs
- SCPI Command Tree

---

**CALCulate<cnum>:FSIMulator:SNP:EXTRapolate <bool>**

*(Read-Write)*  Turns ON and OFF SNP file extrapolation for both 2-port and 4-port embedding/de-embedding. [Learn more.](#)

**Note:** This command affects ALL measurements on the specified channel.

**Parameters**

- **<cnum>**  Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<bool>**  Choose from:
  - **ON or 1** - Turns Extrapolation ON
  - **OFF or 0** - Turns Extrapolation OFF

**Examples**

```plaintext
CALC:FSIM:SNP:EXTR 1
calculate2:fsimulator:snp:extrapolate 0
```

**Query Syntax**

CALCulate<cnum>:FSIMulator:SNP:EXTRapolate?

**Return Type**  Boolean

**Default**  OFF
CALCulate<cnum>:FSIMulator:STATe <bool>

(Read-Write) Turns all three fixturing functions (de-embedding, port matching, impedance conversion) ON or OFF for all ports on the specified channel. Does not affect port extensions.

**Note:** This command affects ALL measurements on the specified channel.

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<bool>** Choose from:
  - **ON or 1 -** Turns Fixturing ON
  - **OFF or 0 -** Turns Fixturing OFF

**Examples**

```
CALC:FSIM:STAT 1
calculate2:fsimulator:state 0
```

**Query Syntax**

```
CALCulate<cnum>:FSIMulator:STATe?
```

**Return Type**

Boolean

**Default**

OFF

---

**Last Modified:**

16-Nov-2010  Added extrapolate
**Calculate:Function Commands**

CALC : FUNCTION

DATA?  DOMain  EXECute  STATistics  TYPE

USER  [STATE]

[RANGE]  START  STOP

Click on a blue keyword to view the command details.

**see Also**

- Example Programs
- Learn about Trace Statistics
- Synchronizing the PNA and Controller
- SCPI Command Tree

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

**CALCulate<cnun>:FUNCTION:DATA?**

(Read-only) Returns the trace statistic data for the selected statistic type for the specified channel. Select the type of statistic with CALC:FUNC:TYPE.

**See Critical Note**

**Parameters**

- `<cnun>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnun>` is set to 1.

**Return Type** Depends on FORM:DATA

**Example** CALCulate2:FUNCTION:DATA?

**Default** Not applicable

**CALCulate<cnun>:FUNCTION:DOMain:USER[:RANGE] <range>**
(Read-Write) Sets the range used to calculate trace statistics. Each channel has 16 user ranges. The x-axis range is specified with the CALC:FUNC:DOM:USER:START and STOP commands.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<range>` Range number. Choose from: 0 to 16
  - 0 is Full Span of the current x-axis range
  - 1 to 16 are user-specified ranges

Examples

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:FUNC:DOM:USER 4</td>
<td>Numeric</td>
<td>0 - Full Span</td>
</tr>
<tr>
<td>calculate2:function:domain:user:range 0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**CALCulate<cnum>:FUNCtion:DOMain:USER:STARt <range>, <start>**

(Read-Write) Sets the start of the specified user-domain range.

To apply this range, use CALC:FUNC:DOM:USER

To set the stop of the range, use CALC:FUNC:DOM:USER:STOP.

See Critical Note

Note: This command does the same as CALC:MARK:FUNC:DOM:USER:STAR

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<range>` Range number that will receive the start value. Choose an integer between 1 and 16
- `<start>` Start value of the specified range. Choose a real number between: the analyzer's Minimum and Maximum x-axis value.

Examples

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:FUNC:DOM:USER:STAR 1,1e9</td>
<td></td>
<td></td>
</tr>
<tr>
<td>calculate2:function:domain:user:start 2,2e9</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Query Syntax**
CALCulate<cnum>:FUNCtion:DOMain:USER:STARt? <range>

**Return Type**
Numeric

**Default**
The analyzer's Minimum x-axis value

---

CALCulate<cnum>:FUNCtion:DOMain:USER:STOP <range>, <stop>

(Read-Write) Sets the stop value of the specified user-domain range.

To apply this range, use CALC:FUNC:DOM:USER.

To set the start of the range, use CALC:FUNC:DOM:USER:START

**See Critical Note**

**Note:** This command does the same as CALC:MARK:FUNC:DOM:USER:STOP

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<range>` Range number that will receive the stop value. Choose an integer between 1 and 16
- `<stop>` Stop value of the specified range. Choose a real number between: the analyzer's Minimum and Maximum x-axis value.

**Examples**
CALC:FUNC:DOM:USER:STOP 4,5e9  
calculate2:func:domain:user:stop 3,8e9

**Query Syntax**
CALCulate<cnum>:FUNCtion:DOMain:USER:STOP? <range>

**Return Type**
Numeric

**Default**
The analyzer's Maximum x-axis value

---

CALCulate<cnum>:FUNCtion:EXECute
(Write-only) For the active trace of specified channel, executes the statistical analysis specified by the
**CALC:FUNC:TYPE** command.

**See Critical Note**

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

  **Examples**
  
  ```
  CALC:FUNC:EXEC
  calculate2:function:execute
  ```

  **Query Syntax** Not Applicable

  **Default** Not Applicable

**CALCulate<cnum>:FUNCTION:STATistics[:STATE] <ON|OFF>**

(Read-Write) Displays and hides the trace statistics (peak-to-peak, mean, standard deviation) on the screen.

The analyzer will display either measurement statistics or Filter Bandwidth statistics; not both.

**See Critical Note**

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

- **<ON|OFF>** ON - Displays trace statistics

  OFF - Hides trace statistics

  **Examples**
  
  ```
  CALC:FUNC:STAT ON
  calculate2:function:statistics:state off
  ```

  **Query Syntax** CALCulate<cnum>:FUNCTION:STATistics[:STATE]?

  **Return Type** Boolean (1 = ON, 0 = OFF)

  **Default** OFF (0)

**CALCulate<cnum>:FUNCTION:TYPE <char>**
(Read-Write) Sets statistic TYPE that you can then query using \texttt{CALC:FUNCtion:DATA?}.

**Note:** In PNA releases 4.2 and prior, this command applied the statistic type to all measurements. Now, this command affects only the selected measurement on the specified channel.

\begin{itemize}
\item **See Critical Note**
\item **Parameters**
\begin{itemize}
\item \texttt{<cnum>} Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, \texttt{<cnum>} is set to 1.
\item \texttt{<char>} Choose from:
\begin{itemize}
\item \texttt{PTPeak} - the difference between the max and min data points on the trace.
\item \texttt{STDEV} - standard deviation of all data points on the trace
\item \texttt{MEAN} - mean (average) of all data points on the trace
\item \texttt{MIN} - lowest data point on the trace
\item \texttt{MAX} - highest data point on the trace
\end{itemize}
\end{itemize}
\item **Examples**
\begin{itemize}
\item \texttt{CALC:FUNC:TYPE PTP}
\item \texttt{calculate2:function:type stdev}
\end{itemize}
\item **Query Syntax** \texttt{CALCulate\textless cnum\textgreater :FUNCtion:TYPE?}
\item **Return Type** Character
\item **Default** \texttt{PTPeak}
\end{itemize}
Calc:GCData Commands

Reads Gain Compression data from the current Gain Compression acquisition.

<table>
<thead>
<tr>
<th>CALCulate:GCData:</th>
<th>DATA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IMAG</td>
</tr>
<tr>
<td></td>
<td>ITERations</td>
</tr>
<tr>
<td></td>
<td>REAL</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

Other Gain Compression commands

The calibration commands listed in this topic are supplemental to the Guided Cal commands.

- **CALC:CUSTom:DEFINE** - creates a gain compression measurement.
- **SENS:GCSetup** - Most Gain Compression settings.
- **CALC:GCMeas:ANAL** - Gain Compression Analysis settings
- Gain compression data can also be saved to a *.csv file. Learn how.

See Also

- **Example Programs**
- **Learn about Gain Compression Application**
- **Synchronizing the PNA and Controller**
- **SCPI Command Tree**

CALCulate<ch>:GCData:DATA? <param>
Returns measurement data at all frequency and power data points for GCA SMART sweeps and 2D sweeps.

- When using SMART sweep, ALL data is returned including ALL background iteration sweeps.
- When using 2D sweeps, ALL data is returned.

Use Calc:Data? to return just the displayed data results (not the background sweeps).

A compression parameter must be present. Learn more.

The format of the data is the same as the format of the measurement that you select using Calc:Par:Select. If the measurement is scalar, than one number is returned per sweep per data point. If complex (such as Smith Chart format) than both real and imaginary numbers are returned.

If correction is on, corrected data are returned. Otherwise, raw data are returned.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<param>` (String) Parameter to read. NOT Case-sensitive. The specified parameter need NOT be displayed or selected. However, a compression parameter must be present. Learn more.

Choose from:

- "pin" - (CompIn21) Input power at the compression point.
- "pout" - (CompOut21) Output power at the compression point.
- "gain" - (CompGain21) Device gain (S21) at the compression point.
- "inputmatch" - (CompS11) Input match at the compression point.
- "DeltaGain" - (DeltaGain21) Measured Gain (watts) / Ref Gain (watts). Learn more.
- "AI1" and "AI2" - ADC measurements at the specified compression level. Learn more.

Learn more about GCA parameters.

**Examples**

```
data = CALC:GCD:DATA? "pin"
data = calculate:gcdata:data? "pout"
```

**Return Type**

Array of data

**Default**

Not Applicable
CALCulate<ch>:GCData:IMAG? <char>, <dpoint>, <param>

(Read-Only) Returns the imaginary part of the specified Gain Compression data. If correction is on, corrected data are returned. Otherwise, raw data are returned. Can be used with Smart and 2D sweeps.

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1
- **<char>** Choose from:
  - **FREQuency** - for the specified frequency data point, returns all of the measured data for each power stimulus.
  - **POWer** - for the specified power data point, returns all of the measured data for each frequency stimulus.
- **<dPoint>** Data point (FREQ or POWer) for which data is returned.
- **<param>** (String) Parameter to read. NOT Case-sensitive. The specified parameter need NOT be displayed. However, a compression parameter must be present. Learn more.
  - "pin" - (CompIn21) Input power at the compression point.
  - "pout" - (CompOut21) Output power at the compression point.
  - "gain" - (CompGain21) Device gain (S21) at the compression point.
  - "inputmatch" - (CompS11) Input match at the compression point.
  - "DeltaGain" - (DeltaGain21) Measured Gain (watts) / Ref Gain (watts). Learn more.
  - "AI1" and "AI2" - ADC measurements at the specified compression level. Learn more.

**Examples**

For the fifth frequency data point, returns 'Power Output' imaginary (phase) data from all power stimulus values. If there are 30 power sweep points, 30 values are returned.

```plaintext
data = CALC:GCD:IMAG? FREQ,5,"pout"
```

For the 30th stimulus power data point, returns 'Power Output' imaginary (phase) data from all frequency stimulus values. If there are 201 power sweep points, 201 values are returned.

```plaintext
data = calculate:gcdata:imag? power,30,"pout"
```
CALCulate<cnum>:GCData:ITERations?

(Read-only) Returns the number of iterations made in a SMART sweep. Returns number of power points for a 2D sweep.

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

Examples CALC:GCD:ITER?

Return Type Numeric

Default Not Applicable

CALCulate<ch>:GCData:REAL? <char>, <dPoint>, <param>

(Read-Only) Returns the real part of the specified Gain Compression data. If correction is on, corrected data are returned. Otherwise, raw data are returned. Can be used with Smart and 2D sweeps.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1
<char> Choose from:

- FREQuency - for the specified frequency data point, returns all of the measured data for each power stimulus.
- POWer - for the specified power data point, returns all of the measured data for each frequency stimulus.

<dPoint> Data point (FREQ or POWer) for which data is returned.

<param> (String) Parameter to read. NOT Case-sensitive. The specified parameter need NOT be displayed. However, a compression parameter must be present. Learn more.

- "pin" - (Compln21) Input power at the compression point.
- "pout" - (CompOut21) Output power at the compression point.
- "gain" - (CompGain21) Device gain (S21) at the compression point.
- "inputmatch" - (CompS11) Input match at the compression point.
- "DeltaGain" - (DeltaGain21) Measured Gain (watts) / Ref Gain (watts). Learn more.
- "AI1" and "AI2" - ADC measurements at the specified compression level. Learn more.

**Examples**

For the fifth frequency data point, returns 'Power Output' real data from all power stimulus values. If there are 30 power sweep points, 30 values are returned.

```
data = CALC:GCD:REAL? FREQ,5,"pout"
```

For the 30th stimulus power data point, returns 'Power Output' real data from all frequency stimulus values. If there are 201 power sweep points, 201 values are returned.

```
data = calculate:gcdata:real? power,30,"pout"
```

**Return Type** Array of data

**Default** Not Applicable

---

Last Modified:

- 13-May-2011  Many clarification edits
- 30-Mar-2009  Added ADC and DeltaGain params
- 24-Apr-2008  Edited for any GCA sweep
- 18-Oct-2007  MX New topic
Gain Compression Analysis Commands

Sets and reads Gain Compression Analysis controls.

<table>
<thead>
<tr>
<th>CALCulate:GCMeas:ANALysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>CWFRequency</td>
</tr>
<tr>
<td>ENABle</td>
</tr>
<tr>
<td>ISDisfreq</td>
</tr>
<tr>
<td>XAXis</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

Other Gain Compression commands

The calibration commands listed in this topic are supplemental to the Guided Cal commands.

- **CALC:CUSTom:DEFine** - creates a gain compression measurement.
- **SENS:GCSetup** - Most Gain Compression settings.
- **GC:DATA** - Gain Compression data commands
- Gain compression data can also be saved to a *.csv file. Learn how.

See Also

- [Example Programs](#)
- [Learn about Compression Analysis](#)
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

**CALCulate<cnum>:GCMeas:ANALysis:ENABle <bool>**
(Read-Write) Enables and disables a compression analysis trace.

**Parameters**

- `<cnum>` Channel number of the GCA measurement. There must be a selected measurement on that channel using `Calc:Par:Sel`. If unspecified, `<cnum>` is set to 1.
- `<bool>` **ON** (or 1) - Enable compression analysis.
  **OFF** (or 0) - Disable compression analysis.

**Examples**

```
CALC:GCM:ANAL:ENAB ON
```

```
calculate2:gcmeas:analysis:enable off
```

**Query Syntax**

`CALCulate<cnum>:GCMeas:ANALysis:ENABle?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

----

**CALCulate<cnum>:GCMeas:ANALysis:CWFRequency <num>**

(Read-Write) Set and return the CW frequency for a compression analysis trace.

**Parameters**

- `<cnum>` Channel number of the GCA measurement. There must be a selected measurement on that channel using `Calc:Par:Sel`. If unspecified, `<cnum>` is set to 1.
- `<num>` CW frequency in Hz. Choose a frequency within the range of the gain compression channel.

**Examples**

```
CALC:GCM:ANAL:CWFR 1e9
```

```
calculate2:gcmeas:analysis:cwfrequency 1e10
```

**Query Syntax**

`CALCulate<cnum>:GCMeas:ANALysis:CWFRequency?`

**Return Type**

Numeric

**Default**

Not Applicable

----

**CALCulate<cnum>:GCMeas:ANALysis:ISDisfrequency <bool>**
(Read-Write) Sets and returns whether the CW frequency for the compression analysis trace can be set to only the discrete frequencies or provides interpolation.

**Parameters**

- `<cnum>` Channel number of the GCA measurement. There must be a selected measurement on that channel using `Calc:Par:Sel`. If unspecified, `<cnum>` is set to 1.
- `<bool>` **ON** (or 1) - Discrete data points only.
  **OFF** (or 0) - Interpolated data points.

**Examples**

```
CALC:GCM:ANAL:ISD ON
calculate2:gcmeas:analysis:isdisfrequency off
```

**Query Syntax**

`CALCulate<cnum>:GCMeas:ANALysis:ISDisfrequency?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

**CALCulate<cnum>:GCMeas:ANALysis:XAXis <char>**

(Read-Write) Sets and returns the type of data to display on the x-axis of a compression analysis trace.

**Parameters**

- `<cnum>` Channel number of the GCA measurement. There must be a selected measurement on that channel using `Calc:Par:Sel`. If unspecified, `<cnum>` is set to 1.
- `<bool>` Data to display on X-axis. Choose from:

  - **PIN** - Input power to the DUT.
  - **PSOsource** - power from the source.

**Examples**

```
CALC:GCM:ANAL:XAX PIN
calculate2:gcmeas:analysis:xaxis psource
```

**Query Syntax**

`CALCulate<cnum>:GCMeas:ANALysis:XAXis?`

**Return Type**

Character

**Default**

PIN
Group Delay Aperture Commands

Controls the Aperture setting used to make Group Delay measurements.

```
CALCulate:GDELay
  FREQuency
  PERCent
  POINts
```

Click on a blue keyword to view the command details.

See Also

- Learn about Group Delay Aperture
- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

CALCulate<cnum>:GDELay:FREQuency <value>

(Read-Write) Sets group delay aperture using a fixed frequency range.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<value>` Frequency range (in Hz) to use for the aperture setting. Choose between the equivalent of two data points and the channel frequency span.

Examples

```
CALC:GDEL:FREQ 1E6
```

Query Syntax

CALCulate<cnum>:GDELay:FREQuency?

Return Type

Numeric

Default

Frequency range that equates to 11 points. This can be changed to two points with a preference setting.

CALCulate<cnum>:GDELay:PERCent <value>
(Read-Write) Sets group delay aperture using a percent of the channel frequency span.

See Critical Note

Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

- **<value>** Percent of frequency span to use for the aperture setting. Choose between the equivalent of two data points and 100 percent of the channel frequency span.

**Examples**

'set to 25 percent of the channel frequency span

```
CALC:GDEL:PERC 25
```

**Query Syntax**

CALCulate<cnun>:GDELay:PERCent?

**Return Type**

Numeric

**Default**

Percent of frequency span that equates to 11 points. This can be changed to two points with a preference setting.

---

**CALCulate<cnun>:GDELay:POINts <value>**

(Read-Write) Sets group delay aperture using a fixed number of data points.

See Critical Note

Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

- **<value>** Number of data points to use for the aperture setting. Choose between two points and the number of points in the channel.

**Examples**

'set to 25 data points

```
CALC:GDEL:POIN 25
```

**Query Syntax**

CALCulate<cnun>:GDELay:POINts?

**Return Type**

Numeric

**Default**

11 points. This can be changed to two points with a preference setting.
Calc:Limit Commands

Controls the limit segments used for pass / fail testing.

<table>
<thead>
<tr>
<th>CALCulate:LIMit:</th>
</tr>
</thead>
<tbody>
<tr>
<td>DATA</td>
</tr>
<tr>
<td>DELete</td>
</tr>
<tr>
<td>DISPlay</td>
</tr>
<tr>
<td>[STATe]</td>
</tr>
<tr>
<td>FAIL?</td>
</tr>
<tr>
<td>SEGment</td>
</tr>
<tr>
<td>AMPLitude</td>
</tr>
<tr>
<td>[STARt]</td>
</tr>
<tr>
<td>[STOP]</td>
</tr>
<tr>
<td>[SHIMulus]</td>
</tr>
<tr>
<td>[STARt]</td>
</tr>
<tr>
<td>[STOP]</td>
</tr>
<tr>
<td>[TYPE]</td>
</tr>
<tr>
<td>SOUNd</td>
</tr>
<tr>
<td>[STATe]</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

see Also

- Example Programs
- Learn about Limit Lines
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one
measurement for each channel using `CalcPar:Select`

CALCulate<cnum>:LIMit:DATA <block>

(Read-Write) Sets data for limit segments.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement for which limit lines are to be set. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<block>` Data for all limit segments in REAL, 64 format. The following is the data format for 1 segment:

  `Type,BegStim, EndStim, BegResp,EndResp`

  - **Type** Type of limit segment. Choose from
    - 0 - Off
    - 1 - Max
    - 2 - Min
  - **BegStim** Start of X-axis value (freq, power, time)
  - **EndStim** End of X-axis value
  - **BegResp** Y-axis value that corresponds with Start of X-axis value
  - **EndResp** Y-axis value that corresponds with End of X-axis value

Examples

The following writes three max limit segments for a bandpass filter.

```
CALC:LIM:DATA 1,3e5,4e9,−60,0,1,4e9,7.5e9,0,0,1,7.5e9,9e9,0,−30
```

Query Syntax

`CALCulate<cnum>:LIMit:DATA?`

Return Type

Depends on `FORM:DATA` - All 100 predefined limit segments are returned.

Default

100 limit segments - all values set to 0

---

CALCulate<cnum>:LIMit:DATA:DELeTe
(Write-only) Deletes all limit line data for the selected measurement on the specified channel.

**See Critical Note**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>Not Applicable</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**CALCulate<cnum>:LIMit:DATA:DEL**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>Not Applicable</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**CALCulate<cnum>:LIMit:DISPlay[:STATe] <ON | OFF>**

(Read-Write) Turns the display of limit segments ON or OFF (if the data trace is turned ON).

**See Critical Note**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>&lt;ON</th>
<th>OFF</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ON</td>
<td>OFF</td>
<td>ON (or 1) - turns the display of limit segments ON. OFF (or 0) - turns the display of limit segments OFF.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:LIM:DISP:STAT ON</td>
<td>ON</td>
</tr>
<tr>
<td>calculate2:limit:display:state off</td>
<td>ON</td>
</tr>
</tbody>
</table>

**CALCulate<cnum>:LIMit:FAIL?**
(Read-only) Returns the Pass / Fail status of the limit line test. Returns 1 (Fail) if any data point fails for any limit segment.

Limit display (CALC:LIM:DISP) does NOT have to be ON.

See Critical Note

Parameters

<cn> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cn> is set to 1.

Examples

CALC:LIM:FAIL?

Return Type

Boolean

- 0 is returned when Pass
- 1 is returned when Fail

Default Not Applicable

CALCulate<cn>:LIMit:SEGment<s>:AMPLitude:STARt <num>

(Read-Write) Sets the start (beginning) of the Y-axis amplitude (response) value.

See Critical Note

Parameters

<cn> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cn> is set to 1.

<s> Segment number; if unspecified, value is set to 1.

<num> Choose any number between -500 and 500

Display value is limited to the Maximum and Minimum displayed Y-axis values.

Examples

CALC:LIM:SEG1:AMPL:STAR 10
calculate2:limit:segment2:amplitude:start 10

Query Syntax

CALCulate<cn>:LIMit:SEGment<s>:AMPLitude:STARt?

Return Type

Numeric

Default 0
CALCulate<cnum>:LIMit:SEGMen<snum>:AMPLitude:STOP <num>
(Read-Write) Sets the stop (end) of the Y-axis amplitude (response) value.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
<snum> Segment number; if unspecified, value is set to 1.
<num> Choose any number between: -500 and 500

Display value is limited to the Maximum and Minimum displayed Y-axis values.

Examples
CALC:LIM:SEG1:AMPL:STOP 10
calculate2:limit:segment2:amplitude:stop 10

Query Syntax
CALCulate<cnum>:LIMit:SEGMen<snum>AMPLitude:STOP?

Return Type Numeric

Default 0

CALCulate<cnum>:LIMit:SEGMen<snum>:STIMulus:STARt <num>
(Read-Write) Sets the start (beginning) of the X-axis stimulus value.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
<snum> Segment number; if unspecified, value is set to 1.
<num> Choose any number within the X-axis span of the analyzer.

Examples
CALC:LIM:SEM1:STIM:STAR 10
calculate2:limit:segment2:stimulus:start 10

Query Syntax
CALCulate<cnum>:LIMit:SEGMen<snum>STIMulus:STARt?

Return Type Numeric

Default 0

CALCulate<cnum>:LIMit:SEGMen<snum>:STIMulus:STOP <num>
(Read-Write) Sets the stop (end) of the X-axis stimulus value.

See Critical Note

Parameters

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<snum>`: Segment number; if unspecified, value is set to 1.
- `<num>`: Choose any number within the X-axis span of the analyzer.

Examples:

```
CALC:LIM:SEGM1:AMPL:STOP 10
CALC:limit:segment2:stimulus:stop 10
```

Query Syntax:
```
CALCulate<cnum>:LIMit:SEGment<snum>:STIMulus:STOP?
```

Return Type: Numeric

Default: 0

CALCulate<cnum>:LIMit:SEGment<snum>:TYPE <char>

(Read-Write) Sets the type of limit segment.

See Critical Note

Parameters

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<snum>`: Segment number. Choose any number between: 1 and 100
  If unspecified, value is set to 1.
- `<char>`: Choose from:
  - `LMAX`: a MAX limit segment. Any response data exceeding the MAX value will fail.
  - `LMIN`: a MIN limit segment. Any response data below the MIN value will fail.
  - `OFF`: the limit segment (display and testing) is turned OFF.

Examples:

```
CALC:LIM:SEGM:TYPE LMIN
CALC:limit:segment3:type lmax
```

Query Syntax:
```
CALCulate<cnum>:LIMit:SEGment<snum>:TYPE?
```
CALCulate<cnum>:LIMit:SOUNd[:STATe] <ON | OFF>

(Read-Write) Turns limit testing fail sound ON or OFF.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<ON | OFF>`
  - ON (or 1) - turns sound ON.
  - OFF (or 0) - turns sound OFF.

Examples

```
CALC:LIM:SOUN ON
calculate2:limit:sound:state off
```

Query Syntax

CALCulate<cnum>:LIMit:SOUNd[:STATe]?

Return Type

- Boolean (1 = ON, 0 = OFF)
- Default: OFF

CALCulate<cnum>:LIMit[:STATe] <ON | OFF>

(Read-Write) Turns limit segment testing ON or OFF.

- Use CALC:LIM:DISP to turn ON and OFF the display of limit segments.
- If using Global Pass/Fail status, trigger the PNA AFTER turning Limit testing ON.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<ON | OFF>`
  - ON (or 1) - turns limit testing ON.
  - OFF (or 0) - turns limit testing OFF.

Examples

```
CALC:LIM:STAT ON
calculate2:limit:state off
```

Query Syntax

CALCulate<cnum>:LIMit[:STATe]?
**Return Type**  Boolean (1 = ON, 0 = OFF)

**Default**  OFF

Last Modified:

- 10-Jan-2011   Minor edits
- 31-Oct-2008   Added Delete and Fail commands (8.33)
## Calculate:Marker Commands

Controls the marker settings used to remotely output specific data to the computer.

<table>
<thead>
<tr>
<th>CALCulate:MARKer:</th>
</tr>
</thead>
<tbody>
<tr>
<td>AOFF</td>
</tr>
<tr>
<td>BWIDth</td>
</tr>
<tr>
<td>COMPression</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>COUPling</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>DELTa</td>
</tr>
<tr>
<td>DISCrete</td>
</tr>
<tr>
<td>DISTance</td>
</tr>
<tr>
<td>FORMat</td>
</tr>
<tr>
<td>FUNCtion</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>PNOP more commands</td>
</tr>
</tbody>
</table>
PSATuration more commands

REFerence

| [STATe] |
| X |
| Y? |

SET

[STATe]

TARGet

TYPE

X

Y?

---

Click on a blue keyword to view the command details.

See Also

- Marker example program
- Marker Readout number and size commands.
- Learn about Markers
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

Note: The Reference Marker is Marker Number 10

---

CALCulate<cnum>:MARKer:AOFF
(Write-only) Turns all markers off for selected measurement.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.

Examples

- `CALC:MARK:AOFF`
- `calculate2:marker:aoff`

Query Syntax Not applicable

Default Not applicable

CALCulate<cnum>:MARKer:BWIDth <num>

(Read-Write) Turns on and sets markers 1 through 4 to calculate filter bandwidth. The `<num>` parameter sets the value below the maximum bandwidth peak that establishes the bandwidth of a filter. For example, if you want to determine the filter bandwidth 3 db below the bandpass peak value, set `<num>` to -3.

To turn off the Bandwidth markers, either turn them off individually or turn them All Off.

The analyzer screen will show either Bandwidth statistics OR Trace statistics; not both.

To search a User Range with the bandwidth search, first activate marker 1 and set the desired User Range. Then send the CALC:MARK:BWID command. The user range used with bandwidth search only applies to marker 1 searching for the max value. The other markers may fall outside the user range.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` Target value below filter peak. Choose any number between : -500 and 500

Examples

- `CALC:MARK:BWID -3`
- `calculate2:marker:bwidth -2.513`

Query Syntax

- `CALCulate<cnum>:MARKer:BWIDth?` Returns the results of bandwidth search:

  Return Type Numeric - Four Character values separated by commas: bandwidth, center Frequency, Q, loss.
**CALCulate\(<\text{cnum}\>):MARKer\(<\text{mkr}\>\):COMPression:LEVel \(<\text{num}\>**

(Read-Write) Set and read the marker compression level. A compression marker must already exist. Use `CALC:MARK ON` and `CALC:MARK:FUNC COMP` to create compression markers.

**See Critical Note**

**Parameters**

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>`: Any existing marker number from 1 to 10; if unspecified, value is set to 1.
- `<num>`: Compression level. Choose any number between: -500 dB to 500 dB

Standard gain compression values are positive.

**Examples**

```
CALC:MARK:COMP:LEV 1
calculate2:marker:compression:level 1.5
```

**Query Syntax**

`CALCulate\(<\text{cnum}\>):MARKer\(<\text{mkr}\>\):COMPression:LEVel?`

**Return Type**

Numeric

**Default**

+1

---

**CALCulate\(<\text{cnum}\>):MARKer\(<\text{mkr}\>\):COMPression:PIN?**

(Read-only) Reads the input power at the marker compression level. First send `CALC:MARK:FUNC:EXEC COMP` or `CALC:MARK:FUNC:TRAC ON`.

**See Critical Note**

**Parameters**

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>`: Any existing marker number from 1 to 10; if unspecified, value is set to 1.

**Examples**

```
CALC:MARK:COMP:PIN?
calculate2:marker:compression:pin?
```

**Return Type**

Numeric

**Default**

Not applicable
CALCulate<cnm>:MARKer<mrk>:COMPression:POUT?

(Read-only) Reads the output power at the marker compression level. First send CALC:MARK:FUNC:EXEC COMP or CALC:MARK:FUNC:TRAC ON

See Critical Note

Parameters

<cnm> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnm> is set to 1.

<mrk> Any existing marker number from 1 to 10; if unspecified, value is set to 1.

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>calculate2:marker:compression:pout?</td>
<td></td>
</tr>
</tbody>
</table>

Return Type Numeric

Default Not applicable

CALCulate<cnm>:MARKer<mrk>:COUPling[:STATe]<ON|OFF>

(Read-Write) Sets and reads the state of Coupled Markers (ON and OFF).

See Critical Note

Parameters

<cnm> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnm> is set to 1.

<mrk> Any existing marker number from 1 to 10; if unspecified, value is set to 1.

<ON|OFF> OFF (0) - Turns Coupled Markers OFF. ON (1) - Turns Coupled Markers ON

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:MARK:COUP ON</td>
<td>For measurement calculation.</td>
</tr>
<tr>
<td>calculate2:marker8:coupling off</td>
<td></td>
</tr>
</tbody>
</table>

Query Syntax CALCulate<cnm>:MARKer<mrk>:COUPling[:STATe]<ON|OFF>

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF

CALCulate<cnm>:MARKer<mrk>:DELTa <ON|OFF>
**CALCulate<cnum>:MARKer<mkr>:DELTa <ON|OFF>**

*(Read-Write)* Specifies whether marker is relative to the Reference marker or absolute.

**Note:** The reference marker must already be turned ON with CALC:MARK:REF:STATE.

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<mkr>** Any existing marker number from 1 to 10; if unspecified, value is set to 1.
- **<ON|OFF>**
  - ON (or 1) - Specified marker is a Delta marker
  - OFF (or 0) - Specified marker is an ABSOLUTE marker

**Examples**

```
CALC:MARK:DELT ON
calculate2:marker8:delta off
```

**Query Syntax**

CALCulate<cnum>:MARKer<mkr>:DELTa?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

---

**CALCulate<cnum>:MARKer<mkr>:DISCrete <ON|OFF>**

*(Read-Write)* Makes the specified marker display either a calculated value between data points (interpolated data) or the actual data points (discrete data).

**See Critical Note**

**Parameters**

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<mkr>** Any existing marker number from 1 to 10; if unspecified, value is set to 1.
- **<ON|OFF>**
  - ON (or 1) - Specified marker displays the actual data points
  - OFF (or 0) - Specified marker displays calculated data between the actual data points.

**Examples**

```
CALC:MARK:DISC ON
calculate2:marker8:discrete off
```

**Query Syntax**

CALCulate<cnum>:MARKer<mkr>:DISCrete?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF
CALCulate<cnum>:MARKer<mkr>:DISTance <num>

(Read-Write) Set or query marker distance on a time domain trace.

The Write command moves the marker to the specified distance value. Once moved, you can read the Y axis value or read the X-axis time value. (Distance is calculated from the X-axis time value.)

The Read command reads the distance of the marker.

If the marker is set as delta, the WRITE and READ data is relative to the reference marker.

See Critical Note

Parameters

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>`: Any existing marker number from 1 to 10; if unspecified, value is set to 1.
- `<num>`: Marker distance in the unit of measure specified with `CALC:TRAN:TIME:MARK:UNIT`

Examples

```
CALC:MARK:DIST .1
calculate2:marker8:distance 5
```

Query Syntax

`CALCulate<cnum>:MARKer<mkr>:DISTance?`

Return Type

Numeric

Default

Not Applicable

---

CALCulate<cnum>:MARKer<mkr>:FORMAT <char>

(Read-Write) Sets the format of the data that will be returned in a marker data query `CALC:MARK:Y?` and the displayed value of the marker readout. The selection does not have to be the same as the measurement's display format.

See Critical Note

Parameters

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>`: Any marker number from 1 to 10; if unspecified, value is set to 1
- `<char>`: Choose from:
  - `DEFAULT` - The format of the selected measurement
| **MLINear** | Linear magnitude |
| **MLOGarithmic** | Logarithmic magnitude |
| **IMPedance** | (R+jX) |
| **ADMittance** | (G+jB) |
| **PHASE** | Phase |
| **IMAGinary** | Imaginary part (Im) |
| **REAL** | Real part (Re) |
| **POLar** | (Re, Im) |
| **GDELay** | Group Delay |
| **LINPhase** | Linear Magnitude and Phase |
| **LOGPhase** | Log Magnitude and Phase |
| **KELVin** | temperature |
| **FAHRENheit** | temperature |
| **CELSius** | temperature |

**Examples**

```
CALC:MARK:FORMat MLIN
calculate2:marker8:format Character
```

**Query Syntax**

```
CALCulate<cnum>:MARKer<mkr>:FORMat?
```

**Return Type**

Character

**Default**

DEFault

**Calculates**

```
CALCulate<cnum>:MARKer<mkr>:FUNCTION:APEak:EXCursion <num>
```
(Read-Write) Sets amplitude peak excursion for the specified marker. The Excursion value determines what is considered a "peak". This command applies to marker peak searches (Next peak, Peak Right, Peak Left).

See Critical Note

Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td>&lt;mkr&gt;</td>
<td>Any existing marker number from 1 to 10; if unspecified, value is set to 1.</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>Excursion value. Choose any number between -500 and 500.</td>
</tr>
</tbody>
</table>

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

```
cALC:MARK:FUNC:APE:EXC 10
```

Query Syntax

```
CALCulate<cnum>:MARKer<mkr>:FUNCtion:APEak:EXCursion?
```

Return Type

Numeric

Default

3

CALCulate<cnum>:MARKer<mkr>:FUNCtion:APEak:THReshold <num>

(Read-Write) Sets peak threshold for the specified marker. If a peak (using the criteria set with :EXCursion) is below this reference value, it will not be considered when searching for peaks. This command applies to marker peak searches (Next peak, Peak Right, Peak Left).

See Critical Note

Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td>&lt;mkr&gt;</td>
<td>Any marker number from 1 to 10; if unspecified, value is set to 1</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>Threshold value. Choose any number between -500 and 500.</td>
</tr>
</tbody>
</table>

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

```
CALC:MARK:FUNC:APE:THR -40
```

```
calculate2:marker8:function:apeak:threshold -55
```
Query Syntax  | CALCulate<cnum>:MARKer<mkr>:FUNCtion:APEak:THreshold?
Return Type   | Numeric
Default       | -100

**CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER[:RANGe] <range>**

*(Read-Write)* Assigns the specified marker to a range number. The x-axis travel of the marker is constrained to the range's span. The span is specified with the **CALC:MARK:FUNC:DOM:USER:START** and **STOP** commands, unless range 0 is specified which is the full span of the analyzer.

Each channel has 16 user ranges. (Trace statistics use the same ranges.) More than one marker can use a domain range.

*See Critical Note*

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1
- `<span>` User span. Choose any Integer from 0 to 16
  - 0 is Full Span of the analyzer
  - 1 to 16 are available for user-defined x-axis span

**Examples**

```
CALC:MARK:FUNC:DOM:USER 1
calculate2:marker8:function:domain:USER:range 1
```

**Query Syntax**  | CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER[:RANGe]?

Returns the user span number that the specified marker is assigned to.

**Return Type**  | Numeric
**Default**      | 0 - Full Span

**CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER:STARt <start>**
(Read-Write) Sets the start of the span that the specified marker's x-axis span will be constrained to.

Use `CALC:MARK:FUNC:DOM:USER<range>` to set range number


**Note:** If the marker is assigned to range 0 (full span), the USER:STARt and STOP commands generate an error. You cannot set the STARt and STOP values for "Full Span".

**Note:** This command does the same as `CALC:FUNC:DOM:USER:STAR`

See Critical Note

### Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1
- `<start>` The analyzer's Minimum x-axis value

### Examples

```
```

### Query Syntax

`CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER:STARt?`

### Return Type

Numeric

**Default** The analyzer's Minimum x-axis value

---

`CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER:STOP <stop>`

(Read-Write) Sets the stop of the span that the marker's x-axis travel will be constrained to.

Use `CALC:MARK:FUNC:DOM:USER<range>` to set range number

Use `CALC:MARK:FUNC:DOM:USER:STARt` to set the stop value.

**Note:** If the marker is assigned to range 0 (full span), the USER:STARt and STOP commands generate an error. You cannot set the STARt and STOP values for "Full Span".

**Note:** This command does the same as `CALC:FUNC:DOM:USER:STOP`

See Critical Note

### Parameters
Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, \(<\text{cnum}>\) is set to 1.

Any marker number from 1 to 10; if unspecified, value is set to 1.

Stop value of x-axis span; Choose any number between the analyzer's MINimum and MAXimum x-axis value.

Examples

```
`calcule2:marker8:function:domain1:USER:stop 1e12
```

Query Syntax

```
CALCulate<\text{cnum}>:MARKer<\text{mkr}}:FUNCtion:DOMain:USER:STOP?
```

Return Type

Numeric

**Default**

The analyzer's MAXimum x-axis value.

---

CALCulate<\text{cnum}>:MARKer<\text{mkr}>:FUNCtion:EXECute <\text{func}>

(Write-only) Immediately executes (performs) the specified search function.

Learn more about Marker Search

See Critical Note

Parameters

- \(<\text{cnum}>\): Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, \(<\text{cnum}>\) is set to 1.
- \(<\text{mkr}>\): Any marker number from 1 to 10; if unspecified, value is set to 1.
- \(<\text{func}>\): The function to be performed. Choose from:
  - MAXimum - finds the highest value
  - MINimum - finds the lowest value
  - RPEak - finds the next valid peak to the right
  - LPEak - finds the next valid peak to the left
  - NPEak - finds the next highest value among the valid peaks
  - TARGet - finds the target value to the right, wraps around to the left
  - LTARget - finds the next target value to the left of the marker
  - RTARget - finds the next target value to the right of the marker
- **COMPression** - finds the compression level on a Power Swept S21 trace.

**Examples**

```
CALC:MARK:FUNC:EXEC MAX
calculate2:marker2:function:execute maximum
```

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**CALCulate<cnum>:MARKer<mkr>:FUNCTION[:SELect] <char>**

(Read-Write) Sets the search function that the specified marker will perform when executed. Use **CALC:MARK:FUNC:TRAC ON** to automatically execute the search every sweep.

Learn more about Marker Search

See Critical Note

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1.
- `<char>` Marker function. Choose from:
  - **MAXimum** - finds the highest value
  - **MINimum** - finds the lowest value
  - **RPEak** - finds the next valid peak to the right
  - **LPEak** - finds the next valid peak to the left
  - **NPEak** - finds the next highest value among the valid peaks
  - **TARGet** - finds the target value to the right, wraps around to the left
  - **LTARget** - finds the next target value to the left of the marker
  - **RTARget** - finds the next target value to the right of the marker
  - **COMPression** - finds the compression level on a power-swept S21 trace.

**Examples**

```
CALC:MARK:FUNC MAX
calculate2:marker8:function:select ltarget
```

**Query Syntax**

CALCulate<cnum>:MARKer<mkr>:FUNCTION[:SELect]?

**Return Type**

Character
CALCulate<cnum>:MARKer<mkr>:TARGet[:VALue] <num>

(Read-Write) Sets the target value for the specified marker when doing Target Searches with CALC:MARK:FUNC:SEL <TARGet | RTARget | LTARget>

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1.
- `<num>` Target value to search for; Units are NOT allowed.

Examples

```
CALC:MARK:TARG 2.5
calculate2:marker8:target:value -10.3
```

Query Syntax

CALCulate<cnum>:MARKer<mkr>:TARGet[:VALue]?

Return Type

Numeric

Default

0

CALCulate<cnum>:MARKer<mkr>:FUNCtion:TRACKing <ON | OFF>

(Read-Write) Sets the tracking capability for the specified marker. The tracking function finds the selected search function every sweep. In effect, turning Tracking ON is the same as doing a CALC:MARK:FUNC:EXECute command every sweep.

Learn more about Marker Search

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1.
- `<ON | OFF>` ON (or 1) - The specified marker will "Track" (find) the selected function every sweep.
  
  OFF (or 0) - The specified marker will find the selected function only when
the CALC:MARK:FUNC:EXECute command is sent.

**Examples**
```
CALC:MARK:FUNC:TRAC ON
calculate2:marker8:function:tracking off
```

**Query Syntax**
```
CALCulate<cnum>:MARKer<mkr>:FUNCtion:TRACking?
```

**Return Type**
Boolean (1 = ON, 0 = OFF)

**Default**
OFF

---

**CALCulate<cnum>:MARKer:REFerence[:STATE] <ON | OFF>**

(Read-Write) Turns the reference marker (marker 10) ON or OFF. When turned OFF, existing Delta markers revert to absolute markers.

**See Critical Note**

**Parameters**

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<ON | OFF>`:  
  - **ON** (or 1) - turns reference marker ON
  - **OFF** (or 0) - turns reference marker ON

**Examples**
```
CALC:MARK:REF ON
calculate2:marker:reference:state OFF
```

**Query Syntax**
```
CALCulate<cnum>:MARKer:REFerence[:STATE]?
```

**Return Type**
Boolean (1 = ON, 0 = OFF)

**Default**
OFF

---

**CALCulate<cnum>:MARKer:REFerence:X <num>**
(Read-Write) Sets and returns the absolute x-axis value of the reference marker (marker 10).

See Critical Note

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>X-axis value. Choose any number within the operating domain of the reference marker.</td>
</tr>
</tbody>
</table>

Examples

```
CALC:MARK:REF:X 1e9
```

```
calculate2:marker:reference:x 1e6
```

Query Syntax

CALCulate<cnum>:MARKer:REFerence:X?

Return Type

Numeric

Default

If the first Marker, turns ON in the middle of the X-axis span. If not, turns ON at the position of the active marker.

---

CALCulate<cnum>:MARKer:REFerence:Y?

(Read-only) Returns the absolute Y-axis value of the reference marker.

See Critical Note

**Parameters**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

Examples

```
CALC:MARK:REF:Y?
```

```
calculate2:marker:reference:y?
```

Return Type

Character

Default

Not applicable

---

CALCulate<cnum>:MARKer<mkr>:SET <char>
(Write-only) Sets the selected instrument setting to assume the value of the specified marker.

Marker Functions CENT, SPAN, STARt, and STOP do not work with channels that are in CW or Segment Sweep mode.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1
- `<char>` Choose from:
  - CENTer - changes center frequency to the value of the marker
  - SPAN - changes the sweep span to the span that is defined by the delta marker and the marker that it references. Unavailable if there is no delta marker.
  - START - changes the start frequency to the value of the marker
  - STOP - changes the stop frequency to the value of the marker
  - RLEVel - changes the reference level to the value of the marker
  - DELay - changes the line length at the receiver input to the phase slope at the active marker stimulus position.
  - CWFReq - Sets the CW frequency to the frequency of the active marker. Does NOT change sweep type. NOT available in CW or Power Sweep. Use this argument to first set the CW Frequency to a value that is known to be within the current calibrated range, THEN set Sweep:Type to POWer or CW.

Examples:

```plaintext
CALC:MARK:SET CENT
calculate2:marker8:set span
```

Query Syntax

Not Applicable

Default

Not Applicable

```
CALCulate<cnum>:MARKe<clk>:[:STATE] <ON|OFF>
```
(Read-Write) Turns the specified marker ON or OFF. **Marker 10 is the Reference Marker.** To turn all markers off, use CALC:MARK:AOFF.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1.
- `<ON|OFF>` **ON** (or 1) - turns marker ON.  
  **OFF** (or 0) - turns marker OFF.

**Examples**

```
CALC:MARK ON
calculate2:marker8 on
```

**Query Syntax**

CALCulate<cnum>:MARKer<mkr>:STATe?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

Off

---

**CALCulate<cnum>:MARKer<mkr>:TYPE <char>**

(Read-Write) Sets the type of the specified marker.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1
- `<char>` Choose from:
  - **NORMal** - a marker that stays on the assigned X-axis position unless moved or searching.
  - **FIXed** - a marker that will not leave the assigned X or current Y-axis position.

**Examples**

```
CALC:MARK:TYPE NORM
calculate2:marker2:type fixed
```

**Query Syntax**

CALCulate<cnum>:MARKer<mkr>:TYPE?

**Return Type**

Character

**Default**

NORMal
**CALCulate<cnum>:MARKer<mkr>:X <num>**

*(Read-Write)* Sets the marker's X-axis value (frequency, power, or time). If the marker is set as delta, the SET and QUERY data is relative to the reference marker.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<mkr>` Any marker number from 1 to 10; if unspecified, value is set to 1.
- `<num>` Any X-axis position within the measurement span of the marker.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
calc:mark:X 100Mhz
calculate2:marker8:x maximum
```

**Query Syntax**

`CALCulate<cnum>:MARKer<mkr>:X?`

**Return Type** Numeric

**Default** First Marker turns ON in the middle of the X-axis span. Subsequent markers turn ON at the position of the active marker.

**CALCulate<cnum>:MARKer<mkr>:Y?**

*(Read-only)* Reads the marker's Y-axis value. The format of the value depends on the current CALC:MARKER:FORMAT setting. If the marker is set as delta, the data is relative to the reference marker. The query always returns two numbers:

- Smith and Polar formats - (Real, Imaginary)
- LINPhase and LOGPhase - (Real, Imaginary)
- All other formats - (Value,0)

**Note:** To accurately read the marker Y-axis value with trace smoothing applied, the requested format must match the displayed format. Otherwise, the returned value is un-smoothed data. For example, to read the smoothed marker value when measuring group delay, both the display format and the marker format must be set to (Group) Delay.

**See Critical Note**
Parameters

<cnum>  Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<mkr>   Any marker number from 1 to 10; if unspecified, value is set to 1.

Examples

CALC:MARK:Y?
calculate2:marker3:y?

Query Syntax

CALCulate<cnum>:MARKer<mkr>:Y?

Return Type

Numeric

Default

Not applicable

Last modified:

10-Jan-2011   Minor edits
9-Mar-2010    Added links to PNOP and PSAT
24-Feb-2009   Replace True/False
12-Feb-2009   Added compression markers (8.50)
2-Oct-2008    Added Calc:Mark:Set CWFR (8.33)
1-Oct-2007    Added temperature formats
27-Mar-2007   Corrected Set?
4-Dec-2006    Added smoothing note to Y?
Calculate: Math Commands

Controls math operations on the currently selected measurement and memory.

CALCulate: MATH

MEMorize  FUNCTION

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Math Operations
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

CALCulate<cnal>:MATH:FUNCTION <char>

(Read-Write) Sets math operations on the currently selected measurement and the trace stored in memory. (There MUST be a trace stored in Memory. See CALC:MATH MEM)

See Critical Note

Parameters

<table>
<thead>
<tr>
<th>&lt;cnal&gt;</th>
<th>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnal&gt; is set to 1.</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;char&gt;</td>
<td>The math operation to be applied. Choose from the following:</td>
</tr>
<tr>
<td></td>
<td>Trace data only</td>
</tr>
<tr>
<td>NORMAL</td>
<td>Data + Memory</td>
</tr>
<tr>
<td>ADD</td>
<td>Data - Memory</td>
</tr>
<tr>
<td>SUBTract</td>
<td>Data * Memory</td>
</tr>
<tr>
<td>MULTiply</td>
<td>Data / Memory</td>
</tr>
<tr>
<td>DIVide</td>
<td></td>
</tr>
</tbody>
</table>
### Examples

<table>
<thead>
<tr>
<th>Example</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:MATH:FUNC NORM</code></td>
<td>Calculation of a mathematical function in normal mode.</td>
</tr>
<tr>
<td><code>calculate2:math:function subtract</code></td>
<td>Calculation of a subtraction function.</td>
</tr>
</tbody>
</table>

### Query Syntax

CALCulate<nnum>:MATH:FUNCtion?

### Return Type

Character

### Default

NORMal

---

**CALCulate<nnum>:MATH:MEMorize**

(Write-only) Puts the currently selected measurement trace into memory. (Data-> Memory).

**See Critical Note**

### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>&lt;nnum&gt;</code></td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <code>&lt;nnum&gt;</code> is set to 1.</td>
</tr>
</tbody>
</table>

### Examples

<table>
<thead>
<tr>
<th>Example</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:MATH:MEM</code></td>
<td>Storage of a measurement trace.</td>
</tr>
<tr>
<td><code>calculate2:math:memorize</code></td>
<td>Memorization of a measurement.</td>
</tr>
</tbody>
</table>

### Query Syntax

Not applicable

### Default

Not applicable
**CALCulate<ch>:MIXer:XAXis <char>**

(Read-Write) Sets or returns the swept parameter to display on the X-axis for the selected FCA and GCX measurement.

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<char>` Parameter to display on the X-axis. Choose from:
  - **INPUT** - Input frequency span
  - **OUTPUT** - Output frequency span
  - **LO_1** - First LO frequency span
  - **LO_2** - Second LO frequency span

**Examples**

```
CALC:MIX:XAX INPUT
calc2:mixer:xaxis output
```

See an example that creates, selects, and calibrates an SMC and VMC measurement using SCPI.

**Query Syntax**

CALCulate<ch>:MIXer:XAXis?

**Return Type**

Character

**Default**

OUTPUT

---

Last Modified:

9-Sep-2010   Edit for GCX
**Calculate:Normalize Commands**

Specifies the normalization features used for a receiver power calibration.

These commands are **Superseded** (Sept 2004).

See the replacement commands in a new [Receiver Power Cal example](#).

```
CALCulate:NORMalize

[IMMediate]  STATe  INTerpolation

[STATe]
```

Click on a blue keyword to view the command details.

See Also

- [Example Programs](#)
- [Learn about Receiver Cal](#)
- [SCPI Command Tree](#)

Save and recall your receiver power calibration (which use .CST file commands):

- `SENS:CORR:CSET:SAVE`
- `SENS:CORR:CSET[:SEL]`

Or use these two commands and specify either .STA or .CST file extensions:

- `MMEM:LOAD`
- `MMEM:STOR`

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using `Calc:Par:Select`

```
CALCulate<cnum>:NORMalize[:IMMediate]  Superseded
```
Note: This command is replaced with SENS:CORR COLL:METH RPOWER and SENS:CORR COLL[:ACQ] POWer

See an example of a Receiver Power Calibration.

(Write only) Stores the selected measurement’s data to that measurement’s “divisor” buffer for use by the Normalization data processing algorithm. This command is not compatible with ratioed measurements such as S-parameters. It is intended for receiver power calibration when the selected measurement is of an unratioed power type.

See Critical Note

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

```
CALC:NORM
calculate1:normalize:immediate
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**CALCulate<cnum>:NORMalize:STATe <ON | OFF> Superseded**

Note: This command is replaced with SENS:CORR[:STATe] ON|OFF

(Read-Write) Specifies whether or not normalization is applied to the measurement. Normalization is enabled only for measurements of unratioed power where it serves as a receiver power calibration.

See Critical Note

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td>&lt;ON</td>
<td>OFF&gt;</td>
</tr>
<tr>
<td></td>
<td>OFF (or 0) – normalization is NOT applied to the measurement.</td>
</tr>
</tbody>
</table>

**Examples**

```
CALC:NORM:STAT ON
calculate2:normalize:state off
```

**Query Syntax**

CALCulate<cnum>:NORMalize:STATe?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF
(Read-Write) Turns normalization interpolation ON or OFF. Normalization is enabled only for measurements of unratioed power, where it serves as a receiver power calibration.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<ON | OFF>` **ON (or 1)** – turns interpolation ON.
  
  **OFF (or 0)** – turns interpolation OFF.

Examples

- `CALC:NORM:INT ON`
- `calculate2:normalize:interpolate:state off`

Query Syntax

- `CALCulate<cnum>:NORMalize:INTerpolate[:STATe]?`

Return Type

- Boolean (1 = ON, 0 = OFF)
- **Default** ON
Calculate:Offset Commands

Allows the data trace magnitude and phase to be offset.

CALCulate:OFFSet

MAGNitude PHASE

SLOPe

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Magnitude Offset
- Learn about Phase Offset
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

CALCulate<cnum>::OFFSet:MAGNitude <num>
(Read-Write) Offsets the data trace magnitude by the specified value.
To offset the data trace magnitude to a slope value that changes with frequency, use
**CALC:OFFS:MAGN:SLOP**

See Critical Note

**Parameters**

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>`: Offset value in dB.

**Examples**

```
CALC:OFFS:MAGN:4
calculate1:offset:magnitude -2
```

**Query Syntax**

`CALCulate<cnum>:OFFSet:MAGNitude?`

**Return Type**

Numeric

**Default**

0

---

**CALCulate<cnum>:OFFSet:MAGNitude:SLOPe <num>**

(Read-Write) Offsets the data trace magnitude to a value that changes linearly with frequency. The offset slope begins at 0 Hz.

See Critical Note

**Parameters**

- `<cnum>`: Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>`: Offset slope value in dB/1GHz.

**Examples**

```
CALC:OFFS:MAGN:SLOP 1 'Offset slope set to 1dB/GHz
calculate1:offset:magnitude:slope -2 'Offset slope set to -2dB/GHz
```

**Query Syntax**

`CALCulate<cnum>:OFFSet:MAGNitude:SLOPe?`

**Return Type**

Numeric

**Default**

0

---

**CALCulate<cnum>:OFFSet:PHASe <num>[<char>]**
(Read-Write) Sets the phase offset for the selected measurement.

See Critical Note

Parameters

<cnun> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnun> is set to 1.

<num> Offset phase value. Choose any number between: -360 and 360

<char> Units for phase. OPTIONAL. Choose either:
        DEG - Degrees (default)
        RAD - Radians

Examples

CALC:OFFS:PHAS 10
calculate:offset:phase 20rad

Query Syntax

CALCulate:OFFSet:PHASe?

Return Type

Numeric, returned value always in degrees

Default

0 degrees
Calculate:Parameter Commands

Lists, creates, selects, and deletes measurements.

```
CALCulate:PARameter:
   CATalog
      | EXTended
   DEFine
      | EXTended
   DELete
      | ALL
   MNUMber
      | [SElect]
   MODify
      | EXTended
   SELect
   TNUMber?
   WNUMber?
```

Click on a blue keyword to view the command details.

Red commands are superseded.

See Also

- Example Programs
- Learn about Measurement Parameters
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

```
CALCulate<cnum>:PARameter:CATalog? Superseded
```
Note: This command is replaced with **CALC:PAR:CAT:EXTended?** which lists parameters with "_" instead of ",," allowing the list to be parsed easily. This command will continue to work.

*(Read-only)* Returns the names and parameters of existing measurements for the specified channel.

**Note:** For Balanced Measurements: **CALC:PAR:CAT?** may have an unexpected behavior. Learn more.

See Critical Note

**Parameters**

- **<cnum>** Channel number of the measurements to be listed. If unspecified, <cnum> is set to 1.

**Examples**

```
CALC:PAR:CAT?
calculate2:parameter:catalog?
```

**Return Type** String - "<measurement name>,<parameter>,[<measurement name>,<parameter>...]"

**Default** "CH1_S11_1,S11"

---

**CALCulate<cnum>:PARameter:CATalog:EXTended?**

*(Read-only)* Returns the names and parameters of existing measurements for the specified channel. This command lists receiver parameters with "_" such that R1,1 is reported as R1_1. This makes the returned string a true "comma-delimited" list all the time.

See Critical Note

**Parameters**

- **<cnum>** Channel number of the measurements to be listed. If unspecified, <cnum> is set to 1.

**Examples**

```
CALC:PAR:CAT:EXT?
calculate2:parameter:catalog:extended?
```

**Return Type** String - "<measurement name>,<parameter>,[<measurement name>,<parameter>...]"

**Default** "CH1_S11_1,S11"

---

**CALCulate<cnum>:PARameter[:DEFine] <Mname>,<param>[,.port]** Superseded

**Note:** This command is replaced with **CALC:PAR:DEFine:EXTended**. This command will continue to work for up to 4 port parameters.
(Write-only) Creates a measurement but does NOT display it.

There is no limit to the number of measurements that can be created. However, there is a limit to the number of measurements that can be displayed. See Traces, Channels, and Windows on the PNA.

- Use `DISP:WIND:STATe` to create a window if it doesn't already exist.
- Use `DISP:WIND<wnum>:TRAC<tnum>:FEED <Mname>` to display the measurement.

For FCA Measurements see `CALC:CUST:DEF`

You must select the measurement (CALC<cnump>:PAR:SEL <mname>) before making additional settings.

See Critical Note

**Parameters**

- `<cnum>` Channel number of the new measurement. If unspecified, value is set to 1.
- `<Mname>` Name of the measurement. Any non-empty, unique string, enclosed in quotes.
- `<param>` For S-parameters:
  
  Any S-parameter available in the PNA

  For ratioed measurements:
  
  Any two receivers that are available in the PNA. (See the block diagram showing the receivers in YOUR PNA.)

  For example: AR1 (this means A/R1)

  For non-ratioed measurements:
  
  Any receiver that is available in the PNA. (See the block diagram showing the receivers in YOUR PNA.)

  For example: A

  For Balanced Measurements:
  
  First create an S-parameter measurement, then change the measurement using `CALC:FSIM:BAL` commands. See an example.

  For Applications see `CALC:CUST:DEF`.

  [port] Optional argument;
For multi-port reflection S-parameter measurements: specifies the PNA port which will provide the load for the calibration. This argument is ignored if a transmission S-parameter is specified.

For all non S-parameter measurements: specifies the source port for the measurement.

**Examples**

```
CALC4:PAR 'ch4_S33',S33,2 'Defines an S33 measurement with a load on port2 of the analyzer.'
```

```
calculate2:parameter:define 'ch1_a', a, 1 'unratioed meas'
calculate2:parameter:define 'ch1_a', ar1,1 'ratioed meas'
```

**Query Syntax**

Not Applicable; see Calc:Par:Cat?

**Default**

Not Applicable

---

**CALCulate<cnum>:PARameter[:DEFine]:EXTended <Mname>,<param>**

**Note:** This command replaces CALC:PAR:DEF as it allows the creating of measurements using external multiport testsets.

*(Write-only)* Creates a measurement but does NOT display it.

There is no limit to the number of measurements that can be created. However, there is a limit to the number of measurements that can be displayed. See Traces, Channels, and Windows on the PNA.

- Use DISP:WIND:STATe to create a window if it doesn't already exist.
- Use DISP:WIND<wnum>:TRAC<tnum>:FEED <Mname> to display the measurement.

You must select the measurement (CALC<cnum>:PAR:SEL <mname>) before making additional settings.

**See Critical Note**

**Parameters**

- `<cnum>` Channel number of the new measurement. If unspecified, value is set to 1.
- `<Mname>` *(String)* Name of the measurement. Any non-empty, unique string, enclosed in quotes.
- `<param>` *(String)* Measurement Parameter to create. Case sensitive.

**For S-parameters:**

Any S-parameter available in the PNA
Single-digit port numbers CAN be separated by "_" (underscore). For example: "S21" or "S2_1"

Double-digit port numbers MUST be separated by underscore. For example: "S10_1"

For **ratioed measurements:**

Any two PNA physical receivers separated by forward slash '/' followed by comma and source port.

For example: "A/R1, 3"

[Learn more about ratioed measurements](#)

See a [block diagram](#) showing the receivers in YOUR PNA.

For **non-ratioed measurements:**

Any PNA physical receiver followed by comma and source port.

For example: "A, 4"

[Learn more about unratioed measurements](#)

See the [block diagram](#) showing the receivers in YOUR PNA.

With PNA Rev 6.2, **Ratioed** and **Unratioed** measurements can also use **logical receiver notation** to refer to receivers. This notation makes it easy to refer to receivers with an external test set connected to the PNA. You do not need to know which physical receiver is used for each test port. [Learn more](#).

For **ADC measurements:**

Any ADC receiver in the PNA followed by a comma, then the source port.

For example: "AI1,2" indicates the Analog Input1 with source port of 2.

[Learn more about ADC receiver measurements](#)

For **Balanced Measurements:**

First create an S-parameter measurement, then change the measurement using `CALC:FSIM:BAL "define" commands. [See an example](#).
receiver notation for unratioed meas of test port 9 receiver with
source port 1.

calculate2:parameter:define:extended 'ch1_a', 'b9/a10,1' 'logical
receiver notation for ratioed meas of test port 9 receiver divided
by the reference receiver for port 10 using source port 1

**Query Syntax**  
Not Applicable; see Calc:Par:Cat?

**Default**  
Not Applicable

---

**CALCulate<cnum>:PARameter:DELete[:NAME] <Mname>**

(Write-only) Deletes the specified measurement.

**See Critical Note**

**Parameters**

- `<cnum>`  
  Channel number of the measurement. There must be a selected measurement
  on that channel. If unspecified, `<cnum>` is set to 1.

- `<Mname>`  
  String - Name of the measurement

**Examples**

CALC:PAR:DEL 'TEST'
calculate2:parameter:delete 'test'

**Query Syntax**  
Not Applicable

**Default**  
Not Applicable

---

**CALCulate:PARameter:DELete:ALL**

(Write-only) Deletes all measurements on the PNA.

**See Critical Note**

**Parameters**

**Examples**

CALC:PAR:DEL:ALL

**Query Syntax**  
Not Applicable

**Default**  
Not Applicable

---

**CALCulate<cnum>:PARameter:MNUMber[:SELetc] <n>**
(Read-Write) Sets and returns the selected measurement for the channel. Most CALC: commands require that this, or CALC:PAR:SEL, be sent before a setting change is made to that measurement. Each channel can have one selected measurement.

These are the same numbers you see in the “Tr1”, “Tr2” annotation next to the parameter name on the PNA screen.

**Parameters**
- `<cnum>` Channel number of the measurement to be selected. If unspecified, `<cnum>` is set to 1.
- `<n>` Numeric - Measurement number.

**Examples**
- `CALC:PAR:MNUM 2`
- `calculate2:parameter:mnumber:select 3`

**Query Syntax**
`CALCulate:PARameter:MNUMber[:SELect]?

**Return Type**
String

**Default** Not Applicable

---

**CALCulate<cnum>**:PARameter:MODify <param>  Superseded

**Note:** This command is replaced with `CALC:PAR:MOD:EXT`. This command will continue to work for up to 4 port parameters.

(Write-only) Modifies a standard measurement using the same arguments as `CALC:PAR:DEF`. To modify an FCA measurement, use `CALC:CUST:MOD`.

**See Critical Note**

**Parameters**
- `<cnum>` Channel number of the measurement. The selected measurement on that channel will be changed. If unspecified, `<cnum>` is set to 1.
- `<param>` Measurement parameter to change to. Use the same `<param>` arguments as `CALC:PAR:DEF`.

**Examples**
- `SYST:PRESET`
- `CALC:PAR:DEF "MyMeas", S11`
- `CALC:PAR:SEL "MyMeas"`
- `CALC:PAR:MOD AR1 'changes the selected S11 measurement to an A/R1 measurement`
CALCulate<cnum>:PARameter:MODify:EXTended <param>

**Note:** This command replaces CALC:PAR:MOD as it allows modification of measurements using external multiport testsets.

(Write-only) Modifies a standard measurement using the same arguments as CALC:PAR:DEF:EXT. To modify an FCA measurement, use CALC: Cust:MOD.

**See Critical Note**

**Parameters**
- `<cnum>` Channel number of the measurement. The selected measurement on that channel will be changed. If unspecified, `<cnum>` is set to 1.
- `<param>` (String) New measurement parameter. Use the same `<param>` arguments as CALC:PAR:DEF:EXT.

**Examples**

```
SYST:PRESET
```

```
CALC:PAR:DEF:EXT "MyMeas", "S10_1"
```

```
CALC:PAR:SEL "MyMeas"
```

```
CALC:PAR:MOD:EXT "a4b4,1" 'changes the selected S10_1 measurement to an a4/b4 measurement with source port 1
```

**Query Syntax** Not Applicable

**Default** Not Applicable

CALCulate<cnum>:PARameter:SELect <Mname>
(Read-Write) Sets the selected measurement. Most CALC: commands require that this command be sent before a setting change is made. One measurement on each channel can be selected at the same time. To obtain a list of currently named measurements, use CALC:PAR:CAT?

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the measurement to be selected. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
<tr>
<td>&lt;Mname&gt;</td>
<td>String - Name of the measurement. (Do NOT include the parameter name.)</td>
</tr>
</tbody>
</table>

**Examples**

```plaintext
CALC:PAR:SEL 'TEST'
calculate2:parameter:select 'test'
```

**Query Syntax**

CALCulate:PARameter:SELect?

**Return Type**

String

**Default**

Not Applicable

---

**CALCulate<cnum>:PARameter:TNUMber?**

(Read-only) Returns the trace number of the selected trace. Select a trace using Calc:Par:Select.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the trace. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

```plaintext
CALC:PAR:TNUM?
calculate2:parameter:tnumber?
```

**Return Type**

Numeric

**Default**

Not Applicable

---

**CALCulate<cnum>:PARameter:WNUMber?**

(Read-only) Returns the window number of the selected trace. Select a trace using Calc:Par:Select.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number of the selected trace. If unspecified, &lt;cnum&gt; is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

```plaintext
CALC:PAR:WNUM?
calculate2:parameter:wnumber?
```

**Return Type**

Numeric

**Default**

Not Applicable
Last modified:

26-May-2011   Added TNUM and WNUM
10-Feb-2011    Removed defaults from MNUM and CPS
31-Oct-2008    Added Mnum select (8.33)
19-Apr-2007    Added ADC meas
    9/12/06    New Extended commands.
**Calculate:RData? Command**

**Critical Note:** CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

**CALCulate<cnum>:RDATA? <char>**

(Read-only) Returns receiver data for the selected measurement. To query measurement data, see CALC:DATA?

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<char>` Choose from any physical receiver in the PNA.

For example: "A"

Also, **REF** returns data for either R1 or R2 data depending on the source port of the selected measurement.

See the [block diagram](#) showing the receivers in YOUR PNA.

**Note:** Logical receiver notation is NOT allowed with this command. [Learn more](#).

**Example**

```
GPIB.Write "INITiate:CONTinuous OFF"
GPIB.Write "INITiate:IMMediate:*wai"
GPIB.Write "CALCulate:RDATA? A"
```

GPIB.Write "CALCulate:RDATA? REF"

**Return Type**  Depends on FORM:DATA - Two numbers per data point

**Default**  Not Applicable

**Notes:**

Generally when you query the analyzer for data, you expect that the number of data values returned will be consistent with the number of points in the sweep.

However, if you query receiver data while the instrument is sweeping, the returned values may contain zeros. For example, if your request for receiver data is handled on the 45th point of a 201 point sweep, the first 45 values will be valid data, and the remainder will contain complex zero.

This can be avoided by synchronizing this request with the end of a sweep or putting the channel in hold mode.
Learn about Unratioed Measurements
Calculate:Smoothing Commands

Controls point-to-point smoothing. Smoothing is a noise reduction technique that averages adjacent data points in a measurement trace. Choose the amount of smoothing by specifying either the number of points or the aperture. Smoothing is not the same as CALC:AVERage which averages each data point over a number of sweeps.

CALCulate:SMOothing

APERture  POINTs  [STATe]

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Smoothing
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

CALCulate<cnum>:SMOothing:APERture <num>

(Read-Write) Sets the amount of smoothing as a percentage of the number of data points in the channel.

See Critical Note

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <num> Percentage value. Choose any number between:
  1 and 25

Examples

CALC:Smo:APER 2
calculate2:smoothing:aperture 20.7

Query Syntax

CALCulate<cnum>:SMOothing:APERture?

Return Type

Numeric
CALCulate<cnum>:SMOothing:POINts <num>

(Read-Write) Sets the number of adjacent data points to average.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<num>` Number of points from 1 point to maximum of 25% of data points in the channel. For example: if number of points in a data trace = 401, the maximum value for points = 100. The points value is always rounded to the closest odd number.

Examples

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:SMO:POIN 50</td>
</tr>
<tr>
<td>calculate2:smoothing:points 21</td>
</tr>
</tbody>
</table>

Query Syntax

CALCulate<cnum>:SMOothing:POINts?

Return Type

Numeric

Default

3

CALCulate<cnum>:SMOothing[:STATe] <ON | OFF>

(Read-Write) Turns data smoothing ON or OFF.

See Critical Note

Parameters

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.
- `<ON | OFF>`
  - **ON** (or 1) - turns smoothing ON.
  - **OFF** (or 0) - turns smoothing OFF.

Examples

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALC:SMO ON</td>
</tr>
<tr>
<td>calculate2:smoothing:state off</td>
</tr>
</tbody>
</table>

Query Syntax

CALCulate<cnum>:SMOothing[:STATe]?

Return Type

Boolean (1 = ON, 0 = OFF)

Default

OFF
Calculate:Transform Commands

Specifies the settings for time domain transform.

CALCulate:TRANSform

COUPLE

TIME

CENTER KBESsel MARKer SPAN STATE STOP

[TYPE]

PARameters IMpulse MODE UNIT START STEP STIMulus

WDTh LPFRequency RTIme

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Time Domain
- Synchronizing the PNA and Controller
- SCPI Command Tree

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using Calc:Par:Select

CALCulate<cnum>:TRANSform:COUPle:PARameters <num>

(Read-Write) Specifies the time domain transform parameters to be coupled. The settings for those parameters will be copied from the selected measurement to all other measurements on the channel.

- To turn coupling ON and OFF, use SENS:COUP:PAR
- To specify Gating parameters to couple, use CALC:FILT:COUP:PAR

Learn more about Time Domain Trace Coupling

See Critical Note

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <num> (Numeric) Parameters to couple. To specify more than one parameter, add the
numbers.

1 - Transform Stimulus (Start, Stop, Center, and Span TIME settings.)

2 - Transform State (ON / OFF)

4 - Transform Window (Kaiser Beta / Impulse Width)

8 - Transform Mode (Low Pass Impulse, Low Pass Step, Band Pass)

16 - Transform Distance Marker Units

Examples

'To couple all parameters:

CALC:TRAN:COUP:PAR 31

'To couple Stimulus and Mode:

calculate2:transform:couple:parameters 9

Query Syntax

CALCulate<cnum>:TRANsform:COUPle:PARameters?

Return Type

Numeric

Default

29 (All parameters except 2 - Transform State)

CALCulate<cnum>:TRANsform:TIME:CENTer <num>

(Read-Write) Sets the center time for time domain measurements.

See Critical Note

Parameters

<cnun> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<num> Center time in seconds; any number between:
± (number of points-1) / frequency span

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

CALC:TRAN:TIME:CENT 1e-8
calculate2:transform:time:center 15 ps

Query Syntax

CALCulate<cnum>:TRANsform:TIME:CENTer?

Return Type

Numeric

Default

0
CALCulate<cnum>:TRAnSform:TIME:IMPulse:WIDTh <num>

(Read-Write) Sets the impulse width for the transform window.

See Critical Note

Parameters

<table>
  <tr><td><cnum></td><td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.</td></tr>
  <tr><td><num></td><td>Impulse width in seconds; Choose any number between: \( .6 / \text{frequency span} \) and \( 1.39 / \text{frequency span} \)</td></tr>
</table>

Examples

- CALC:TRAN:TIME:IMP:WIDTh 10
- calculate2:transform:time:impulse:width 13

Query Syntax

CALCulate<cnum>:TRAnSform:TIME:IMPulse:WIDTh?

Return Type

Numeric

Default

.98 / Default Span

CALCulate<cnum>:TRAnSform:TIME:KBESsel <num>

(Read-Write) Sets the parametric window for the Kaiser Bessel window.

See Critical Note

Parameters

<table>
  <tr><td><cnum></td><td>Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.</td></tr>
  <tr><td><num></td><td>Window width for Kaiser Bessel in seconds; Choose any number between: 0.0 and 13.0</td></tr>
</table>

Examples

- CALC:TRAN:TIME:KBES 10
- calculate2:transform:time:kbessel 13

Query Syntax

CALCulate<cnum>:TRAnSform:TIME:KBESsel?

Return Type

Numeric

Default

6

CALCulate<cnum>:TRAnSform:TIME:LPFREQuency
(Write-only) Sets the start frequencies in LowPass Mode.

See Critical Note

Parameters

<n> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <n> is set to 1.

Examples

CALC:TRAN:TIME:LPFR
calculate2:transform:time:lpfrequency

Query Syntax

Not applicable

Default

Not applicable

CALCulate<n>:TRANsform:TIME:MARKer:MODE <char>

(Read-Write) Specifies the measurement type in order to determine the correct marker distance.

- Select Auto for S-Parameter measurements.
- Select Reflection or Transmission for arbitrary ratio or unratioed measurements.

This setting affects the display of ALL markers for only the ACTIVE measurement.

Learn more about Distance Markers.

See Critical Note

Parameters

<n> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <n> is set to 1.

<char> Choose from:

AUTO If the active measurement is an S-Parameter, automatically chooses reflection or transmission. If non S-Parameter measurements, reflection is chosen.

REFLaction Displays the distance from the source to the receiver divided by two (to compensate for the return trip.)

TRANsmission Displays the distance from the source to the receiver.

Examples

CALC:TRAN:TIME:MARKer:MODE REFL
calculate2:transform:time:marker:mode auto

Query Syntax

CALCulate<n>:TRANsform:TIME:MARKer:MODE?
CALCulate<cnum>:TRANsform:TIME:MARKer:UNIT <char>

(Read-Write) Specifies the unit of measure for the display of marker distance values. This setting affects the display of all markers for only the active measurement (unless Distance Maker Units are coupled using CALC:TRAN:COUP:PAR).

Learn more about Distance Markers.

See Critical Note

Parameters

- **<cnum>** Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- **<char>** Choose from:
  - METRs
  - FEET
  - INCHes

Examples

- CALC:TRAN:TIME:MARK:UNIT INCH
- calculate2:transform:time:marker:unit feet

Query Syntax

CALCulate<cnum>:TRANsform:TIME:MARKer:UNIT?

Return Type: Character

Default: METRs

CALCulate<cnum>:TRANsform:TIME:SPAN <num>
(Read-Write) Sets the span time for time domain measurements.

See Critical Note

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.

- `<num>` Span time in seconds; any number between: 0 and 2*[(number of points-1) / frequency span]

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>CALC:TRAN:TIME:SPAN 1e-8</code></td>
<td>calculate2:transform:time:span maximum</td>
</tr>
</tbody>
</table>

**Query Syntax**

`CALCulate<cnum>:TRANsform:TIME:SPAN?`

**Return Type**

Numeric

**Default**

20 ns

---

**CALCulate<cnum>:TRANsform:TIME:STARt <num>**

(Read-Write) Sets the start time for time domain measurements.

See Critical Note

**Parameters**

- `<cnum>` Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, `<cnum>` is set to 1.

- `<num>` Start time in seconds; any number between: ±(number of points-1) / frequency span

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>

**Query Syntax**

`CALCulate<cnum>:TRANsform:TIME:STARt?`

**Return Type**

Numeric

**Default**

-10 ns
CALCulate<cnum>:=TRANsform:TIME:STATe <ON | OFF>

(Read-Write) Turns the time domain transform capability ON or OFF.

See Critical Note

Note: Sweep type must be set to Linear Frequency in order to use Time Domain Transform.

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <ON|OFF> ON (or 1) - turns time domain ON.
  OFF (or 0) - turns time domain OFF.

Examples

- CALC:TRAN:TIME:STAT ON
- calculate2:transform:time:state off

Query Syntax
CALCulate<cnum>:TRANsform:TIME:STATe?

Return Type
Boolean (1 = ON, 0 = OFF)

Default
OFF

CALCulate<cnum>:=TRANsform:TIME:STOP <num>

(Read-Write) Sets the stop time for time domain measurements.

See Critical Note

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <num> Stop time in seconds; any number between:
  ± (number of points-1) / frequency span

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples

- CALC:TRAN:TIME:STOP 1e-8
- calculate2:transform:time:stop maximum

Query Syntax
CALCulate<cnum>:=TRANsform:TIME:STOP?

Return Type
Numeric

Default
10 ns
CALCulate<cnum>:TRANsform:TIME:STEP:RTIMe <num>

(Read-Write) Sets the step rise time for the transform window.

See Critical Note

Parameters

(cnun) Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<num> Rise time in seconds; Choose any number between: 
0.45 / frequency span and 1.48 / frequency span

Examples
CALC:TRAN:TIME:STEP:RTIM 1e-8
calculate2:transform:time:step:rtime 15 ps

Query Syntax
CALCulate<cnum>:TRANsform:TIME:STEP:RTIMe?

Return Type
Numeric

Default
.99 / Default Span

CALCulate<cnum>:TRANsform:TIME:STIMulus <char>

(Read-Write) Sets the type of simulated stimulus that will be incident on the DUT.

See Critical Note

Parameters

(cnun) Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<char> Choose from:
STEP - simulates a step DUT stimulus
IMPulse - simulates a pulse DUT stimulus

STEP can ONLY be used when CALC:TRAN:TIME:TYPE is set to LPAs (Lowpass). (STEP cannot be used with TYPE = BPAs.)

:STIM STEP will set :TYPE to LPAs

:TYPE BPAs will set :STIM to IMPulse

Examples
CALC:TRAN:TIME:STIM STEP
calculate2:transform:time:stimulus impulse

Query Syntax
CALCulate<cnum>:TRANsform:TIME:STIMulus?

Return Type
Character
CALCulate<cnum>:TRANsform:TIME[:TYPE] <char>

(Read-Write) Sets the type of time domain measurement.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<char> Type of measurement. Choose from:
- **LPASs** - Lowpass; Must also send CALC:TRAN:TIME:LPFRequency before calibrating.
- **BPASs** - Bandpass;

BPASs can only be used when CALC:TRAN:TIME:STIM is set to IMPulse. (BPASs cannot be used with :STIM = STEP)

:STIM STEP will set :TYPE to LPASs

:TYPE BPASs will set :STIM to IMPulse

Examples

| CALC:TRAN:TIME LPAS |
| calculate2:transform:time:type bpas |

Query Syntax

CALCulate<cnum>:TRANsform:TIME[:TYPE]?

Return Type

Character

**Default** BPAS
**X Values Command**

**CALCulate\(<cnum>:X[:VALues]\)?**

*(Read-only)* Returns the stimulus values for the selected measurement in the current units. You can select one measurement for each channel using `Calc:Par:Select`. This command can be used for all Measurement Classes.

**Parameters**

\(<cnum>\)  
Any existing channel number; if unspecified, value is set to 1.

**Examples**

1. `Calc:Par:Sel "MyGCATrace"`
2. `CALC:X?`

**Return Type**  
Depends on `FORM:DATA` command

**Default**  
Not applicable

Last Modified:

12-Aug-2009    MX New topic
Control Commands

Specifies the settings to remotely control the rear panel connectors, an external test set, Calpod modules, and ECal Module state.

CONTrol

AUXiliary - More Commands

CALPod:COMMand

CHANnel:INTerface:CONTrol:

| CONFig:RECall
| [STATe]

ECAL:MODule:

| PATH:

| COUNt?
| STATe

| STATe

EXTernal:TESTset - More Commands

HANDler - More Commands

NOISE:SOURce[:STATe]

SIGNal:

| TRIGger

| ATBA
| OUTP

Click on a blue keyword to view the command details.

Red command is superseded.

See Also
Example Programs

Synchronizing the PNA and Controller

SCPI Command Tree

See a pinout and detailed description of the rear panel connectors:

- Auxilliary IO connector
- External Test Set IO connector
- Material Handler IO connector

**CONTrol:CALPod:COMMand <string>**

*(Write-only)* Sends commands that control a Calpod module.

See all Calpod commands.

Learn more about Calpod.

**Parameters**

<string> Calpod command. See ALL Calpod commands that can be used in this string.

**Examples**

```
'CONT:CALP:COMM 'CALP:INIT:ACT'

Enclose all strings in SINGLE quotes (NOT double quotes)
```

**Query Syntax**

CONTrol:CALPod:COMMand? <string>

Relevant only for query strings.

**Return Type** String

**Default** Not Applicable

**CONTrol:CHANnel:INTerface:CONTrol:CONFig:RECall[:STATE] <string>**
(Write-only) Recalls an Interface Control configuration file. Learn more about Interface Control.

**Parameters**

<string> File name and extension (.xml) of the configuration file to recall. Files are typically stored in the default folder "C:/Program Files/Agilent/Network Analyzer/Documents". To recall from a different folder, specify the full path name.

**Examples**

CONT:CHAN:INT:CONT:CONF:REC 'MyConfigFile.xml'


**Query Syntax** Not Applicable

**Default** Not Applicable

---

**CONTrol:CHANnel:INTERface:CONtroL[:STATe] <bool>**

(Read-Write) Enables and disables ALL Interface Control settings. To send data, the individual interfaces must also be enabled. Learn more about Interface Control.

**Parameters**

<bool> Boolean

OFF (0) - Interface Control is disabled; NO control data is sent.

ON (1) - Interface Control is enabled.

**Examples**

CONT:CHAN:INT:CONT 1

control:channel:interface:control:state 0

**Query Syntax** CONTrol:CHANnel:INTERface:CONtroL[:STATe]?

**Return Type** Boolean

**Default** OFF (0)

---

**CONTrol:ECAL:MODule<num>:PATH:COUNt? <name>**

(Read-only) Returns the number of unique states that exist for the specified path name on the selected ECal module.

This command performs exactly the same function as SENS:CORR:CKIT:ECAL:PATH:COUNt?

Use the CONT:ECAL:MOD:PATH:STAT command to set the module into one of those states.
Use `SENS:CORR:CKIT:ECAL:PATH:DATA?` to read the data for a state.

**Parameters**

[num] Optional argument. USB number of the ECal module. If unspecified (only one ECal module is connected to the USB), `<num>` is set to 1. If two or more modules are connected, use `SENS:CORR:CKIT:ECAL:LIST?` to determine how many, and `SENS:CORR:CKIT:ECAL:INF?` to verify their identities.

<name> Name of the path for which to read number of states. Choose from:

**Reflection paths**

- A
- B
- C (4-port modules)
- D (4-port modules)

**Transmission paths**

- AB
- AC (4-port modules)
- AD (4-port modules)
- BC (4-port modules)
- BD (4-port modules)
- CD (4-port modules)

Note: For each transmission path, the first of the available states is the through state, the second is the confidence (attenuator) state.

**Examples**

```
CONT:ECAL:MOD:PATH:COUNt? A
control:ecal:module2:path:count? cd
```

See example program

**Return Type** Integer

**Default** Not Applicable

```
CONTro1:ECAL:MODule<num>:PATH:STATe <path>, <stateNum>
```
(Write-only) Sets the internal state of the selected ECAL module. This command supersedes CONT:ECAL:MOD:STAT.

- Use CONT:ECAL:MOD:PATH:COUN? to read the number of unique states that exist for the specified path name on the module.
- Use SENS:CORR:CKIT:ECAL:PATH:DATA? to read the data for a state (from the module memory) corresponding to the stimulus values of a channel.

**Parameters**

[num]  Optional argument. USB number of the ECAL module. If unspecified (only one ECAL module is connected to the USB), <num> is set to 1. If two or more modules are connected, use SENS:CORR:CKIT:ECAL:LIST? to determine how many, and SENS:CORR:CKIT:ECAL:INF? to verify their identities.

<path>  Path name for which to set a state.

**Note:** The impedance paths are not independent. For example, changing the impedance presented on path A will cause a change to the impedance on path B.

Choose from:

**Reflection paths**

- A
- B
- C (4-port modules)
- D (4-port modules)

**Transmission paths**

- AB
- AC (4-port modules)
- AD (4-port modules)
- BC (4-port modules)
- BD (4-port modules)
- CD (4-port modules)

<stateNum>  Number of the state to set. Refer to the following table to associate the
<stateNum> with a state in your ECal module.

In addition, CONT:ECAL:MOD:PATH:COUNt? returns the number of states in the specified ECal module.

<table>
<thead>
<tr>
<th>&lt;stateNum&gt;</th>
<th>N4432A and N4433A States</th>
<th>N4431A States</th>
<th>N469x States**</th>
<th>8509x States</th>
</tr>
</thead>
<tbody>
<tr>
<td>One-Port Reflection States</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Open</td>
<td>Open</td>
<td>Impedance 1</td>
<td>Open</td>
</tr>
<tr>
<td>2</td>
<td>Short</td>
<td>Short</td>
<td>Impedance 2</td>
<td>Short</td>
</tr>
<tr>
<td>3</td>
<td>Impedance 1</td>
<td>Impedance 1</td>
<td>Impedance 3</td>
<td>Impedance 1</td>
</tr>
<tr>
<td>4</td>
<td>Impedance 2</td>
<td>Impedance 2</td>
<td>Impedance 4</td>
<td>Impedance 2</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td>Impedance 5</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td>Impedance 6</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td>Impedance 7</td>
<td></td>
</tr>
<tr>
<td>Two-Port Transmission States</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Thru</td>
<td>Thru</td>
<td>Thru</td>
<td>Thru</td>
</tr>
<tr>
<td>2</td>
<td>Confidence</td>
<td>Confidence</td>
<td>Confidence</td>
<td>Confidence</td>
</tr>
</tbody>
</table>

** The following modules have only FOUR Impedance states (1, 2, 3, 4): N4690B, N4691B, N4692A, N4696B.

Examples

```
control:ecal:module2:state BC,1
```

See example program

Query Syntax Not Applicable

Default Not Applicable

CONTrol:ECAL:MODule<num>:STATe <value> Superseded
This command is replaced with `CONT:ECAL:MOD:PATH:STATe`.

*(Write-only)* Sets the internal state of the selected ECAL module.

**Parameters**

[**num**] Optional argument. USB number of the ECal module. If unspecified (only one ECal module is connected to the USB), `<num>` is set to 1. If two or more modules are connected, use `SENS:CORR:COLL:CKIT:INF?` to verify their identity.

<**value**> Integer code for switching the module. The following are codes for Agilent ECal modules.

### Agilent 8509x Modules

<table>
<thead>
<tr>
<th>State</th>
<th>Port A</th>
<th>Port B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Short</td>
<td>43</td>
<td>43</td>
</tr>
<tr>
<td>Load</td>
<td>33</td>
<td>33</td>
</tr>
<tr>
<td>Mismatch</td>
<td>4</td>
<td>16</td>
</tr>
<tr>
<td>Thru</td>
<td></td>
<td>84</td>
</tr>
<tr>
<td>Confidence</td>
<td></td>
<td>20</td>
</tr>
</tbody>
</table>

### Agilent N469x Modules

<table>
<thead>
<tr>
<th>State</th>
<th>Port A</th>
<th>Port B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open</td>
<td>36</td>
<td>33</td>
</tr>
<tr>
<td>Short</td>
<td>39</td>
<td>45</td>
</tr>
<tr>
<td>Load</td>
<td>37</td>
<td>37</td>
</tr>
<tr>
<td>Mismatch (Offset short)</td>
<td>53</td>
<td>53</td>
</tr>
<tr>
<td>Impedance 5 (Offset open)</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Impedance 6 (Offset short)</td>
<td>21</td>
<td>21</td>
</tr>
<tr>
<td>Impedance 7 (Offset short)</td>
<td>38</td>
<td>41</td>
</tr>
<tr>
<td>Thru</td>
<td></td>
<td>42</td>
</tr>
<tr>
<td>Confidence</td>
<td></td>
<td>40</td>
</tr>
<tr>
<td>Agilent N4431A Modules</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>State</strong></td>
<td><strong>Port A</strong></td>
<td><strong>Port B</strong></td>
</tr>
<tr>
<td>-----------</td>
<td>------------</td>
<td>------------</td>
</tr>
<tr>
<td>Open</td>
<td>-1398</td>
<td>-1384</td>
</tr>
<tr>
<td>Short</td>
<td>-1350</td>
<td>-1381</td>
</tr>
<tr>
<td>Load</td>
<td>26985</td>
<td>-26986</td>
</tr>
<tr>
<td>Mismatch</td>
<td>-26986</td>
<td>26985</td>
</tr>
<tr>
<td><strong>Path</strong></td>
<td><strong>Thru</strong></td>
<td><strong>Confidence</strong></td>
</tr>
<tr>
<td>AB Path</td>
<td>-2590</td>
<td>598</td>
</tr>
<tr>
<td>AC Path</td>
<td>-4011</td>
<td>85</td>
</tr>
<tr>
<td>AD Path</td>
<td>-2517</td>
<td>16042</td>
</tr>
<tr>
<td>BC Path</td>
<td>-1650</td>
<td>598</td>
</tr>
<tr>
<td>BD Path</td>
<td>-4011</td>
<td>85</td>
</tr>
<tr>
<td>CD Path</td>
<td>-1352</td>
<td>16042</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Agilent N4432A and N4433A Modules</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>State</strong></td>
</tr>
<tr>
<td>-----------</td>
</tr>
<tr>
<td>Open</td>
</tr>
<tr>
<td>Short</td>
</tr>
<tr>
<td>Load</td>
</tr>
<tr>
<td>Offset Short</td>
</tr>
<tr>
<td><strong>Path</strong></td>
</tr>
<tr>
<td>AB Path</td>
</tr>
<tr>
<td>AC Path</td>
</tr>
<tr>
<td>AD Path</td>
</tr>
<tr>
<td>BC Path</td>
</tr>
<tr>
<td>BD Path</td>
</tr>
<tr>
<td>CD Path</td>
</tr>
</tbody>
</table>

**Examples**

CONT:ECAL:MOD:STAT 36
control:ecal:module2:state 38

**Query Syntax**

<table>
<thead>
<tr>
<th>Default</th>
<th>Not Applicable</th>
</tr>
</thead>
</table>

**CONTrol:NOISe:SOURce[:STATe] <bool>**
(Read-Write) Set and read the noise source (28V) ON and OFF.

**Parameters**

<bool> Boolean  

**OFF (0)** - Noise Source OFF  

**ON (1)** - Noise Source ON

**Examples**

CONT:NOIS:SOUR 1  

control:noise:source:state 0

**Query Syntax**

CONTrol:NOISe:SOURce[:STATe]?

**Return Type**

Boolean

**Default**

For PNA models with a Noise Figure option (028/029/H29), the 28V line is ON at application start and after a preset. The ON/OFF state is also available from a PNA softkey menu.

For PNA models WITHOUT a Noise Figure option (028/029/H29), the 28V line is OFF at application start and its state is not affected by a preset. The ON/OFF state is NOT available from a PNA softkey menu.

---

**CONTrol:SIGNal <conn>,<char>**

(Read-Write) Configures external triggering in the PNA.

- To control BNC1 and BNC2 with this command, then you **MUST** have TRIG:PREF:AIGLoabl = **ON**. [Learn more](#)
- **Trigger:Sequence:Source** is automatically set to External when CONTrol:SIGNal is sent.
- Edge triggering is only available on some Microwave PNA models.
- For more information, see External Triggering in the PNA.

**Parameters**

<conn> Rear Panel connector to send or receive trigger signals. Choose from:

**BNC1** Trigger IN from rear-panel Trigger IN BNC connector

**AUXT** Trigger IN from AUX IO connector Pin 19

**Note:** Only one of the input connectors is active at a time. When a command is sent to one, the PNA automatically makes the other INACTIVE.
BNC2  Trigger OUT to rear-panel Trigger OUT BNC connector.

MATHtrigger - Trigger IN from rear-panel Material Handler connector Pin 18

RDY - Ready for trigger OUT. (Not available on E836x models)

- PNA-X:  Meas Trig RDY
- PNA-L: AUX I/O p18, Handler I/O p21 (Some models)

**INACTIVE** - Disables the specified connector <conn>.

Choose from ONLY the following when <conn> is set to BNC1 or AUX1 or MATHtrigger:

- **TIENEGATIVE** - (Trigger In Edge Negative) - Triggers the PNA when receiving a negative going signal
- **TIEPOSITIVE** - (Trigger In Edge Positive) - Triggers the PNA when receiving a positive going signal
- **TILLOW** - (Trigger In Level Low) - Triggers the PNA when receiving a low level signal
- **TILHIGH** - (Trigger In Level High) - Triggers the PNA when receiving a High-level signal

Choose from ONLY the following when <conn> is set to BNC2:

Use **CONTrl:SIGNal:TRIGger:OUTP** to enable the BNC2 output.

The following selections send a positive or negative pulse before or after each trigger acquisition. This normally occurs each sweep unless a channel is in point trigger mode.

- **TOPPAFTER** - (Trigger Out Pulse Positive After) - Sends a POSITIVE going TTL pulse at the END of each trigger acquisition.
- **TOPPBEFORE** - (Trigger Out Pulse Positive Before) - Sends a POSITIVE going TTL pulse at the START of each trigger acquisition.
- **TOPNAFTER** - (Trigger Out Pulse Negative After) - Sends a NEGATIVE going TTL pulse at the END of each trigger acquisition.
- **TOPNBEFORE** - (Trigger Out Pulse Negative Before) - Sends a NEGATIVE going TTL pulse at the START of each trigger acquisition.

Choose from ONLY the following when <conn> is set to RDY:
• **LOW** Outputs a TTL low when the PNA is ready for trigger. (Default setting)

• **HIGH** Outputs a TTL high when the PNA is ready for trigger.

### Examples

```plaintext
CONT:SIGN BNC1, TIENEGATIVE
control:signal bnc2, toppbefore
CONT:SIGN RDY, LOW
```

### Query Syntax

`CONTrol:SIGNal? <conn>`

In addition to the arguments listed above, the following is also a possible returned value:

**NAVAILABLE** - This feature is not available on this PNA

### Return Type

Character

**Default** At Preset:

- BNC1 = INACTIVE
- BNC2 = INACTIVE
- AUXT = TILHIGH

When **Output is enabled**:

- BNC1 = INACTIVE
- BNC2 = TOPPAFTER
- AUXT = TILHIGH

---

**CONTrol:SIGNal:TRIGger:ATBA <bool>**

*(Read-Write)* **Accept Trigger Before Armed** Determines what happens to an EDGE trigger signal if it occurs before the PNA is ready to be triggered. (LEVEL trigger signals are always ignored.) For more information, see [External triggering](#).

#### Parameters

- `<bool>` Boolean

  **OFF (0)** - A trigger signal is ignored if it occurs before the PNA is ready to be triggered.

  **ON (1)** - A trigger signal is remembered and then used when the PNA becomes armed (ready to be triggered). The PNA remembers only one trigger signal.

### Examples

```plaintext
CONT:SIGN:TRIG:ATBA 0
control:signal:trigger:atba ON
```
**Query Syntax**  
CONTrol:SIGNal:TRIGger:ATBA?

**Return Type**  
Boolean

**Default**  
OFF

---

**CONTroll:SIGNal:TRIGger:OUTP <bool>**

(Read-Write) **Output Enabled**  
The PNA can be enabled to send trigger signals out the rear-panel TRIGGER OUT BNC connector. Use CONTrol:SIGNal to configure for output triggers.

For more information, see [External triggering](#).

**Parameters**

<bool>  
Boolean

OFF (0) - PNA does NOT output trigger signals.

ON (1) - PNA DOES output trigger signals.

**Examples**

CONT:SIGN:TRIG:OUTP 1

control:signal:trigger:outp OFF

---

**Query Syntax**  
CONTrol:SIGNal:TRIGger:OUTP?

**Return Type**  
Boolean

**Default**  
OFF

---

**Last Modified:**

- 7-Jan-2011  
  Added Calpod

- 3-Nov-2010  
  Noise source on/off for all models

- 24-Feb-2009  
  Replaced True/False

- 14-Mar-2008  
  Added RDY argument to CONT:SIGN

- 25-Feb-2008  
  Clarified CONT:SIGN

- 30-Jan-2008  
  Added ECal states note

- 22-Aug-2007  
  Added noise command (8.0)

- 18-Jan-2007  
  Fixed count? example
CSET:Fixture Commands

Manages several aspects of Cal Sets.

CSET:

| EXISts? |
| FIXTure: |
| DEEMbed |
| EMBed |

Click on a blue keyword to view the command details.

Note: There is no user-interface equivalent for these commands.

See Also

- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

CSET:EXISts? <string>

(Read-only) Returns whether or not the specified Cal Set exists on the PNA.

**Parameters**

| <string> | Name or GUID of the Cal Set enclosed in quotes. |

**Examples**

```plaintext
dim check

check = CSET:EXISts? "MyCalSet"

check = CSET:EXISts? "7C4EEA5E-40D2-4D70-A048-33BFFE704163"
```

**Return Type** Boolean

- **ON** or **1** - Cal Set exists.
- **OFF** or **0** - Cal Set does NOT exist.

**Default** Not Applicable
CSET:FIXTure:DEEMbed <cs1>,<cs2>,<s2p>,<port>, <compPwr>[,extrap]

(Write-only) De-embeds a fixture from an existing Cal Set based on an S2P file. A new Cal Set is created with the effects of the fixture removed.

When the new Cal Set is applied to a channel, the effects of fixturing are removed from the measurement data. Do NOT enable fixturing. The effects of the fixture are removed when the new Cal Set is selected and correction is turned ON.

**Parameters**

- `<cs1>` (String) Name of an existing Cal Set which resides on the PNA.
- `<cs2>` (String) Name of new Cal Set which contains updated error terms with fixture de-embedded.
- `<s2p>` (String) Name of the S2P file which characterizes the adapter(fixture).
- `<port>` (Numeric) Port number from which fixture will be de-embedded.
- `<compPwr>` (Boolean) 
  - **ON (1)** - When the Cal Set contains a power correction array for the fixture port, that array will be compensated for the fixture loss.
  - **OFF (0)** - Do not compensate for loss in source power through the fixture.
- `[extrap]` (Boolean) Optional argument.
  - **ON (1)** - Applies a simple extrapolation when the S2P file has a narrower frequency range than the Cal Set. The values for the first and last data points are extended in either direction to cover the frequency range of the Cal Set.
  - **OFF (0)** - Extrapolation is NOT performed (default setting).

**Examples**

```plaintext
CSET:FIXT:DEEM "MyCalSet","MyNewCalSet","Fixture.s2p",1,1
```

```plaintext
iset:fixture:deembed
"MyCalSet","MyNewCalSet","Fixture.s2p",1,1,1 'extrapolation is performed if the s2p frequency range is narrower than that of the Cal Set.
```

**Query Syntax**  Not Applicable

**Default**  Not Applicable
CSET:FIXTure:EMBed <cs1>,<cs2>,<s2p>,<port>, <compPwr>[,extrap]

(Write-only) Embeds a fixture (usually a matching network) into an existing Cal Set based on an S2P file. A new Cal Set is created with the effects of the matching network included in the correction data.

When the new Cal Set is applied to a channel, the effects of the fixture are included in the measurement data. Do NOT enable fixturing. The effects of the matching network are included when the new Cal Set is selected and correction is turned ON.

Parameters

<cs1> (String) Name of an existing Cal Set which resides on the PNA.
<cs2> (String) Name of new Cal Set which contains updated error terms with fixture embedded.
<s2p> (String) Name of the S2P file which characterizes the fixture / matching network.
<port> (Numeric) Port number to which fixture will be added.
<compPwr> (Boolean)

ON (1) - Increase the source power to compensate for the loss through the fixture. The result is that the specified power level will be correct at the DUT input.

Warning: enabling power compensation can result in an increase in test port power and consequently, increased power to the DUT. Use with caution.

OFF (0) - Do not compensate for loss in source power through the matching network.

[extrap] (Boolean) Optional argument.

ON (1) - Applies a simple extrapolation when the S2P file has a narrower frequency range than the Cal Set. The values for the first and last data points are extended in either direction to cover the frequency range of the Cal Set.

OFF (0) - Extrapolation is NOT performed (default setting).

Examples

CSET:FIXT:DEEM "MyCalSet","MyNewCalSet","Fixture.s2p",1,1

cset:fixture:deembed
"MyCalSet","MyNewCalSet","Fixture.s2p",1,1,1 'extrapolation is performed if the s2p frequency range is narrower than that of the Cal Set.'

Query Syntax Not Applicable

Default Not Applicable
Last Modified:

30-Nov-2010  Added extrapolation argument
23-Aug-2010   Fixed FIXT
13-Apr-2010   MX New topic
**Display Commands**

Controls the settings of the front panel screen.

| Display: | 
| --- | --- |
| ANNotation | FREQuency[:STATE] |
| | MESSage:STATe |
| | STATus |
| ARRange | CATalog? |
| COLoR | More Commands |
| ENABle | FSIGn |
| | TMAX |
| TILE | 
| WINDow | ANNotation |
| | MARKer[:STATE] |
| | NUMBer |
| | RESolution |
| | RESPonse |
| | STIMulus |
| | SINGLE[:STATE] |
| | SIZE |
| | STATe |
| | SYMBol |
| | XPOSition |
| | YPOSition |
| | TRACe:STATe |
| CATalog? | ENABle |
| | SIZE |
| | [STATe] |
| TABLE | TITLE |
| | DATA |
| | [STATe] |
| TRACe | DELete |
| | FEED |
| | GRATicule:GRID:LTYPE |
| | MEMory[:STATE] |
Click on a blue keyword to view the command details.

Red keywords are superseded.

See Also

- To read the window number of the selected trace, use Calc:Par:WNUM.
- See an example using some of these commands
- Synchronizing the PNA and Controller
- Learn about Screen Setup
- SCPI Command Tree

**DISPlay:ANNotation:FREQuency[:STATe] <ON | OFF>**

*(Read-Write)* Turns frequency information on the display title bar ON or OFF for all windows.

**Parameters**

- `<ON | OFF>`
  - **ON** (or 1) - turns frequency annotation ON.
  - **OFF** (or 0) - turns frequency annotation OFF.

**Examples**

```
DISP:ANN:FREQ ON
display:annotation:frequency:state off
```

**Query Syntax**

`DISPlay:ANNotation:FREQuency[:STATe]?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON (1)
**DISPlay:ANAnnotation:MESSage:STATe <ON | OFF>**

(Read-Write) Enables and disables error pop-up messages on the display.

**Parameters**

<ON | OFF>  
ON (or 1) - enables error pop-up messages  
OFF (or 0) - disables error pop-up messages

**Examples**

```
DISP:ANN:MESS:STAT ON  
display:annotation:message:state off
```

**Query Syntax**  
DISPlay:ANAnnotation:MESSage:STATe?

**Return Type**  
Boolean (1 = ON, 0 = OFF)

**Default**  
ON (1)

---

**DISPlay:ANAnnotation[:STATus] <ON | OFF>**

(Read-Write) Turns the status bar at the bottom of the screen ON or OFF. The status bar displays information for the active window.

**Parameters**

<ON | OFF>  
ON (or 1) - turns status bar ON  
OFF (or 0) - turns status bar OFF

**Examples**

```
DISP:ANN ON  
display:annotation:status off
```

**Query Syntax**  
DISPlay:ANAnnotation[:STATus]?

**Return Type**  
Boolean (1 = ON, 0 = OFF)

**Default**  
Last state that was set

---

**DISPlay:ARRange <char>**
(Write-only) Places EXISTING measurements into pre- configured window arrangements. Overlay, Stack(2), Split(3), and Quad(4) creates new windows. To learn more, see Arrange Existing Measurements.

**Parameters**

<char> Window arrangement. Choose from:

- TILE - tiles existing windows
- CASCade - overlaps existing windows
- OVERlay - all traces placed in 1 window
- STACk - 2 windows
- SPLit - 3 windows
- QUAD - 4 windows

**Examples**

```
DISP:ARR CASC
display:arrange cascade
```

**Query Syntax**

Not Applicable

**Default**

TILE

**DISPlay:CATalog?**

(Read-only) Returns the existing Window numbers.

To read the window number of the selected trace, use Calc:Par:WNUM.

**Return Type**

String of Character values, separated by commas

**Example**

Two windows with numbers 1 and 2 returns: "1,2"

**Default**

Not applicable

**DISPlay:ENABLE <ON | OFF>**
(Read-Write) Specifies whether to disable or enable all analyzer display information in all windows in the analyzer application. Marker data is not updated. More CPU time is spent making measurements instead of updating the display.

**Parameters**

- **<ON | OFF>**
  - **ON** (or 1) - turns the display ON.
  - **OFF** (or 0) - turns the display OFF.

**Examples**

- `DISP:ENAB ON`
- `display:enable off`

**Query Syntax**

- `DISP:ENABle?`

**Return Type**

- Boolean (1 = ON, 0 = OFF)

**Default**

- ON

---

**DISPlay:FSIGn <ON | OFF>**

(Read-Write) Shows or hides the window which displays global pass/fail results.

**Parameters**

- **<ON | OFF>**
  - **ON** (or 1) - displays the pass/fail dialog
  - **OFF** (or 0) - hides the pass/fail dialog

**Examples**

- `DISP:FSIG ON`
- `display:fsign off`

**Query Syntax**

- `DISP:FSIGn?`

**Return Type**

- Boolean (1 = ON, 0 = OFF)

**Default**

- OFF

---

**DISPlay:TMAX <bool>**
(Read-Write) Maximizes (isolates) or restores the active trace in the active window. When turned ON, the active trace is the ONLY trace on the display. All other traces are hidden. Learn more.

**Parameters**

- **<bool> ON (or 1) - Maximize / isolates the active trace.**
- **OFF (or 0) - Restores other traces to the normal window setting.**

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:TMAX ON</td>
<td></td>
</tr>
<tr>
<td>display:tmax 0</td>
<td></td>
</tr>
</tbody>
</table>

**Query Syntax**

`DISPlay:TMAX?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

---

**DISPlay[:TILE] - Superseded**

This command is replaced by **DISP:ARRange**

(Write-only) Tiles the windows on the screen.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP</td>
<td></td>
</tr>
<tr>
<td>display:tile</td>
<td></td>
</tr>
</tbody>
</table>

**Default**

Not Applicable

---

**DISPlay:WINDow<wnum>:ANNotation:MARKer:NUMBer <num>**

This command replaces **DISP:WIND:ANN:MARK:SINGle**

(Read-Write) Sets the number of marker readouts to display per trace. Display up to 20 marker readouts per window.

See other SCPI Marker commands. Learn more about Marker readout.

**Parameters**

- **<wnum> Any existing window number. If unspecified, value is set to 1.**
- **<num> Number of marker readouts to display. Choose a value between 1 and 10.**

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:ANN:MARK:NUMB 7</td>
<td></td>
</tr>
<tr>
<td>display:window:annotation:marker:number 2</td>
<td></td>
</tr>
</tbody>
</table>

**Query Syntax**

`DISPlay:WINDow:ANNotation:MARKer:NUMBer?`

**Return Type**

Numeric
**DISPlay:WINDoW<wnum>:ANNotation:MARKer:RESolution:STIMulus <num>**

(Read-Write) For the X-axis (stimulus), sets the number digits to display after the decimal point in marker readouts.

See other SCPI Marker commands. Learn more about Marker readout.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;wnum&gt;</td>
<td>Any existing window number. If unspecified, value is set to 1.</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>Number of digits to display. Choose a value between 2 and 6.</td>
</tr>
</tbody>
</table>

**Examples**

```
DISP:WIN:ANN:MARK:RES:STIM 2
display:window:annotation:marker:resolution:stimulus 4
```

**Query Syntax**

```
DISPlay:WINDoW:ANNotation:MARKer:RESolution:STIMulus?
```

**Return Type**

Numeric

**Default**

3

**DISPlay:WINDoW<wnum>:ANNotation:MARKer:RESolution:RESPonse <num>**

(Read-Write) For the Y-axis (response), sets the number digits to display after the decimal point in marker readouts.

See other SCPI Marker commands. Learn more about Marker readout.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;wnum&gt;</td>
<td>Any existing window number. If unspecified, value is set to 1.</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>Number of digits to display. Choose a value between 1 and 4.</td>
</tr>
</tbody>
</table>

**Examples**

```
DISP:WIN:ANN:MARK:RES:RESP 1
display:window:annotation:marker:resolution:stimulus 2
```

**Query Syntax**

```
DISPlay:WINDoW:ANNotation:MARKer:RESolution:RESPonse?
```

**Return Type**

Numeric

**Default**

2

**DISPlay:WINDoW<wnum>:ANNotation:MARKer:SINGle[:STATe] <bool> - Superseded**
Note: This command is replaced by **DISP:WIND:ANN:MARK:NUMB**

(Read-Write) Either shows marker readout of only the active trace or other traces simultaneously.

See other SCPI Mark commands. Learn more about Marker readout.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<bool>` **ON** (or 1) - Shows the readout of only the active marker for each trace.
  **OFF** (or 0) - Shows up to 5 marker readouts per trace, up to 20 total readouts.

**Examples**

```plaintext
DISP:WIND:ANN:MARK:SING ON
display:window:annotation:marker:single off
```

**Query Syntax**

`DISP:WIND:ANN:MARK:SINGLE?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default** OFF

**DISPlay:WINDow<wnum>:ANNotation:MARKer:SIZE <char>**

(Read-Write) Specifies the size of the marker readout text. See other SCPI Mark commands. Learn more about Marker readout.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<char>` Readout text size. Choose from: NORMal | LARGe

**Examples**

```plaintext
DISP:WIND:ANN:MARK:SIZE LARG
display:window:annotation:marker:size normal
```

**Query Syntax**

`DISP:WIND:ANN:MARK:SIZE?`

**Return Type**

Character

**Default** NORMal

**DISPlay:WINDow<wnum>:ANNotation:MARKer[:STATe] <ON | OFF>**
Specifies whether to show or hide the Marker readout (when markers are ON) on the selected window. See other SCPI Marker commands. Learn more about Marker readout.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<ON | OFF>` ON (or 1) - turns marker readout ON.

**Examples**

```plaintext
DISP:WIND:ANN:MARK ON
display:window:annotation:marker:state off
```

**Query Syntax**

DISPlay:WINDow:ANNotation:MARKer[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

**DISPlay:WINDow<wnum>:ANNotation:MARKer:SYMBol <char>**

(Read-Write) Sets the symbol to display for marker position.

See other SCPI Marker commands.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<char>` Marker symbol. Choose from:
  - TRIangle
  - FLAG
  - LINE

**Examples**

```plaintext
DISP:WIND:ANN:MARK:SYMB TRI
display:window:annotation:marker:symbol line
```

**Query Syntax**

DISPlay:WINDow:ANNotation:MARKer:SYMBol?

**Return Type**

Character

**Default**

TRIangle

**DISPlay:WINDow<wnum>:ANNotation:MARKer:XPOSition <num>**
(Read-Write) Sets the X-axis position of marker readouts. Readouts are right-justified at the specified position.

See other SCPI Marker commands. Learn more about Marker readout.

Parameters

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<num>` X-axis position. Choose a value between 1 (far left) and 10 (far right).

Examples

```
DISP:WIND:ANN:MARK:XPOS 1.5
display:window:annotation:marker:xposition 5
```

Query Syntax

```
DISPlay:WINDow:ANNotation:MARKer:XPOSition?
```

Return Type

Numeric

Default

10

-----

DISPlay:WINDow<wnum>:ANNotation:MARKer:YPOSition <num>

(Read-Write) Sets the Y-axis position of marker readouts. Readouts are top-justified at the specified position.

See other SCPI Marker commands. Learn more about Marker readout.

Parameters

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<num>` Y-axis position. Choose a value between 1 (bottom) and 10 (top).

Examples

```
DISP:WIND:ANN:MARK:YPOS 1.5
display:window:annotation:marker:yposition 5
```

Query Syntax

```
DISPlay:WINDow:ANNotation:MARKer:YPOSition?
```

Return Type

Numeric

Default

10

-----

DISPlay:WINDow<wnum>:ANNotation[:TRACE][:STATe] <ON | OFF>
(Read-Write) Specifies whether to show or hide the Trace Status buttons on the left of the display.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<ON | OFF>` **ON** (or 1) - turns the buttons ON. **OFF** (or 0) - turns the buttons OFF.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:ANN ON</td>
<td>Display:window:annotation:trace:state on</td>
</tr>
<tr>
<td>display:window:annotation:trace:state off</td>
<td></td>
</tr>
</tbody>
</table>

**Query Syntax**

`DISPlay:WINDow<wnum>:ANNotation[:TRACe][:STATe]?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

`DISPlay:WINDow<wnum>:CATalog?`

(Read-only) Returns the trace numbers for the specified window.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.

**Example**

Window 1 with four traces:

`DISPlay:WINDow1:CATalog?`  
Returns:  
"1, 2, 3, 4"

**Return Type**

String of Character values separated by commas

**Default**

Not applicable

---

`DISPlay:WINDow<wnum>:ENABle <ON | OFF>`
(Read-Write) Specifies whether to disable or enable all analyzer display information in the specified window. Marker data is not updated. More CPU time is spent making measurements instead of updating the display.

**Parameters**

- **<wnum>** Any existing window number. If unspecified, value is set to 1.
- **<ON | OFF>**
  - **ON** (or 1) - turns the display ON.
  - **OFF** (or 0) - turns the display OFF.

**Examples**

```plaintext
DISP:WIND:ENABLE ON
display:window1:enable off
```

**Query Syntax**

```
DISP:WINDow<wnum>:ENABLE?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**DISPlay:WINDow<wnum>:SIZE <char>**

(Read-Write) Sets or returns the window setting of Maximized, Minimized, or Normal. To arrange all of the windows, use **DISP:ARR**.

**Parameters**

- **<wnum>** Any existing window number. If unspecified, value is set to 1
- **<char>** Window size. Choose from:
  - **MIN | MAX | NORM**

**Examples**

```plaintext
DISP:WIND:SIZE MAX
display:window:size norm
```

**Query Syntax**

```
DISP:WINDow:SIZE?
```

**Default**

Not Applicable

---

**DISPlay:WINDow<wnum>[::STATe] <ON | OFF>**
(Read-Write) Write to create or delete a window on the screen or Read whether a window is present.

**Parameters**

- `<wnum>`  Window number to create; choose any integer between 1 and the maximum number of windows allowed in the PNA.
- `<ON | OFF>`  ON (or 1) - The window `<wnum>` is created.  
  OFF (or 0) - The window `<wnum>` is deleted.

**Examples**

```
DISP:WIND ON
display:window2:state off
```

**Query Syntax**

`DISPlay:WINDow<wnum>[:STATe]?`

**Return Type**  Boolean (1 = ON, 0 = OFF)

**Default**  Window number "1" ON

---

**DISPlay:WINDow<wnum>:TABLe <char>**

(Read-Write) Write to show the specified table at the bottom of the analyzer screen or Read to determine what table is visible.

**Parameters**

- `<wnum>`  Any existing window number. If unspecified, value is set to 1
- `<char>`  Table to show. Choose from:
  - OFF | MARKer | LIMit | SEGMent

**Examples**

```
DISP:WIND:TABLE SEGm
display:window:table off
```

**Query Syntax**

`DISPlay:WINDow:TABLE?`

**Default**  OFF

---

**DISPlay:WINDow<wnum>:TITLe:DATA <string>**
(Read-Write) Sets data in the window title area. The title is turned ON and OFF with DISP:WIND:TITL:STAT OFF.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1.
<string> Title to be displayed. Any characters, enclosed with quotes. If the title string exceeds 50 characters, an error will be generated and the title not accepted. Newer entries replace (not append) older entries.

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:TITL:DATA 'hello'</td>
<td>Display title 'hello'</td>
</tr>
<tr>
<td>display:window2:title:data 'hello'</td>
<td>Display title 'hello'</td>
</tr>
</tbody>
</table>

Query Syntax

DISPlay:WINDow<wnum>:TITLe:DATA?

Return Type

String

Default

NA

DISPlay:WINDow<wnum>:TITLe[:STATe] <ON | OFF>

(Read-Write) Turns display of the title string ON or OFF. When OFF, the string remains, ready to be redisplayed when turned back ON.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1
<ON | OFF> ON (or 1) - turns the title string ON.
OFF (or 0) - turns the title string OFF.

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:TITL ON</td>
<td>Display title ON</td>
</tr>
<tr>
<td>Display:window1:title:state off</td>
<td>Display title OFF</td>
</tr>
</tbody>
</table>

Query Syntax

DISPlay:WINDow<wnum>:TITLe[:STATe]?

Return Type

Boolean (1 = ON, 0 = OFF)

Default

ON

DISPlay:WINDow<wnum>:TRACe<tnum>:DELeTe
(Write-only) Deletes the specified trace from the specified window. The measurement parameter associated with the trace is not deleted.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` The number of the trace to be deleted; if unspecified, value is set to 1.

**Note:** This is NOT the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:TRAC:DEL</td>
</tr>
<tr>
<td>display:window2:trace2:delete</td>
</tr>
</tbody>
</table>

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:FEED <name>**

(Write-only) Creates a new trace `<tnum>` and associates (feeds) a measurement `<name>` to the specified window `<wnum>`. This command should be executed immediately after creating a new measurement with CALC:PAR:DEF `<name>,<parameter>`.

To feed the same measurement to multiple traces, create another measurement with the same `<name>,<parameter>` using the CALC:PAR:DEF command. The analyzer will collect the data only once.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Trace number to be created. Choose any Integer between 1 and the PNA maximum number of traces per window allowed.

**Note:** This is NOT the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<name>` Name of the measurement that was defined with CALC:PAR:DEF `<name>,<parameter>`

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:TRAC:FEED 'test'</td>
</tr>
<tr>
<td>display:window2:trace2:feed 'test'</td>
</tr>
</tbody>
</table>

**Query Syntax** Not applicable

**Default** "CH1_S11"
**DISPlay:WINDow:TRACe:GRATicule:GRID:LTYPE** `<value>`

(Read-Write) Sets and returns the grid line type (solid | dotted) for all open windows. Grid is returned to solid when the PNA is Preset. Learn more.

**Parameters**

- `<value>`: Line type. Choose from:
  - `SOLid` - solid lines
  - `DOTTed` - dotted lines

**Examples**

`DISP:WIND:TRAC:GREAT:GRID:LTYPE SOL`  
`display:window:trace:graticule:grid:ltype dotted`

**Query Syntax**

`DISPlay:WINDow:TRACe:GRATicule:GRID:LTYPE?`

**Return Type** Character

**Default** `SOLID`

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:MEMory[:STATe]** `<ON | OFF>`

(Read-Write) Turns the memory trace ON or OFF.

**Parameters**

- `<wnum>`: Any existing window number. If unspecified, value is set to 1.
- `<tnum>`: Any existing trace number; if unspecified, value is set to 1.

**Note:** This is NOT the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<ON | OFF>`:
  - `ON` (or 1) - turns the memory trace ON.
  - `OFF` (or 0) - turns the memory trace OFF.

**Examples**

`DISP:WIND:TRAC:MEM ON`  
`display:window2:trace2:memory:state off`

**Query Syntax**

`DISPlay:WIND<wnum>:TRACe<tnum>:MEMory[:STATe]?`

**Return Type** Boolean (1 = ON, 0 = OFF)

**Default** `OFF`

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:SELect**
(Write-only) Activates the specified trace in the specified window for front panel use.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Any existing trace number; if unspecified, value is set to 1.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

**Examples**

```
DISP:WIND:TRAC:SEL
display:window2:trace2:select
```

**Query Syntax**

Not applicable

**Default**

NA

---

**DISPlay:WINDow<wnum>:TRACe<tnum>[:STATe] <ON | OFF>**

(Read-Write) Turns the display of the specified trace in the specified window ON or OFF. When OFF, the measurement behind the trace is still active.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Any existing trace number; if unspecified, value is set to 1.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<ON | OFF>`  
  - **ON** (or 1) - turns the trace ON.  
  - **OFF** (or 0) - turns the trace OFF.

**Examples**

```
DISP:WIND:TRAC ON
display:window2:trace2:state off
```

**Query Syntax**

DISPlay:WIND<wnum>:TRACe<tnum>[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:TITLe:DATA <string>**
(Read-Write) Writes and read data to the trace title area. The trace title is embedded in the trace status field. Learn more.

Newer entries replace (not append) older entries. The title is turned ON and OFF with DISP:WIND:TRAC:TITL:STAT.

**Parameters**

<wnum> Any existing window number. If unspecified, value is set to 1.
<tnum> Trace number of the specified window. If unspecified, value is set to 1. Use Display:Cat? to read the window numbers. Use Disp:Window:Cat? to read the trace numbers of the specified window.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This <tnum> is the trace number within the specified window, and is used ONLY for remote programs.

<string> Title to be displayed. Any characters (not spaces) enclosed with quotes.

**Examples**

DISP:WIND:TRAC:TITL:DATA 'MyNewMeas'
display:window2:trace3:title:data 'hello'

**Query Syntax**

DISPlay:WINDow<wnum>:TRACe<tnum>TITLe:DATA?

**Return Type** String

**Default** Not Applicable

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:TITLe[:STATe] <bool>**

(Read-Write) Turns display of the Trace Title ON or OFF. When turned OFF, the previous trace title returns. Set a new trace title using DISP:WIND:TRAC:TITL:DATA.

Learn more about Trace Titles

**Parameters**

<wnum> Any existing window number. If unspecified, value is set to 1
<tnum> Trace number of the specified window. If unspecified, value is set to 1. Use Display:Cat? to read the window numbers. Use Disp:Window:Cat? to read the trace numbers of the specified window.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This <tnum> is the trace number within the specified window, and is used ONLY for remote programs.
<bool> **ON** (or 1) - turns the title ON.

**OFF** (or 0) - turns the title OFF.

**Examples**

```plaintext
DISP:WIND:TRAC:TITL ON
Display:window2:trace3:title:state off
```

**Query Syntax**

```
DISPlay:WINDow<wnum>:TRACe<tnum>:TITLe[:STATe]?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:AUTO**

(Write-only) Performs an Autoscale on the specified trace in the specified window, providing the best fit display.

Autoscale is performed only when the command is sent; it does NOT keep the trace autoscaled indefinitely.

Autoscale behaves differently when scale coupling is enabled. How it behaves depends on the scale coupling method. Learn more.

See Also, **DISPlay:WINDow:Y:AUTO** which performs an Autoscale All.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Any existing trace number; if unspecified, value is set to 1.

**Note:** This is NOT the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

**Examples**

```
DISP:WIND:TRAC:Y:AUTO
display:window2:trace2:y:scale:auto
```

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**DISPlay:WINDow:TRACe:Y[:SCALe]:COUPle:METHod <char>**
(Read-Write) Sets and returns the method of scale coupling. Learn more about Scale coupling.

**Parameters**

- `<char>` **OFF** - NO scale coupling for any windows.
- **WINDoW** - Scale settings are coupled for traces in each window.
- **ALL** - Scale settings are coupled for traces in ALL selected windows.

Enable the selected windows using `DISP:WIND:TRAC:Y:COUP ON`

**Examples**

```bash
```

```bash
Display:window2:trace:y:scale:method window
```

**Query Syntax**

`DISP:WIND:TRAC:Y[:SCALE]:COUPle:METHod?`

**Return Type**

Character

**Default**

OFF

---

`DISP:WIND<wnum>:TRACe<tnum>:Y[:SCALE]:PDIVision <num>`

(Read-Write) Enables and disables scale coupling for the specified window. Learn more about Scale coupling.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1
- `<bool>` **ON** (or 1) - Scale coupling enabled for specified window.
- **OFF** (or 0) - Scale coupling disabled for specified window.

**Examples**

```bash
DISP:WIND<wnum>:TRACe<tnum>:Y:COUP ON
```

```bash
Display:window2:trace:y:scale:couple:state off
```

**Query Syntax**

`DISP:WIND<wnum>:TRACe<tnum>:Y[:SCALE]:COUPle[:STATE] ?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

`DISP:WIND<wnum>:TRACe<tnum>:Y[:SCALE]:PDIVision <num>`
(Read-Write) Sets the Y axis **Per Division** value of the specified trace in the specified window.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Any existing trace number; if unspecified, value is set to 1.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<num>` Units / division value. The range of acceptable values is dependent on format and domain.

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
DISP:WIND:TRAC:Y:PDIV 1
display:window2:trace2:y:scale:pdivision maximum
```

**Query Syntax**

```
DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:PDIVision?
```

**Return Type**

Numeric

**Default**

10

---

**DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:RLEVel <num>**

(Read-Write) Sets the Y axis Reference Level of the specified trace in the specified window.

**Parameters**

- `<wnum>` Any existing window number. If unspecified, value is set to 1.
- `<tnum>` Any existing trace number; if unspecified, value is set to 1.

**Note:** This is **NOT** the trace number of the channel which appears as the Tr annotation on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<num>` Reference level value. The range of acceptable values is dependent on format and domain.

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
DISP:WIND:TRAC:Y:RLEV 0
display:window2:trace2:y:scale:rlevel minimum
```

**Query Syntax**

```
DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:RLEVel?
```
**Return Type**  Numeric  
**Default**  Not Applicable

**DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:RPOSition <num>**

(Read-Write) Sets the **Reference Position** of the specified trace in the specified window

**Parameters**

- `<wnum>`  Any existing window number. If unspecified, value is set to 1.
- `<tnum>`  Any existing trace number; if unspecified, value is set to 1.

**Note:** This is **NOT** the trace number of the channel which appears as the **Tr annotation** on the Trace Status display. This `<tnum>` is the trace number within the specified window, and is used ONLY for remote programs.

- `<num>`  Reference position on the screen measured in horizontal graticules from the bottom. The range of acceptable values is dependent on format and domain.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

- DISP:WIND:TRAC:Y:RPOS 0
- display:window2:trace2:y:rposition maximum

**Query Syntax**  
DISPlay:WINDow<wnum>:TRACe<tnum>:Y[:SCALe]:RPOSition?

**Return Type**  Numeric  
**Default**  5

**DISPlay:VISible <ON | OFF>**
(Read-Write) Makes the PNA application visible or not visible. In the Not Visible state, the analyzer cycle time for making measurements, and especially data transfer, can be significantly faster because the display does not process data.

**Parameters**

<ON | OFF>  
ON (or 1) - PNA app is visible  
OFF (or 0) - PNA app is NOT visible  

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:VIS ON</td>
<td>ON (or 1) - PNA app is visible</td>
</tr>
<tr>
<td>display:visible off</td>
<td>OFF (or 0) - PNA app is NOT visible</td>
</tr>
</tbody>
</table>

**Query Syntax**

DISPlay:VISible?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**DISPlay:WINDow<wnum>:Y:AUTO**

(Write-only) Scales ALL of the traces to fit in the same window. This is equivalent to "Autoscale All" from the front panel.

Autoscale behaves differently when scale coupling is enabled. How it behaves depends on the scale coupling method. [Learn more.](#)

Autoscale is performed only when the command is sent; it does NOT keep the trace autoscaled indefinitely.

See Also, **DISPlay:WINDow:TRACe:Y:AUTO** which Autoscales only the specified trace.

**Parameters**

<wnum>  
Any existing window number. If unspecified, value is set to 1.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISP:WIND:y:AUTO</td>
<td>Autoscales ALL traces</td>
</tr>
<tr>
<td>display:window2:y:auto</td>
<td>Autoscales only window 2</td>
</tr>
</tbody>
</table>

**Query Syntax**

Not applicable

**Default**

Not applicable

---

Last modified:
11-Jan-2011  Minor edits
15-Sep-2010  Added autoscale all and scale coupling note.
  6-Aug-2010  Added marker readout (A.09.30)
  16-Mar-2010  Added grid line (9.2)
  23-Mar-2009  Added <tnum> note
  26-Aug-2008  Added TraceMax
  12-Sep-2006  Modified for number of windows.
Format Commands

Specifies the way that data will be transferred when moving large amounts of data.

```
FORMat

BORDer [DATA]
```

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

**FORMat:BORDer <char>**

*(Read-Write)* Set the byte order used for GPIB data transfer. Some computers read data from the analyzer in the reverse order. This command is only implemented if FORMAT:DATA is set to :REAL.

If FORMAT:DATA is set to :ASCII, the swapped command is ignored.

**Parameters**

- `<char>` Choose from:
  - **NORMal** - Use when your controller is anything other than an IBM compatible computers.
  - **SWAPped** - for IBM compatible computers.

**Note:** Use NORMal if you are using VEE, LabView, or T&M Tool kit.

**Examples**

```
FORM:BORD SWAP
format:border normal
```

**Query Syntax**

```
FORMat:BORDer?
```

**Return Type**

Character

**Default** Normal

**FORMat[:DATA] <char>**
(Read-Write) Sets the data format for data transfers.

- To transfer measurement data, use **CALC:DATA**.
- To transfer Cal Set data, use **SENS:CORR:CSET:DATA**
- To transfer Source Power correction data, use:
  - **SOURce:POWer:CORRection:COLLect:TABLE:DATA**
  - **SOURce:POWer:CORRection:COLLect:TABLE:FREQuency**
  - **SOURce:POWer:CORRection:DATA**

**Parameters**

<char> Choose from:

- **REAL,32** - (default value for REAL) Best for transferring large amounts of measurement data.
- **REAL,64** - Slower but has more significant digits than REAL,32. Use REAL,64 if you have a computer that doesn't support REAL,32.
- **ASCii,0** - The easiest to implement, but very slow. Use when you have small amounts of data to transfer.

**Notes:**

- The higher frequencies used on PNA can exceed the maximum value that can be represented by a 32-bit floating point number.
- The **REAL,32** and **REAL,64** arguments transfer data in block format as explained in Transferring Measurement Data.

**Examples**

```
FORM REAL, 64
format: data ascii
```

**Query Syntax**  FORMat:DATA?

**Return Type**  Character,Character

**Default**  ASCii,0

Syst:Preset does NOT reset this command.
However, *RST does reset this command to ASCII,0

---

Last Modified:
18-May-2009  Added Real 64 note
17-Sep-2008  Added *RST vs Syst:Pres note.
Hardcopy Command

Controls printing of the PNA screen and optional data to a printer or a file.

HCOPY:
  DPRinter
  FILE
  [IMMediate]

ITEM
  | AWINdow
  | CTABle
  | GPFail
  | LOGO
  | MKRData
  | PNUMber
  | SEGData
  | SWINdow
  | TIME
  | TTABle
  | WFRaction
  | WINDows

PAGE
  | DIMension
  | LLEFt
  | URIGht
  | ORIentation
  | SIZE

SDUMP
  | DATA?
  | FORMat

PRINters?

Click on a blue keyword to view the command details.

Red commands are superseded or obsolete.

See Also

- Learn more about PNA Printing
- Example Programs
### HCOPy:DPRinter <string>

*(Read-Write)* Sets the default printer and selects as the current printer. Use HCOPy:PRINters? to return a list of locally installed printers.

This setting survives instrument preset and PNA application restart.

**Parameters**

- `<string>`: Name of the printer to become the default.

**Examples**

- HCOP: DPR "MyPrinter"
- hcopy: dprinter "YourPrinter"

**Query Syntax**

HCOPy:DPRinter?

**Return Type**

String

**Default**

Not Applicable

---

### HCOPy:FILE <filename>

*(Write-only)* Saves the screen image to a file. The image does NOT include the optional print data invoked by many HCOPy commands.

**Parameters**

- `<filename>`: Name of the file to save the screen to. The file is saved to the current working directory unless a valid full path name is specified.

Use one of the following suffixes:

- .bmp - not recommended due to large file size
- .jpg - not recommended due to poor quality
- .png - recommended

**Examples**

- HCOPY:FILE "myFile.png"
- hcopy:file "c:/data/myfile.png"

**Query Syntax**

Not Applicable

**Default**

Not Applicable
HCOPy[:IMMediate]

(Write-only) Prints the screen to the default printer.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>HCOP</td>
</tr>
<tr>
<td>hcopy:immediate</td>
</tr>
</tbody>
</table>

**Query Syntax**

Not applicable

**Default**

Not Applicable

HCOPy:ITEM:AWINdow[:STATe] <bool>

(Read-Write) When ON, prints only the Active window. When OFF, prints all windows.

This setting survives instrument preset and PNA application restart.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;bool&gt;</td>
</tr>
<tr>
<td>Active window state. Chose from:</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>OFF or (0) - Print ALL windows.</td>
</tr>
<tr>
<td>ON or (1) - Print Active window only.</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>HCOP</td>
</tr>
<tr>
<td>hcopy:item:awindow:state off</td>
</tr>
</tbody>
</table>

**Query Syntax**

HCOPy:ITEM:AWINdow[:STATe]?

**Return Type**

Boolean

**Default**

OFF (0)

HCOPy:ITEM:CTABLE[:STATe] <bool>
(Read-Write) When ON, prints the channel settings table.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Channel table print state. Chose from:

- **OFF** or (0) - Does NOT print the channel settings table.
- **ON** or (1) - Prints channel settings table.

**Examples**

```
HCOP:ITEM:CTAB 1
hcopy:item:ctable:state off
```

**Query Syntax**

```
HCOPy:ITEM:CTABle[:STATe]
```

**Return Type**

Boolean

**Default**

OFF (0)

---

**HCOPy:ITEM:GPFail[:STATe] <bool>**

(Read-Write) When ON, prints the **Global Pass/Fail** status in the page header.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Pass / Fail print state. Chose from:

- **OFF** or (0) - Does NOT print Pass / Fail status.
- **ON** or (1) - Print Pass / Fail status

**Examples**

```
HCOP:ITEM:GPF 1
hcop:item:gpfail:state off
```

**Query Syntax**

```
HCOPy:ITEM:GPFail[:STATe]
```

**Return Type**

Boolean

**Default**

OFF (0)

---

**HCOPy:ITEM:LOGO[:STATe] <bool>**
When ON, prints the Agilent Technologies logo in the page header.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Agilent logo print state. Chose from:

- **OFF** or (0) - Prints the Agilent logo.
- **ON** or (1) - Does NOT print the Agilent logo.

**Examples**

```
HCOP:ITEM:LOGO 1
hcop:item:logo:state off
```

**Query Syntax**

```
HCOPy:ITEM:LOGO[:STATe]?
```

**Return Type**

Boolean

**Default**

OFF (0)

---

**HCOPy:ITEM:MKRData[:STATe] <bool>**

(Read-Write) When ON, includes marker data as part of the [trace attributes table](#).

To print marker data, **HCOP:ITEM:TTABle** must also be set to ON.

This setting does not affect the limited [marker readout data](#) that can be displayed in the measurement window.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Marker data print state. Chose from:

- **OFF** or (0) - Does NOT print Marker data.
- **ON** or (1) - Print Marker data.

**Examples**

```
HCOP:ITEM:MKRD 1
hcop:item:mkrdata:state off
```

**Query Syntax**

```
HCOPy:ITEM:MKRData[:STATe]?
```

**Return Type**

Boolean

**Default**

OFF (0)
HCOPy:ITEM:PNUMber[:STATE] <bool>

(Read-Write) When ON, prints page numbers (1 of n) in the header at the top of each page.

This setting survives instrument preset and PNA application restart.

Parameters

<bool> Page number print state. Chose from:

  OFF or (0) - Does NOT print page numbers.

  ON or (1) - Print page numbers.

Examples

HCOP:ITEM:PNUM 1
hc:copy:item:pnumber:state off

Query Syntax

HCOPy:ITEM:PNUMber[:STATE]

Return Type

Boolean

Default

OFF (0)

HCOPy:ITEM:SEGData[:STATE] <bool> - Obsolete

Note: This command no longer works beginning with A.09.40

(Read-Write) When ON, includes ALL segment data as part of the channel settings table.

To print ALL segment data, HCOP:ITEM:CTAB must also be set to ON.

This setting survives instrument preset and PNA application restart.

Parameters

<bool> Expanded segment data print state. Chose from:

  OFF or (0) - Does NOT print expanded segment data, but summary data is printed.

  ON or (1) - Print expanded segment data.

Examples

HCOP:ITEM:SEG 1
hc:copy:item:segdata:state off

Query Syntax

HCOPy:ITEM:SEGData[:STATE]

Return Type

Boolean

Default

OFF (0)
HCOPY:ITEM:SWINdow[:STATe] <bool>

(Read-Write) When ON, prints a single measurement window per page. When OFF, prints up to four measurement windows per page.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Single window print state. Chose from:

OFF or (0) - Print up to four windows per page.

ON or (1) - Print only one window per page.

**Examples**

HCOP:ITEM:SWIN 1
hcop:item:swindow:state off

**Query Syntax**

HCOPY:ITEM:SWIN[:STATe]?

**Return Type**

Boolean

**Default**

OFF (0)

HCOPY:ITEM:TIME[:STATe] <bool>

(Read-Write) When ON, prints the PNA computer date and time in the header.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Time stamp print state. Chose from:

OFF or (0) - Does NOT print time stamp.

ON or (1) - Print time stamp.

**Examples**

HCOP:ITEM:TIME 1
hcop:item:time:state off

**Query Syntax**

HCOPY:ITEM:TIME:[STATe]?

**Return Type**

Boolean

**Default**

OFF (0)

HCOPY:ITEM:TTABle[:STATe] <bool>
(Read-Write) When ON, prints the trace attributes table.

This setting survives instrument preset and PNA application restart.

**Parameters**

<bool> Trace attributes table print state. Chose from:

- **OFF** or (0) - Does NOT print the trace attributes table.
- **ON** or (1) - Print the trace attributes table.

**Examples**

```
HCOP:ITEM:TTABle 1
hcopen:ITEM:ttable:state off
```

**Query Syntax**

```
HCOPY:ITEM:TTABle[:STATe]?
```

**Return Type**

Boolean

**Default**

OFF (0)

---

**HCOPY:ITEM:WFRaction <value>**

(Read-Write) Sets the vertical amount of a page that is filled by the measurement windows.

This setting survives instrument preset and PNA application restart.

**Parameters**

<value> Window size as a fraction of the page. Chose a value from .4 (40%) to 1.0 (100%)

**Examples**

```
HCOP:ITEM:WFR .8
hcopen:ITEM:wfraction .5
```

**Query Syntax**

```
HCOPY:ITEM:WFRaction?
```

**Return Type**

Numeric

**Default**

.4

---

**HCOPY:ITEM:WINDows[:STATe] <bool>**
(Read-Write) When ON, prints measurement windows.

Use `HCOPy:ITEM:AWIn Dow` to specify all windows or only the active window.

This setting survives instrument preset and PNA application restart.

**Parameters**

- `<bool>` Windows print state. Choose from:
  - **OFF** or (0) - Does not print measurement windows.
  - **ON** or (1) - Print measurement windows.

**Examples**

- `HCOP:ITEM:WIND 1`
- `hcopy:item:windows:state off`

**Query Syntax**

`HCOPy:ITEM:WINDows[:STATe]?

**Return Type**

Boolean

**Default**

OFF (0)

---

**HCOPy:PAGE:DIMensions:LLEFt <left, lower>**

(Read-Write) Sets the left and lower page margins.

This setting survives instrument preset and PNA application restart.

**Parameters**

- `<left>` Left page margin as a percentage of entire page width. Value must be between 0 and 1.
- `<lower>` Lower page margin as a percentage of entire page length. Value must be between 0 and 1.

**Examples**

- `HCOP:PAGE:DIM:LLEF .10,.10`
- `hcopy:page:dimensions:lleft .5,.7`

**Query Syntax**

`HCOPy:PAGE:DIMensions:LLEFt?`

**Return Type**

Numeric, Numeric

**Default**

Depends on selected page size

---

**HCOPy:PAGE:DIMensions:URIGht <right, upper>**
(Read-Write) Sets the right and upper page margins.

This setting survives instrument preset and PNA application restart.

**Parameters**

- **<right>** Right page margin as a percentage of entire page width. Value must be between 0 and 1.
- **<upper>** Upper page margin as a percentage of entire page length. Value must be between 0 and 1.

**Examples**

```
HCOP:PAGE:DIM:URIG .10,.10
hcopy:page:dimensions:uright .5,.7
```

**Query Syntax**

```
HCOPy:PAGE:DI Mensions:URIGht?
```

**Return Type** Numeric, Numeric

**Default** Depends on selected page size

---

**HCOPy:PAGE:ORIentation <char>**

(Read-Write) Sets the page orientation.

This setting survives instrument preset and PNA application restart.

**Parameters**

- **<char>** Choose from:
  - PORTrait
  - LANDscape

**Examples**

```
HCOP:PAGE:ORI PORT
hcopy:page:orientation landscape
```

**Query Syntax**

```
HCOPy:PAGE:ORIentation?
```

**Return Type** Character

**Default** PORTrait

---

**HCOPy:PAGE:SIZE <int>**
(Read-Write) Sets the paper type, which implies the page size.

This setting survives instrument preset and PNA application restart.

**Parameters**

Choose from:

<table>
<thead>
<tr>
<th>Integer</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Letter 8 1/2 x 11 in</td>
</tr>
<tr>
<td>2</td>
<td>Letter Small 8 1/2 x 11 in</td>
</tr>
<tr>
<td>3</td>
<td>Tabloid 11 x 17 in</td>
</tr>
<tr>
<td>4</td>
<td>Ledger 17 x 11 in</td>
</tr>
<tr>
<td>5</td>
<td>Legal 8 1/2 x 14 in</td>
</tr>
<tr>
<td>6</td>
<td>Statement 5 1/2 x 8 1/2 in</td>
</tr>
<tr>
<td>7</td>
<td>Executive 7 1/4 x 10 1/2 in</td>
</tr>
<tr>
<td>8</td>
<td>A3 297 x 420 mm</td>
</tr>
<tr>
<td>9</td>
<td>A4 210 x 297 mm</td>
</tr>
<tr>
<td>10</td>
<td>A4 Small 210 x 297 mm</td>
</tr>
<tr>
<td>11</td>
<td>A5 148 x 210 mm</td>
</tr>
<tr>
<td>12</td>
<td>B4 (JIS) 250 x 354</td>
</tr>
<tr>
<td>13</td>
<td>B5 (JIS) 182 x 257 mm</td>
</tr>
</tbody>
</table>

For more paper type choices, see Microsoft's "wingdi.h" file, which can be downloaded as part of the Platform SDK.

**Examples**

HCOP:PAGE:SIZE 2
hcop:page:size 5

**Query Syntax**

HCOPy:PAGE:SIZE?

**Return Type**

Integer

**Default**

1
Returns the display image in a definite-length arbitrary binary block. The format of the data is PNG by default. Use `HCOP:SDUMp:DATA:FORMat` to change the format.

This command is equivalent to saving an image to the PNA (`HCOPy:FILE`) and then using `MMEM:TRAN` to transfer the file to the computer.

<table>
<thead>
<tr>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>A definite-length arbitrary binary block</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**Examples**

```
HCOP : SDUM?
hcopy : sdump?
```

**HCOPy:SDUMp:DATA:FORMat <char>**

(Read-Write) Sets the graphic format for `HCOPy:SDUMp:DATA`?

<table>
<thead>
<tr>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>&lt;char&gt;</code></td>
</tr>
</tbody>
</table>

Choose from: JPG | BMP | PNG

<table>
<thead>
<tr>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>HCOP : SDUMP : DATA : FORMAT BMP</td>
</tr>
</tbody>
</table>

**Query Syntax**

HCOPy:SDUMp:DATA:FORMat?

**Return Type**

Character

**Default**

PNG

**HCOPy:PRINters?**

(Read-only) Returns a comma-separated list of printers installed on the PNA. Select a printer using `HCOPy:DPRinter`.

This setting survives instrument preset and PNA application restart.

<table>
<thead>
<tr>
<th>Query Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>HCOPy:PRINters?</td>
</tr>
</tbody>
</table>

**Return Type**

String

**Default**

Not Applicable

Last modified:
13-Apr-2011 Obsolete seg data
15-Mar-2010 Added SDUMP commands (A.09.20)
Nov. 1, 2006 Added new commands
### Initiate Commands

Controls triggering signals

---

**INITiate**

**CONTinuous** [IMMediate]

Click on a blue keyword to view the command details.

**See Also**

- Example [Triggering the PNA](#)
- [Learn about Triggering](#)
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

---

**INITiate:CONTinuous <boolean>**

*(Read-Write)* Specifies whether the PNA trigger source is set to Internal (continuous) or Manual.

This command is a subset of [TRIG:SEQ:SOURce](#), which can also set the trigger source to External.

To set how a channel responds to trigger signals, use [SENS:SWE:MODE](#).

See a map of user interface to SCPI triggering commands.

For more information on triggering, see the [PNA Trigger Model](#).

**See the Example program:** [Triggering the PNA using SCPI](#)

**Parameters**

- **<boolean>**
  - **ON** (or 1) - Internal (continuous) trigger.
  - **OFF** (or 0) - Manual sweep. Use [INIT:IMMediate](#) to send a trigger signal

**Examples**

- **INIT:CONT ON**
- **initiate:continuous off**

**Query Syntax**

INITiate:CONTinuous?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON
INITiate<cnum>[:IMMediate]

(Write-only) Stops the current sweeps and immediately sends a trigger. (Same as Trigger! on the PNA front panel).

See the Example program: Triggering the PNA using SCPI

**Note:** An SMC Fixed Output measurement cannot be triggered using this command. For more information, see the example program.

This command requires Trigger:Source to be set to Manual. This causes ONE trigger signal to be sent each time INIT:IMM is issued.

Sens<ch>:Sweep:Mode sets the number of trigger signals each channel will accept (Continuous - unlimited, Groups - a specific number, or HOLD - none.)

**To trigger ALL channels in turn:**

Set ALL channels to Sens<ch>:Sweep:Mode Continuous. The <ch> argument in INIT<ch>:IMM is ignored.

Then…

- TRIG:SCOP ALL triggers ALL channels (in sequence) each time Init:Imm is sent.
- TRIG:SCOP CURRent triggers ONLY the NEXT channel each time Init:Imm is sent.

**To trigger ONLY a specified channel:**

1. Set ALL channels to Sens<ch>:Sweep:Mode HOLD
2. Send TRIG:SCOP CURRent
3. Send Init<ch>:Imm where <ch> is the channel to be triggered.

**Advanced** Situations that require some channels to be in CONT and others in HOLD are rare. The following describes the behavior of the Init:Imm command in these situations:

**When Trigger:Scope = Global:**

- If the SPECIFIED <cnum> channel is in hold mode, it is put in single trigger (accepts 1 trigger signal) and goes to the end of the queue of channels to be triggered. The other 'non-hold' channels are triggered. The next Init:Imm triggers the specified channel first.

  For example: ch1 is in Hold, ch2 and ch3 are in CONT and we send INIT1:IMM
On the first INIT:IMM, ch2 and ch3 is triggered.

next INIT:IMM, ch1, ch2, ch3 is triggered.

next INIT:IMM, ch2 and ch3 is triggered.

next INIT:IMM, ch1, ch2, ch3 is triggered, and so forth.

**When Trigger:Scope = Channel**

- Only ONE channel is triggered for each issued INIT<ch>:IMM command.

- If the specified channel is in hold, it is put in single trigger (accepts 1 trigger signal) and goes the end of the queue of channels to be triggered as in the 'Global' example.

This is one of the PNA overlapped commands. [Learn more.](#)

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>&lt;chnum&gt;</code></td>
<td>Any existing channel number. If unspecified, value is set to 1</td>
</tr>
</tbody>
</table>

**Examples**

```
INIT
initiate2:immediate
```

**Query Syntax**

- Not applicable

**Default**

- Not applicable

Last modified:

April 23, 2007    Updated Init:Imm
The memory commands control saving and loading instrument states and measurement trace data to the hard drive. To read and write trace data in GPIB format, see CALC:DATA.

Click on a blue keyword to view the command details.

Red commands are superseded.

See Also

- Example Programs
- Learn about Save / Recall and File Types
- Synchronizing the PNA and Controller
- SCPI Command Tree
Specifying Path Names

The MMEM commands use the following rules to specify path names:

- The default folder is "C:/Program Files/Agilent/Network Analyzer/Documents"
- You can change the active directory using MMEMory:CDIRectory.
- Specify only the file name if using the active directory.
- You can also use an absolute path name to specify the folder and file.

**MMEMory:CATalog[:<char>]? [<folder>]**

(Read-only) Returns a comma-separated string of file names that are in the specified folder. If there are no files of the specified type, "NO CATALOG" is returned. [Learn about File Types](#)

**Parameters**

- **<char>** The type of files to list. Choose from:
  - **STATe** - Instrument states (.sta)
  - **CORRection** - Calibration Data (.cal)
  - **CSARchive** - Instrument state and calibration data (.csa)
  - **CSTate** - Instrument state and link to Calibration data (.cst)

  If unspecified then ALL file types (even unknown types) are listed.

- **<folder>** String - Any existing folder name. See [Specifying Path Names](#)

**Examples**

- `MMEM:CAT?` 'lists all files from the current folder
- `mmemory:catalog:correction?` 'C:/Program Files/Agilent/Network Analyzer/Documents' 'lists .cal files from the specified folder

**Default** Not applicable

**MMEMory:CDIRectory <folder>**
(Read-Write) Changes the folder name.

**Parameters**

- `<folder>`: Any drive and folder name that already exists. If the same level as "C:/Program Files/Agilent/Network Analyzer/Documents", then no punctuation is required.

**MMEM:CDIR Service**

If the new folder is at a different level than "C:/Program Files/Agilent/Network Analyzer/Documents", use a slash (/) before the folder name and enclose in quotes.

```mmemory:cdirectory '/automation' 'changes default directory up one level.'```

You can use an absolute path to specify the new folder.

```mmemory:cdirectory 'c:/automation/service'```

**Query Syntax**

`MMEMory:CDIRectory?` Returns the current folder name

**Return Type**

String

**Default**

'C:/Program Files/Agilent/Network Analyzer/Documents'

---

**MMEMory:COPY <file1>,<file2>**

(Write-only) Copies file1 to file2. Extensions must be specified.

**Parameters**

- `<file1>`: String - Name of the file to be copied. See [Specifying Path Names](#).
- `<file2>`: String - Name of the file to be created from file1.

**Examples**

```MMEM:COPY 'MyFile.cst','YourFile.cst'```

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**MMEMory:DELete <file>**
(Write-only) Deletes file. Extensions must be specified.

**Parameters**

<file> String - Name of the file to be deleted. See [Specifying Path Names](#)

**Examples**

MMEM:DEL 'MyFile.cst'

**Query Syntax** Not applicable

**Default** Not applicable

---

**MMEMory:LOAD[:<char>] <file>**

(Write-only) Loads the specified file. [Learn about File Types](#)

**Parameters**

<char> The type of file to load. Choose from:

- **STATe** - Instrument states (.sta)
- **CORRection** - Calibration Data (.cal)
- **CSARchive** - Instrument state and calibration data (.csa)
- **CSTate** - Instrument state and link to Calibration data (.cst)
- **ENR** - Excess Noise Source data ([Noise Figure App only](#))

  - When <char> is **ENR**, then include **CAL**, - See example below.

  - "*.sNp files CAN be recalled to the PNA although no <char> is used. See example below.

If <char> is unspecified, the extension must be included in the filename.

If an extension is specified in <file> that does not agree with <char> then no action is taken.

<file> String - Name of the file to be loaded. See [Specifying Path Names](#)

**Examples**

MMEM:LOAD 'MyFile.cst'

mmemory:load:state 'MyInstState'

MMEM:LOAD:ENR CAL, C:/data/calset/346C_16500.enr"

MMEM:LOAD "MyFile.s2p"

**Query Syntax** Not applicable

**Default** Not applicable
**MMEMory:MDIRectory** <folder>

(Write-only) Makes a folder.

**Parameters**

- `<folder>` String - Name of the folder to make. See [Specifying Path Names](#).

**Examples**

- `MMEM:MDIR 'MyFolder'
- `mmemory:mdirectory 'c:/NewFolder'

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**MMEMory:MOVE** <file1>,<file2>

(Write-only) Renames `<file1>` to `<file2>`. File extensions must be specified.

**Parameters**

- `<file1>` String - Name of the file to be renamed. See [Specifying Path Names](#).
- `<file2>` String - Name of the new file.

**Examples**

- `MMEM:MOVE 'MyFile.cst','YourFile.cst'

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**MMEMory:RDIrectory** <folder>

(Write-only) Removes the specified folder.

**Parameters**

- `<folder>` String - Name of the folder to remove. See [Specifying Path Names](#).

**Examples**

- `MMEM:RDIR 'MyFolder'

**Query Syntax**

Not applicable

**Default**

Not applicable

---

**MMEMory:STORe[:<char>]** <file>

---

2418
(Write-only) Saves instrument / calibration state files.

To save data files, use `MMEM:STOR:DATA`.

To save ENR files, use `MMEMory:STORe:ENR`

**Parameters**

<char> Optional argument. The type of file to store. Choose from:

- **STATE** - Instrument states (.sta)
- **CORRection** - Calibration Data (.cal)
- **CSARchive** - Instrument state and calibration data (.csa)
- **CSTate** - Instrument state and link to Calibration data (.cst)

Include either <char> or the file extension. If both <char> and the extension are specified, they must agree or an error is returned and no action is taken. See examples below.

**Learn about File Types**

=file> String - Name of any valid file that does not already exist. See [Specifying Path Names](#).

**Examples**

```
MMEM:STOR:STAT 'myState'
mmemory:store 'c:/bin/myState.sta'
```

**Query Syntax**

Not applicable

**Default**

Not applicable

---

`MMEMory:STORe:CITifile:DATA <filename>` - Superseded
This command is replaced with `MMEMory:STORe:DATA`.

(Write only) Saves UNFORMATTED trace data to .cti file. Learn more.

**Parameters**

*<filename>* Any drive and folder name that already exists.
If the same level as "C:/Program Files/Agilent/Network Analyzer/Documents", then no punctuation is required

```
MMEM:STOR:CIT:DATA MYCTIFile
```

If the new folder is at a different level than "C:/Program Files/Agilent/Network Analyzer/Documents", use a slash (/) before the folder name and enclose in quotes.

```
memory:cdirectory '/automation' 'changes default directory up one level.'
```

You can use an absolute path to specify the new folder.

```
memory:cdirectory 'c:/automation/service'
```

**Query Syntax** Not Applicable

**Default** C:/Program Files/Agilent/Network Analyzer/Documents'

---

This command is replaced with `MMEMory:STORe:DATA`.

(Write only) Saves FORMATTED trace data to .cti file. Learn more.

**Parameters**

*<filename>* Any drive and folder name that already exists.
If the same level as "C:/Program Files/Agilent/Network Analyzer/Documents", then no punctuation is required

```
MMEM:STOR:CIT:DATA MYFile
```

If the new folder is at a different level than "C:/Program Files/Agilent/Network Analyzer/Documents", use a slash (/) before the folder name and enclose in quotes.

```
memory:cdirectory '/automation' 'changes default directory up one level.'
```

You can use an absolute path to specify the new folder.
**MMEMory:STORe:DATA <filename>,<type>,<scope>,<format>,<selector>**

*(Write-only)* Stores trace data to the following file types: *.prn, *.cti, *.csv, *.mdf

To save SNP files, use **Calc:Data:SNP:PORTs:SAVE**

To save state and calibration files, use **MMEM:STORe**

This command replaces the following:

- **MMEMory:STORe:CITifile:DATA**
- **MMEMory:STORe:CITifile:FORMat**
- **MMEMory:STORe:TRACe:FORMat:CITifile**
- **MMEMory:STORe:TRACe:CONTent:CITifile**

**Parameters**

- <filename>  (String) Name of the file in which data will be saved.  
  
  See rules for specifying a filename.

Choose from the following valid parameter combinations for ALL measurement classes:

<table>
<thead>
<tr>
<th>Type of file to save</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>&lt;type&gt; (String)</td>
</tr>
<tr>
<td>* .prn</td>
<td>&quot;PRN Trace Data&quot;</td>
</tr>
<tr>
<td>* .cti</td>
<td>&quot;Citifile Data&quot;</td>
</tr>
</tbody>
</table>

Example: MMEMory:STORe:DATA "myData.prn","PRN Trace Data","Trace","Displayed",2
Example: MMEMory:STORe:DATA "myData.cti","Citifile Data Data","AUTO","RI",3

<table>
<thead>
<tr>
<th>*.cti (unformatted)</th>
<th>&quot;Citifile Data Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot;</th>
<th>-1</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.cti","Citifile Data Data","AUTO","RI",3

<table>
<thead>
<tr>
<th>*.cti (formatted)</th>
<th>&quot;Citifile Formatted Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>*(formatted)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Example: MMEMory:STORe:DATA "myData.cti","Citifile Formatted Data","AUTO","MA",3

<table>
<thead>
<tr>
<th>*.cti (formatted)</th>
<th>&quot;Citifile Formatted Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.cti","Citifile Formatted Data","DISPLAYED","MA",-1

<table>
<thead>
<tr>
<th>*.csv</th>
<th>&quot;CSV Formatted Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.csv","CSV Formatted Data","Trace","DB",3

<table>
<thead>
<tr>
<th>*.csv</th>
<th>&quot;CSV Formatted Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot; or &quot;MA&quot; or &quot;DB&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.csv","CSV Formatted Data","displayed","RI",-1

<table>
<thead>
<tr>
<th>*.mdf</th>
<th>&quot;MDIF Data&quot;</th>
<th>&quot;Trace&quot; or &quot;Auto&quot;</th>
<th>&quot;RI&quot; or &quot;Displayed&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.mdf","MDIF Data","trace","displayed",1

<table>
<thead>
<tr>
<th>*.mdf</th>
<th>&quot;MDIF Data&quot;</th>
<th>&quot;Displayed&quot;</th>
<th>&quot;RI&quot; or &quot;Displayed&quot;</th>
<th>Measurement number</th>
</tr>
</thead>
</table>

Example: MMEMory:STORe:DATA "myData.mdf","MDIF Data","displayed","displayed",-1

Notes (for above file types)

Use Calc:Par:MNUM? to read the measurement number of the selected trace.

Scope:

"Trace" - specified measurement number only.

"Displayed" - all displayed measurements.

"Auto" - for all Standard Meas Class (S-parameter) channels:
When correction is OFF, saves the specified trace

When correction is ON, saves all corrected parameters associated with the calibrated ports in the Cal Set.

"Auto" - for all other channels:

When correction is OFF or ON, saves the specified trace

The following parameter combinations save *.csv files in specific formats for GCA and Swept IMD classes:

<table>
<thead>
<tr>
<th>Parameters</th>
<th>&lt;type&gt; (String)</th>
<th>&lt;scope&gt; (String)</th>
<th>&lt;format&gt; (String)</th>
<th>&lt;selector&gt; (Numeric)</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCA channels ONLY:</td>
<td>&quot;GCA Sweep Data&quot;</td>
<td>&quot;Auto&quot;</td>
<td>&quot;DB&quot;</td>
<td>GCA channel number</td>
</tr>
</tbody>
</table>

**Learn more**

Example: MMEMory:STORe:DATA "myData.csv", "GCA Sweep Data", "Auto", "db", 1

<table>
<thead>
<tr>
<th>Swept IMD channels ONLY:</th>
<th>&quot;IMD Sweep Data&quot;</th>
<th>&quot;Auto&quot;</th>
<th>&quot;DB&quot;</th>
<th>Swept IMD channel number</th>
</tr>
</thead>
</table>

**Learn more**

Example: MMEMory:STORe:DATA "myData.csv", "IMD Sweep Data", "Auto", "db", 1

**Query Syntax** Not applicable

**Default** Not applicable

**MMEMory:STORe:ENR CAL, <file>**

*(Write-only)* Stores an ENR (Excess Noise Source) data *(Noise Figure App only)*

**Parameters**

<file> String - Name of any valid file that is not already in existence. See **Specifying Path Names**

**Examples**

```
MMEM:STOR:ENR CAL, C:/data/calset/346C_16500.enr
```

**Query Syntax** Not applicable

**Default** Not applicable
**MMEMory:STORe:TRACe:FORMat:CITifile <char> - Superseded**

This command is replaced with MMEMory:STORe:DATA.

(Read-Write) Specifies the format of subsequent citifile save statements.

**Parameters**

- `<char>` Format in which the citifile will be saved with subsequent MMEMory:STORe:CIT:FORMat statements. Choose from:
  - **MA** - Linear Magnitude / degrees
  - **DB** - Log Magnitude / degrees
  - **RI** - Real / Imaginary
  - **AUTO** - Format in which the trace is already displayed. If other than Log Mag, Linear Magnitude, or Real/Imag, then the format will be in Real/Imag.
  - **DISP** - Displayed format.

**Examples**

```
MMEM:STOR:TRAC:FORM:CIT MA
```

**Query Syntax** MMEMory:STORe:TRACe:FORMat:CITifile?

**Return Type** Character

**Default** Auto

---

**MMEMory:STORe:TRACe:CONTents:CITifile <char> - Superseded**

This command is replaced with MMEMory:STORe:DATA.

(Read-Write) Specifies the contents of subsequent citifile save statements. (See Data Define Saves)

**Parameters**

- `<char>` Choose from:
  - **SING** - Single trace
  - **DISP** - All displayed traces
  - **AUTO** - All displayed traces

**Examples**

```
MMEM:STOR:TRAC:CONT:CIT SING
```

**Query Syntax** MMEMory:STORe:TRACe:CONTents?

**Return Type** Character
**MMEMory:STORe:TRACe:FORMat:SNP <char>**

*(Read-Write)* Specifies the format of subsequent .s1p, .s2p, .s3p; s4p save statements. Learn more.

To save SNP data, use **CALC:DATA:SNP:PORTs:SAVE**

**Parameters**

- **<char>** Choose from:
  - **MA** - Linear Magnitude / degrees
  - **DB** - Log Magnitude / degrees
  - **RI** - Real / Imaginary
  - **AUTO** - data is output in currently selected trace format. If other than LogMag, LinMag, or Real/Imag, then output is in Real/Imag.

**Examples**

```
MMEM:STOR:TRAC:FORM:SNP MA
```

**Query Syntax**

`MMEMory:STORe:TRACe:FORMat:SNP?`

**Return Type**

Character

**Default** Auto

---

**MMEMory:TRANsfer <fileName>,<dataBlock>**

*(Read-Write)* Transfers data between the PNA and an external controller. Other MMEM commands transfer data between the PNA application and the PNA hard drive. If `<fileName>` already exists, it will be overwritten. The file must be no larger than 20MB.

To read **trace data** from the PNA in block format, use **CALC:DATA**.

**Parameters**

- **<fileName>** String - File name. See Specifying Path Names
- **<dataBlock>** Block Data - The contents of the file.

The data block is a block of binary data. Use the following syntax:

```
#<num digits><byte count><data bytes><NL><END>
```

where:
<num_digits> specifies how many digits are contained in <byte_count>

<byte_count> specifies how many data bytes will follow in <data bytes>

Example  See example program

Query Syntax  MMEMory:TRANsfer? <fileName>

               Reads block data from the specified file location.

Default  Not applicable

Last modified:

2-Mar-2010   Updated for csv and mdif formats (9.2)
30-Oct-2008  Fixed noise keywords for load and store.
30-Jul-2007  Added noise keywords to load and store.
9/12/06      MQ Store command has reference to PORTS Snp.
Output Commands

Controls two output functions: RF power and Noise Source.

```
OUTPut:
  | MANual:NOISe[:STATe]
  | [:STATe]
```

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

**OUTPut:MANual:NOISe[:STATe] <bool>**

*(Read-Write)* Sets and reads the noise source *(28V)* ON or OFF.

**Parameters**

<bool>  
  ON (1) - Noise source ON  
  OFF (0) - Noise source OFF

**Examples**

```
OUTP:MAN:NOIS 0
output:manual:noise:state 1
```

**Query Syntax**

OUTPut:MANual:NOISe[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

For PNA models with a Noise Figure option (028/029/H29), the 28V line is always ON. The ON/OFF state is also available from a PNA softkey menu.

For PNA models WITHOUT a Noise Figure option (028/029/H29), the 28V line is OFF by default and survives a preset. The ON/OFF state is NOT available from a PNA softkey menu.

**OUTPut[:STATe] <ON | OFF>**
(Read-Write) Turns RF power from the source ON or OFF.

See note about source power state with instrument state save and recall.

**Parameters**

<ON | OFF>  
- **ON** (or 1) - turns RF power ON  
- **OFF** (or 0) - turns RF power OFF

**Examples**

- OUTP ON  
- output:state off

**Query Syntax**

OUTPut[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**Last Modified:**

- 24-Feb-2009  Replaced True/False  
- 22-Aug-2007  Added noise command
**Route Command**

Learn about Frequency Offset  
SCPI Command Tree

**ROUTe<cnum>:PATH:LOOP[:R1] <char>**

*(Read-Write)*  Throws internal switch to reference receiver when the specified channel is measured.  
This feature is available on PNA models with [Option 081](#) (external reference switch) and ALL PNA-X models.

**Parameters**

- `<cnum>`  Any existing channel number. If unspecified, value is set to 1
- `<char>`  Position of the switch. Choose from:
  - **INTernal** - bypass R1 Loop. Connects the port 1 source directly to the R1 receiver.
  - **EXTernal** - flow through R1 Loop. Allows direct access to the R1 receiver through the Reference 1 front-panel connectors.

**Examples**

```
ROUTe:PATH:LOOP INT
route2:PATH:LOOP:R1 external
```

**Query Syntax**  
`ROUTe<cnum>:PATH:LOOP:R1?`

**Return Type**  Character

**Default**  INTernal

Last Modified:  
30-Apr-2009  Added PNA-X
Sense: Average Commands

Sets sweep-to-sweep averaging parameters. Averaging is a noise reduction technique that averages each data point over a user-specified number of sweeps. Averaging affects all of the measurements in the channel.

<table>
<thead>
<tr>
<th>SENSE:AVERAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLEar</td>
</tr>
<tr>
<td>COUNt</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>[STATe]</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

See Also

- Example using some of these commands.
- Learn about Averaging
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe<cnun>:AVERAGE:CLEar

(Write-only) Clears and restarts averaging of the measurement data. Does NOT apply to point averaging.

**Parameters**

<cnun> Any existing channel number; if unspecified, value is set to 1.

**Examples**

SENSE:AVER:CLE
sense2:average:clear

**Query Syntax** Not applicable

**Default** Not applicable

SENSe<cnun>:AVERAGE:COUNt <num>
(Read-Write) Sets the number of measurements to combine for an average. Must also set SENS:AVER[:STATE] ON

Parameters
<cnm> Any existing channel number; if unspecified, value is set to 1.
<num> Number of measurements to average. Choose any number between 1 and 65536 (2^16).

Examples
SENS:AVER:COUN 999
sense2:average:count 73

Query Syntax
SENS<cnm>:AVERage:COUNt?

Return Type
Numeric

Default
1

SENSe<cnm>:AVERage:MODE <char>
(Read-Write) Sets the type of averaging to perform: Point or Sweep.

Parameters
<cnm> Any existing channel number; if unspecified, value is set to 1.
<num> Averaging Type. Choose from:

POINT - Averaging measurements are made on each data point before stepping to the next data point. (Not available on 'C' models).

SWEEP - Averaging measurements are made on subsequent sweeps until the required number of averaging sweeps are performed.

Examples
SENS:AVER:MODE POIN
sense2:average:mode sweep

Query Syntax
SENS<cnm>:AVERage:MODE?

Return Type
Character

Default
Sweep

SENSe<cnm>:AVERage[:STATE] <ON | OFF>
(Read-Write) Turns trace averaging ON or OFF.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<ON | OFF>`
  - **ON** (or 1) - turns averaging ON.
  - **OFF** (or 0) - turns averaging OFF.

**Examples**

```plaintext
SENS:AVER ON
sense2:average:state off
```

**Query Syntax**
SENSe<cnum>:AVERage[:STATe]?

**Return Type** Boolean (1 = ON, 0 = OFF)

**Default** Off
**Sense:Bandwidth Commands**

SENSe:BANDwidth:
  
  RESolution <num>
  TRACk <bool>

See Also

- Example Programs
- Learn about IF Bandwidth
- Synchronizing the PNA and Controller
- SCPI Command Tree

---

**SENSe<cnum>:BANDwidth | BWIDth[:RESolution] <num>**

*(Read-Write)* Sets the bandwidth of the digital IF filter to be used in the measurement. (Use either *Sense:Bandwidth* or *Sense:Width*)

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. *(Click to see the lists.)* If an invalid number is specified, the analyzer will round up to the closest valid number.

  This parameter supports MIN and MAX as arguments. [Learn more.](#)

**Examples**

```
SENS:BWID 1KHZ
sense2:bandwidth:resolution 1000
```

**Query Syntax**

SENSe<cnum>:BANDwidth | BWIDth[:RESolution]?

**Return Type**

Numeric

**Default**

See [Preset IFBW](#) for your PNA model.

---

**SENSe<cnum>:BANDwidth | BWIDth:TRACk <bool>**
(Read-Write) Sets and returns the state of the Reduce IF BW at Low Frequencies feature. (Use either Sense:Bandwidth:Track or Sense:Bwidth:Track).

Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<bool>`: Choose from:
  - ON or 1 - Reduce IF BW at Low Frequencies is set ON
  - OFF or 0 - Reduce IF BW at Low Frequencies is set OFF

Examples

- SENS:BWID:TRAC OFF
- sense2:bandwidth:track 1

Query Syntax

SENSe<cnum>:BANDwidth | BWIDth:TRACk?

Return Type

Boolean

Default

ON

Last Modified:

15-Jan-2008   MIN and MAX added
Sense:Class Command

SENSe:CLASs:NAME?

Click on a blue keyword to view the command details.

See Also

- Learn about Measurement Class
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe<cnum>:CLASs:NAME?

(Read-only) Returns the measurement class name of the specified channel. Use Calc:Custom commands to create measurements from classes other than standard S-Parameters. Use Cal:Par:Define:Ext to create standard measurements.

Parameters

| <cnum> | Any existing channel number; if unspecified, value is set to 1. |

Examples

SENs:CLAS:NAME?
sense2:classe:name?

For a standard S-Parameter channel, returns...
"Standard"

Default
Not applicable

Last Modified:

12-May-2008   MX New topic
Sense:Correction Commands

Performs and applies calibration and other error correction features.

- To perform a Guided Calibration, use ONLY the Sens:Corr Coll:GUIDed commands.
- To perform an Unguided Calibration, do NOT use the Sens:Corr:Coll:Guided commands.
- See the "Unguided example programs" for clarification.

SENSe:CORRection
  CCheck
    | [ACQuire]
    | DONE
    | PARameter
  CKIT - More Commands
  COLLect
    | [ACQuire]
    | APPLy
    | CKIT - More Commands
    | DISPlay:WINDow
      | AOFF
      | [STATe]
  GUIDed - More Commands
  ISOLation:
    | AVER:INCRement
    | ECAL:[STATe]
  NOISe
    | ENR:ADAP:DEEMbed[:STATe]
    | PSEN:ADAP:DEEMbed[:STATe]
    | LO:PCAL[:STATe]
  METHod
  SAVE
  SESSion - More Commands
  SWEep:CHANnel
    | AOFF
    | [STATe]
    | WINDow[:STATe]
CSET - More Commands
ENR:CALibration:TABLE
  | DATA
  | ID:DATA
| SERial:DATA |
| EXTension - More Commands |
| GCSetup |
| | POWer |
| | SENSor: |
| | CKIT |
| | CONNector |
| IMD - More Commands |
| IMPedance:INPut |
| | MAGNitude |
| INTerpolate[:STATe] |
| ISOLation[:STATe] |
| METHods:MATCh |
| PREFerence |
| | CALibration |
| | [FOM:]RANGe |
| | CSET |
| | SAVE |
| | SAVUser |
| | ECAL |
| | ORIentation |
| | PMAP |
| | SIMCal |
| | TRIG:FREE |
| RPOWER:OFFSet |
| | [AMPLitude] |
| RVELocity |
| | COAX |
| SFOWard |
| | [STATe] |
| [STATe] |
| TCOLd:USER:VALue |
| TSTandards |
| | [STATe] |
| TYPE |
| | CATalog? |

Click on a blue keyword to view the command details.

Red commands are superseded.

See Also
SENSe<cnum>:CORRection:CCHeck[:ACQuire] <mod>[,char]

(Write-only) Reads the 'confidence data' associated with the specified ECal module and puts it into memory. The measurement is selected using SENS:CORR:CCH:PAR. This command is compatible with *OPC.

Note: A confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1.
<mod> ECal Module that contains the confidence data. Choose from:

ECAL1
..through..

ECAL8

[char] Optional argument. Specifies which characterization within the ECal module that the confidence data will be read from.

CHAR0 Factory characterization (data that was stored in the ECal module by Agilent). Default if not specified.

CHAR1 User characterization #1

CHAR2 User characterization #2

...and so forth up to:

CHAR12 User characterization #12

Examples SENS:CORR:CCHCheck ECAL

sense2:correction:ccheck:acquire ecal1,char1

Query Syntax Not applicable
**SENSe<cnum>:CORRection:CCHeck:DONE**

(Write-only) Concludes the Confidence Check and sets the ECal module back into the idle state.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1

**Examples**

- `SENSe:CORRection:CCHeck: DONE`
- `sense2:correction:ccheck:done`

**Query Syntax**

- **Default**: Not applicable

---

**SENSe<cnum>:CORRection:CCHeck:PARameter <Mname>**

(Read-Write) Specifies an existing measurement to be used for the Confidence Check.

**Note:** A confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<Mname>`: Name of the measurement you are selecting for the confidence check. The measurement must already exist.

**Examples**

- `SENSe:CORRection:CCHeck:PAR 'TEST'`
  - 'selects the measurement "test" on channel 1 for the confidence check'
- `sense2:correction:ccheck:parameter 'test'`
  - 'selects the measurement "test" on channel 2 for the confidence check'

**Query Syntax**

- `SENSe<cnum>:CORRection:CCHeck:PARameter?`
  - Returns the name of the selected measurement on channel `<cnum>`.

**Return Type**

- **String**

**Default**

- Not applicable
For UNGUIDED calibration, measures the specified standards from the selected calibration kit. The calibration kit is selected using the `Sense:Correction:Collect:CKIT` command. For using two sets of standards, see `SENS:CORR:TST`.

**Note:** Before using this command you must select two items:
1. Select a calibration method using `SENS:CORR:COLL:METH`
2. Select a measurement using `CALC:PAR:SEL`. You can select one measurement for each channel.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<class>` **Measures the standards associated with these class labels.** Choose from:

<table>
<thead>
<tr>
<th>Label</th>
<th>Forward</th>
<th>Reverse</th>
</tr>
</thead>
<tbody>
<tr>
<td>STAN1</td>
<td>S11A</td>
<td>S22A</td>
</tr>
<tr>
<td>STAN2</td>
<td>S11B</td>
<td>S22B</td>
</tr>
<tr>
<td>STAN3</td>
<td>S11C</td>
<td>S22C</td>
</tr>
<tr>
<td>STAN4</td>
<td>S21T</td>
<td>S12T</td>
</tr>
<tr>
<td>STAN5</td>
<td>Generic Isolation; not associated with calibration kit definition.</td>
<td></td>
</tr>
<tr>
<td>ECAL1</td>
<td>ECAL modules through</td>
<td></td>
</tr>
<tr>
<td>ECAL2</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL3</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL4</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL5</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL6</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL7</td>
<td>through</td>
<td></td>
</tr>
<tr>
<td>ECAL8</td>
<td>through</td>
<td></td>
</tr>
</tbody>
</table>

**RESPonse** Same as `Normalize` selection in Unguided Cal. (subclass is ignored)

**POWER** Take a receiver power cal sweep and turn correction ON

**SLSET** Sets 'sliding load type', and increments the "number of slides" count. The total number of slides is critical to the correct calculation of the sliding load algorithm. See a sliding load cal example.

**SLDONE** Computes the sliding load using a circle fit algorithm.

[subclass] Optional argument. For mechanical calibration kits, choose from the following to specify the standard to be acquired from the `SENS:CORR:COLL:CKIT:ORDer` list. If not specified, subclass is set to `SST1`. 
SST1  First standard in the order list
SST2  Second standard in the order list
SST3  Third standard in the order list
SST4  Fourth standard in the order list
SST5  Fifth standard in the order list
SST6  Sixth standard in the order list
SST7  Seventh standard in the order list

If an ECAL module (1 through 8) is specified for <class>, choose one of the following for specifying which characterization within the ECal module will be used for the acquire. If not specified, the default is CHAR0.

  CHAR0  Factory characterization (data that was stored in the ECal module by Agilent)
  CHAR1  User characterization #1
  CHAR2  User characterization #2

...and so forth up to:

  CHAR12 User characterization #12

[sync] Optional argument. Choose from:

  SYNChronous - blocks SCPI commands during standard measurement (default behavior)
  ASYNchronous - does NOT block SCPI commands during standard measurement.

Learn more.

Examples

SENS:CORR:COLL STAN1

'If SENS:CORR:COLL:CKIT:ORDer2 5,3,7 was specified, the following command measures standard 3 (the second in the order list)
sense1:correction:collect:acquire stan3,sst2

SENS:CORR:COLL ECAL4,ASYN; *OPC?
sense2:correction:collect:acquire ecal2,char1

Query Syntax

  Default  Not applicable

  Not applicable
SENSe<cnun>:CORRection:COLLect:APPLy

(Write-only) Applies error terms to the measurement that is selected using Calc:Par:Select.

Note: Before using this command you must select a measurement using CALC:PAR:SEL. You can select one measurement for each channel.

Note: This command is only necessary if you need to modify error terms. If you do not need to modify error terms, SENSe<cnun>:CORRection:COLLect:SAVE calculates and then automatically applies error terms after you use SENS:CORR:COLL:ACQuire to measure cal standards.

Parameters

<cnun> Any existing channel number. If unspecified, value is set to 1

Example

1. CALCulate2:PARameter:SELect S21_2 'select the measurement to apply terms to
2. SENSe2:CORRection:COLLect:METHod SPARSOLT 'set type of cal method.
3. CALCulate2:DATA? SCORR1 'download the error term of interest
4. 'Modify the error term here
5. CALCulate2:DATA SCORR1 'upload the error term of interest
6. SENSe2:CORRection:COLLect:APPLy 'applies the error terms to the measurement

Query Syntax

Not applicable

Default

Not applicable

SENSe:CORRection:COLLect:DISPlay:WINDow:AOFF

(Write-only) Clears the flags for windows to be shown during calibrations. To flag a window to be shown see SENS:CORR:COLL:DISP:WIN.

Examples

SENSe:CORRection:COLLect:DISP:WIN:AOFF

sense:correction:collect:display:window:aoff

See an example using this command.

Query Syntax

Not Applicable

Default

Not Applicable

SENSe:CORRection:COLLect:DISPlay:WINDow<wNum>[:STATE] <bool>
(Write-only) Set the 'show' state of the window to be displayed during a calibration. Learn more.

When this command is sent, the specified window is 'flagged' to be shown during calibration. The flag is cleared when the window is closed. A Preset or Instrument State Recall also closes the window. If the same window number is reopened, this command must be sent again to show the window during a calibration. The flag is NOT saved with an instrument state.

Send this command for each additional window to show during a calibration.

**Parameters**

- `<wNum>` Window number to show during a calibration. The calibration window will also be shown with this window.
  
The window must already be created.
  
  Use `DISPlay:CATalog?` to read all existing window numbers.
- `<bool>` Window state. Choose from:
  
  **ON** (or 1) - Show the specified window during calibration.
  
  **OFF** (or 0) - Do NOT show the specified window during calibration.

**Examples**

```
SENS:CORR:COLL:DISP:WIND1 1
sense:correction:collect:display:window2:state off
```

See an example using this command.

**Query Syntax** Not Applicable

**Default** OFF

**SENSe:CORRection:COLLect:ISOLation:AVERage:INCRement <num>**
(Read-Write) Specifies amount to increment (increase) the channel averaging factor during isolation measurement of the ECal module during an unguided ECal calibration.

**Note:** if the channel currently has averaging turned OFF and `<num>` is greater than 1, averaging will be turned ON only during the isolation measurements and with the averaging factor equal to `<num>`.

**Parameters**

- `<num>`: Incremental Averaging factor. The maximum averaging factor is 65536 (2^16).

**Examples**

- `SENS:CORR:COLL:ISOL:AVER:INCR 16`
- `sense:correction:collect:isolation:average:increment 0`

**Query Syntax**

SENS:CORRection:COLLect:ISOLation:AVERage:INCRement?

**Return Type**

Numeric

**Default**

8 - If this command is NOT sent, but ECal isolation is measured, then averaging will be turned ON with factor set to 8 during the isolation measurement.

---

**SENSe<cnum>:CORRection:COLLect:ISOLation:ECAL[:STATe] <bool>**

(Read-Write) Specifies whether or not the isolation state of the ECal module will be measured as part of an unguided ECal calibration.

An unguided calibration is performed using the SENS:CORR:COLL:METH and SENS:CORR:COLL:ACQ commands.

**Note:** The inherent isolation of the PNA is better than that attained with this command. ONLY use this command when using an external test set, and ONLY using a 8509x ECal module.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<bool>`: **ON** (or 1) - isolation is measured during the unguided ECal calibration.
  
  **OFF** (or 0) isolation is NOT measured during the unguided ECal calibration.

**Examples**

- `SENS1:CORR:COLL:ISOL:ECAL ON`
- `sense2:correction:collect:isolation:ecal:state 0`

**Query Syntax**

SENS:CORRection:COLLect:ISOLation:ECAL[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF
SENSe<nump>:CORRection:COLLect:METHod <char>

(Read-Write) For UNGUIDED calibration, sets the calibration method (also known as 'Calibration Type' on calibration dialog box.) To select a Cal Type from a Cal Set, use CALC:CORR:TYPE.

**Note:** Before using this command you must select a measurement using CALC:PAR:SEL. You can select one measurement for each channel.

**Parameters**

- `<nump>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Choose from:

  **Method** | **Description**
  --- | ---
  NONE | No Cal method
  REFL1OPEN | Response Open
  REFL1SHORT or | Response Short
  REFL1 |  
  REFL3 | Full 1 port
  RESPONSE | Same as Normalize selection in Unguided Cal.
  RPOWER | Receiver Power Cal - Used only with receiver measurements.
  TRAN1 | Response Thru - Requires a Thru standard.
  TRAN2 | Response Thru and Isolation - Requires a Thru standard.
  SPARSOLT | Full SOLT 2 port
  SPARTRL | TRL Cal (not available on all PNAs.)

**Examples**

```
SENS:COR:COLL:METH REFL1
sense2:correction:collect:method sparsolt
```

**Query Syntax**

SENSe<nump>:CORRection:COLLect:METHOD?

**Return Type**

Character

**Default** Not Applicable

---

(Read-Write) Set and read the state of ENR Adapter de-embedding. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<bool>` ENR Adapter de-embed state. Choose from:
  - **OFF** or **0** - Do not force de-embedding.
  - **ON** or **1** - Force de-embedding.

**Examples**

```
SENSe:CORR:COLL:NOIs:ENR:ADAP:DEEM 0
```

**Query Syntax**

`SENSe:CORRection:COLLect:NOISe:ENR:ADAPter:DEEMbed:[STATe]?`

**Return Type**

Boolean

**Default**

O - OFF

---

**SENSe<ch>:CORRection:COLLect:NOISe:PSENsor:ADAPter:DEEMbed:[STATe] <bool>**

(Read-Write) Set and read the state of power sensor adapter de-embedding. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<bool>` Power sensor adapter de-embed state. Choose from:
  - **OFF** or **0** - Do not force de-embedding.
  - **ON** or **1** - Force de-embedding.

**Examples**

```
SENSe:CORR:COLL:NOIs:PSEN:ADAP:DEEM 0
```

**Query Syntax**

`SENSe:CORRection:COLLect:NOISe:PSENsor:ADAPter:DEEMbed:[STATe]?`

**Return Type**

Boolean

**Default**

O - OFF

---

**SENSe<ch>:CORRection:COLLect:NOISe:LO<n>:PCAL[:STATe] <bool>**
(Read-Write) Enables and disables LO power calibration for NFX.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` LO Stage (number). Choose 1 for NFX.
- `<bool>` LO Power Cal state. Choose from:
  - **OFF** or **0** - Disable LO Power Cal
  - **ON** or **1** - Enable LO Power Cal

**Examples**

```
SENS:CORR:COLL:NOIS:LO1:PCAL 0
sense2:correction:collect:noise:lo1:pcal:state ON
```

**Query Syntax**

`SENSe:CORRection:COLLect:NOISe:LO<n>:PCAL:STATe?`

**Return Type**

Boolean

**Default**

O - OFF

---

**SENSe<cnum>:CORRection:COLLect:SAVE**

(Write-only) For UNGUIDED calibrations ONLY. This command does the following:

- calculates the error terms using the selected :METHod
- applies the error terms to the selected measurement (turns error correction ON.)
- saves the calibration error-terms to the channels Cal Register or a User Cal Set.

The Cal Register or User Cal Set is determined by the setting of the `SENS:CORR:PREFerence:CSET:SAVE` command.

Do NOT use this command during an ECAL. When performing an ECAL calibration using `SENS:CORR:COLL:ACQuire`, this SAVE operation is performed automatically before the completion of a successful ACQuire.

Before using this command you must select a measurement using `CALC:PAR:SEL`. You can select one measurement for each channel.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
SENSe:CORRection:COLLect:SWEep:CHANnel:AOFF

(Write-only) Clears ALL flags for channels to sweep during calibration. To flag a channel, see SENSe:CORR:COLL:SWE:CHAN.

Examples  
SENSe:CORRection:COLLect:SWEep:CHANnel:AOFF

sense:correction:collect:sweep:channel:aoff

See an example using this command.

Default  Not applicable

SENSe<cnum>:CORRection:COLLect:SWEep:CHANnel<cnum2>[[:STATe] <bool>]

(Write-only) Specifies the channel to sweep during a Calibration.

When this command is sent, the <cnum2> channel is 'flagged' to be swept during calibration.

The flag is cleared when the channel is deleted, if the Measurement Class is changed, or if all measurements are deleted from the channel.

If the same channel number is recreated, this command must be sent again to sweep the channel during a calibration. The flag is NOT saved with an instrument state.

A Preset or Instrument State Recall deletes the channel.

Parameters

<cnum>  The channel to be calibrated. If unspecified, value is set to 1.
<cnum2>  The channel to sweep when waiting to measure a standard.

This channel must already exist with at least one measurement in the channel. If this channel is in continuous sweep mode, it must have the same attenuator settings and path configuration (PNA-X only).
<bool> Channel sweep state. Choose from:

ON (or 1) - Sweep the channel during calibration.

OFF (or 0) - Do NOT sweep the channel during calibration.

Examples

SENSe:CORRection:COLL:SWE:CHAN2 1
sense2:correction:collect:sweep:channel3:state off

See an example using this command.

Query Syntax Not Applicable

Default OFF

SENSe:CORRection:ENR:CALibration:TABLE:DATA <freq, value, freq, value...>

(Read-Write) Set and read the ENR calibration data. All of the frequency and ENR data must be sent at the same time. Use MMEM:LOAD and MMEM:SAVE to load and save ENR table data from disk. Learn more about Noise Source ENR files.

Parameters

<freq, value> (Numeric) ENR data. Frequency value in Hz followed by a ENR noise value in dB. Enter as many pairs as necessary.

Examples

SENSe:CORRection:ENR:CALibration:TABLE:DATA 1.0E9,14.37,2.5E9,15.28
sense:correction:enr:calibration:table:data 1.0E9,14.37,2.5E9,15.28

Query Syntax SENSe:CORRection:ENR:CALibration:TABLE:DATA?

Return Type Comma separated numeric values

Default Not Applicable

SENSe:CORRection:ENR:CALibration:TABLE:ID:DATA <id>
(Read-Write) Sets and returns ID of ENR table. While this is for informational purposes only, it can be used to record the model of the noise source. Learn more about ENR files.

**Parameters**

- **<id>** (String) Identifier for the ENR table.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>

**Query Syntax**

SENS:CORRection:ENR:CALibration:TABLe:ID:DATA?

**Return Type**

String

**Default**

Not Applicable

---

**SENS:CORRection:ENR:CALibration:TABLe:SERial:DATA <sn>**

(Read-Write) Sets and returns the serial number of noise source. This is for informational purposes only to identify the specific noise source for which the data pertains. Learn more about ENR files.

**Parameters**

- **<sn>** Serial number of the noise source for which the data applies, enclosed in quotes.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>

**Query Syntax**

SENS:CORRection:ENR:CALibration:TABLe:SERial:DATA?

**Return Type**

String

**Default**

Not Applicable

---

**SENS<ch>:CORRection:GCSetup:POWer <num>**

(Read-Write) Set and read the power level at which to perform the Source Power Cal portion of a Gain Compression (Opt 086) Calibration. Learn more about this setting.

**Parameters**

- **<num>** Power level in dB. Choose a value from +30 to -30.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:CORR:GCS:POW 0</td>
<td>Sense:correction:gcsetup:power 5</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENS:CORRection:GCSetup:POWer?

**Return Type**

Numeric

**Default**

0
SENSe<ch>:CORRection:GCSetup:SENSor:CKIT <string>

(Read-Write) Set and read the cal kit to be used for calibrating at the port 1 reference plane when the power sensor connector is different from the DUT port 1. Learn more.

Parameters


Examples

```
SENS:CORR:GCS:SENS:CKIT "85052B"
```

Query Syntax

SENSe:CORRection:GCSetup:SENSor:CKIT?

Return Type

String

Default

Not Applicable

SENSe<ch>:CORRection:GCSetup:SENSor:CONNector<string>

(Read-Write) Set and read the power sensor connector type which is used to perform the Source Power Cal portion of a Gain Compression Calibration. Learn more.

Parameters


Examples

```
SENS:CORR:GCS:SENS:CKIT "3.5 mm (50) male"
```

Query Syntax

SENSe:CORRection:GCSetup:SENSor:CKIT?

Return Type

String

Default

Not Applicable

SENSe:CORRection:IMPedance:INPut:MAGNitude <num>
(Read-Write) Sets and returns the system impedance value for the analyzer.

**Parameters**

<num> System Impedance value in ohms. Choose any number between 0 and 1000 ohms.

**Examples**

SENS:CORR:IMP:INP:MAGN 75
sense:correction:impedance:input:magnitude 50.5

**Query Syntax**

SENSe:CORRection:IMPedance:INPut:MAGNitude?

**Return Type** Numeric

**Default** 50

**SENSe<ch>:CORRection:INTerpolate[:STATe] <ON | OFF>**

(Read-Write) Turns correction interpolation ON or OFF.

**Note:** Before using this command you must select a measurement using CALC:PAR:SEL. You can select one measurement for each channel.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<ON | OFF> ON (or 1) - turns interpolation ON.
OFF (or 0) - turns interpolation OFF.

**Examples**

SENS:CORR:INT ON
sense2:correction:interpolate:state off

**Query Syntax**

SENSe<cnum>:CORRection:INTerpolate[:STATe]?

**Return Type** Boolean (1 = ON, 0 = OFF)

**Default** ON

**SENSe<ch>:CORRection:ISOLation[:STATe] <ON | OFF> OBSOLETE**
This command no longer works beginning in the PNA 5.2 release. To perform isolation as part of an unguided calibration, you must explicitly measure the isolation standard using **SENS:CORR:COLL:ACQ Stan5**. See Example program.

To measure isolation as part of an ECal, use **SENS:CORR:COLL:ISOL:ECAL**.

(Read-Write) Turns isolation cal ON or OFF during Full 2-port calibration. If this command is not sent, the default state is to **disable** Isolation.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>` **ON** (or 1) - turns isolation ON.  
  **OFF** (or 0) - turns isolation OFF.

**Examples**

```
SENS:CORR:ISOL ON
sense2:correction:isolation:state off
```

**Query Syntax**

```
SENSe<cnum>:CORRection:ISOLation[:STATe]?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF - (Isolation disabled)

**SENS<ch>:CORRection:METHods:MATCh <bool>**

(Read-Write) Turns match-correction ON or OFF. Use this command AFTER performing an Guided Power Cal. [Learn more.]

**Parameters**

- `<ch>` Channel number on which Guided Power Cal was performed. If unspecified, value is set to 1
- `<bool>` **ON** (or 1) - Turns match-correction ON  
  **OFF** (or 0) - Turns match-correction OFF.

**Examples**

```
SENS:CORR:METH:MATC 0
sense2:correction:methods:match off
```

**Query Syntax**

```
SENSe<cnum>:CORRection:METHods:MATCh?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

**SENS:CORRection:PREFerence:CALibration[:FOM]:RANGe <char>**
(Read-Write) Specifies the FOM frequency range to use when performing calibration.

**Parameters**

<char> Choose from:

- **PRIMary** - Used for calibrating at the mmWave frequencies when NOT using a test set. Learn more.

- **AUTO** - All other calibration situations.

**Examples**

```
SENS:CORR:PREF:CAL:RANG PRIM
sense:correction:preference:calibration:fom:range auto
```

**Query Syntax**

```
SENSe:CORRection:PREFerence:CALibration[:FOM]:RANGe?
```

**Return Type** Character

**Default** AUTO

---

**SENSe:CORRection:PREFerence:CSET:SAVE <char>**

**Important Notes:**

- This command replaces SENS:CORR:PREF:CSET:SAVU

- With 6.0 we implemented a change that defaults to saving completed calibrations to Cal Registers instead of User Cal Sets. To revert to the old behavior, send this command with the USER argument.

(Read-Write) Specifies the default manner in which calibrations that are performed using SCPI or COM are to be stored. Cal data is ALWAYS stored to the channel Cal Register regardless of this setting.

This setting survives instrument preset and reboot. It remains until changed by another execution of this command.

**Note:** Cal Set arguments used with commands such as SENS:CORR:COLL:GUID:INIT, SENS:CORR:COLL:GUID:SAVE and SENS:CORR:COLL:GUID:SAVE:CSET will override of any of these default preference settings.

Learn about Cal Registers and User Cal Sets.

**Parameters**

<char> **CALRegister** - Cal is saved ONLY to the channel Cal Register.

- **USER** - Cal is saved to a new User Cal Set file when performing a SCPI calibration. The Cal Set name is automatically generated. To change the name,
send `SENS:CORR:CSET:NAME` after the cal is complete. This reverts to pre-6.0 behavior.

**REUSE** - The cal is saved to the Cal Set that is currently selected on the specified channel, which could be the channel Cal Register. If the channel does not yet have a selected Cal Set, the cal will be saved to a new User Cal Set with an automatically-generated name.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SENS:CORR:PREF:CSET:SAVE USER</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><em>sense:correction:preference:cset:save reuse</em></td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>SENSE:CORRection:PREFerence:CSET:SAVE?</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Return Type</strong></td>
<td>Character</td>
</tr>
<tr>
<td><strong>Default</strong></td>
<td>CALRegister</td>
</tr>
</tbody>
</table>

**SENSe:CORRection:PREFerence:CSET:SAVUser <bool> Superseded**

This command is replace with `SENS:CORR:PREF:CSET:SAVE`.

**NOTE:** With 6.0 we implemented a change that defaults to saving completed calibrations to Cal Registers instead of User Cal Sets. To revert to the old behavior, send this command as ON (1). For UI and COM use, this can be done from the GPIB console.

(Read-Write) Specifies whether cal data is automatically saved to a User Cal Set file after performing a SCPI calibration. Cal data is always saved to a Cal Register regardless of this setting.

This setting survives instrument preset and reboot. It remains until changed by another execution of this command.

Learn about [Cal Registers and User Cal Sets](https://example.com/cal_registers_and_user_cal_sets).

**Parameters**

`<bool>`

- **ON** or **1** - Cal is automatically saved to a User Cal Set file when performing a SCPI calibration. The Cal Set name is automatically generated. To change the name, send `SENS:CORR:CSET:NAME` after the cal is complete. Reverts to pre-6.0 behavior.

- **OFF** or **0** - Cal is NOT automatically saved to a User Cal Set. To save a calibration to a User Cal Set, use `SENS:CORR:COLL:GUID:INIT`.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SENS:CORR:PREF:CSET:SAVU 1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><em>sense:correction:preference:cset:saveuser 0</em></td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>SENSE:CORRection:PREFerence:CSET:SAVUser?</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Return Type</strong></td>
<td>Boolean (1 = ON, 0 = OFF)</td>
</tr>
</tbody>
</table>

2455
SENSe:CORRection:PREFerence:ECAL:ORIentation[:STATe] <ON|OFF>

(Read-Write) Specifies whether or not the PNA should perform orientation of the ECal module during calibration. Orientation is a technique by which the PNA automatically determines which ports of the module are connected to which ports of the PNA. Orientation begins to fail at very low power levels or if there is much attenuation in the path between the PNA and the ECal module. If orientation is turned OFF, the SENS:CORR:PREF:ECAL:PMAP command must be used to specify the port connections before performing a cal.

**Note:** For 3-port or 4-port measurements, when orientation is OFF, you are not allowed to specify how the ECAL module is connected. Instead, the PNA determines the orientation. Use SENS:CORR:COLL:GUID:DESC? to query the orientation. The PNA does not verify that you made the connection properly.

This setting remains until the PNA is restarted or this command is sent again.

**Parameters**

<bool> ECAL orientation state. Choose from:

**ON** or 1 - PNA performs orientation of the ECal module.

**OFF** or 0 - PNA does NOT performs orientation of the ECal module.

**Examples**

SENS:CORR:PREF:ECAL:ORI OFF

SENSe:CORRection:PREFerence:ECAL:ORIentation[:STATe] ON

**Query Syntax**

SENSe:CORRection:PREFerence:ECAL:ORIentation[:STATe]?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON (1)

SENSe:CORRection:PREFerence:ECAL:PMAP <module>,<string>

(Read-Write) When ECal module orientation is turned OFF (SENS:CORR:PREF:ECAL:ORI), this command specifies the port mapping (which ports of the module are connected to which ports of the PNA) prior to performing ECal calibrations.

This setting remains until the PNA is restarted or this command is sent again.

**Parameters**

<module> Specifies which ECal module this port map is being applied to. Choose from:
Format this parameter in the following manner:

\[ A_w, B_x, C_y, D_z \]

where

- \( A, B, C, \) and \( D \) are literal ports on the ECAL module
- \( w, x, y, \) and \( z \) are substituted for PNA port numbers to which the ECAL module port is connected.

Ports of the module which are not used are omitted from the string.

For example, on a 4-port ECal module with

- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1

the string would be: \( A_2, B_3, D_1 \)

If either the receive port or source port (or load port for 2-port cal) of the CALC:PAR:SELeected measurement is not in this string and orientation is OFF, an attempt to perform an ECal calibration will fail.

**Examples**

```
SENSe:CORR:PREF:ECAL:PMAP ECAL2, 'A1,B2'
SENSe:CORR:PREF:ECAL:PMAP ECAL3, 'a2,b1,c3'
```

**Query Syntax**

SENSe:CORR:PREF:ECAL:PMAP? <module>

**Return Type**

String

**Default**

Null string ()
(Read-Write) Sets and returns a preference for the Unguided Cal behavior described below. This setting persists until it is changed.

This preference can also be set ON by executing the script on the PNA at C:/Program Files/Agilent/Network Analyzer/System/wincal32.reg.

Parameters

<bool> Boolean - Choose from:

0 - OFF - Reverts to new (preferred) behavior. An error is returned if standard data is not acquired before sending SENS:CORR:COLL:SAVE.

1 - ON - (WinCal compatible) Prevents SENS:CORR:COLL:SAVE from failing when standard data has not, and will not, be acquired.

Learn more about old and new behaviors.

Examples

SENS:CORR:PREF:SIMC 0
sense:correction:preference:simcal 1

Query Syntax
SENSe:CORRection:PREFerence:SIMCal?

Return Type
Boolean

Default
0

SENSe:CORRection:PREFerence:TRIG:FREE <char>, <bool>

(Read-Write) Sets and returns the preference for the trigger behavior during a calibration. This setting persists until it is changed.

Note: If TRIGger:SOURce = Manual, during a calibration the PNA ALWAYS switches to Internal for one trigger, then back to Manual, regardless of this preference command.

Parameters

<char> Character - Calibration type. Choose from:

GUIDed - preference setting pertains to a Guided calibration.

UNGuided - preference setting pertains to an Unguided calibration.

<bool> Boolean - Choose from:

0 - OFF - The trigger behavior during the specified calibration type DOES respect the setting of the TRIGger:SOURce command. For example, when Trigger source = External, the single trigger method will wait for the External
trigger signal and then allow only one sweep.

1 - ON - (Pre-6.0 behavior) The trigger behavior during the specified calibration type does NOT respect the setting of the TRIGger:SOURce command. For example, when Trigger source = External, during calibration the PNA switches to Internal sweep, responds to one trigger signal to measure the standard, then switches back to External.

**Examples**

<table>
<thead>
<tr>
<th>SENS:CORR:PREF:TRIG:FREE GUID,1</th>
</tr>
</thead>
<tbody>
<tr>
<td>sense:correction:preference:trig:free unguided,0</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENSe:CORRection:PREFerence:TRIG:FREE? <char>

**Return Type**

Boolean

**Default**

OFF for both calibration types.

---

**SENSe<cnum>:CORRection:RPOWer:OFFSet[:AMPLitude] <num>**

(Read-Write) Adjusts a receiver power cal to account for components or adapters that are added between the source port and receiver while performing this cal. For more information, see Receiver Cal.

**Parameters**

- <cnum>: Any existing channel number. If unspecified, value is set to 1
- <num>: Offset Value in dB. Specify loss as a negative number; and gain as a positive number. Choose a number between -200 and 200.

**Examples**

<table>
<thead>
<tr>
<th>SENS:CORR:RPOW:OFFS .5</th>
</tr>
</thead>
<tbody>
<tr>
<td>sense2:correction:rpower:offset:amplitude .-5</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENSe<cnum>:CORRection:RPOWer:OFFSet[:AMPLitude]?

**Return Type**

Numeric

**Default**

0

---

**SENSe<cnum>:CORRection:RVELocity:COAX <num>**
(Read-Write) Sets the velocity factor to be used with Electrical Delay and Port Extensions.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Velocity factor. Choose a number between 0 and 10

(0.66 polyethylene dielectric; .7 PTFE dielectric)

**Examples**

```
SENS:CORR:RVEL:COAX .66
sense2:correction:rvelocity:coax .70
```

**Query Syntax**

```
SENSe<cnum>:CORRection:RVELocity:COAX?
```

**Return Type** Numeric

**Default** 1

---

**SENSe<cnum>:CORRection:SFORward[:STATe] <boolean>**

(Read-Write) Sets the direction a calibration will be performed when only one set of standards is used.

Use `SENSe:CORRection:TSTandards[:STATe] OFF` to specify that only one set of standards will be used.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<boolean>` ON (1) - FORWARD direction of a 2-port calibration will be performed
  
  OFF (0) - REVERSE direction of a 2-port calibration will be performed

**Examples**

```
SENS:CORR:SFOR 1
sense2:correction:sforward:state 0
```

See an example using this command

**Query Syntax**

```
SENSe<cnum>:CORRection:SFORward[:STATe]?
```

**Return Type** Boolean

**Default** ON

---

**SENSe<cnum>:CORRection[:STATe] <ON | OFF>**
(Read-Write) Turns error correction ON and OFF for the specified channel.

**Note:** Before using this command you must select a measurement using `CALC:PAR:SEL`. You can select one measurement for each channel.

### Parameters

- `<cnum>`  
  Any existing channel number. If unspecified, value is set to 1.
- `<ON | OFF>`  
  - **ON** (or 1) - correction is applied to the channel.
  - **OFF** (or 0) - correction is NOT applied to the channel.

### Examples

```plaintext
SENS:CORR ON
sense2:correction:state off
```

### Query Syntax

```
SENSe<cnum>:CORRection[:STATe]?
```

To query the error correction state for a measurement, use `CALC:CORR:STATe?`

### Return Type

- **Boolean** (1 = ON, 0 = OFF)

### Default

OFF

---

**SENSe<cnum>:CORRRection:TCOLOd:USER:VALue <num>**

(Read-Write) Sets and returns the temperature of the noise source connector. Learn more about Noise Figure Calibration.

### Parameters

- `<cnum>`  
  Any existing channel number. If unspecified, value is set to 1.
- `<num>`  
  Noise source temperature in Kelvin.

### Examples

```plaintext
SENS:CORR:TCOL 295
sense2:correction:tcold 298
```

See an example using this command

### Query Syntax

```
SENSe<cnum>:CORRection:TCOLd:USER:VALue?
```

### Return Type

- **Numeric**

### Default

Not Applicable

---

**SENSe<cnum>:CORRRection:TSTandards[:STATe] <boolean>**
(Read-Write) Specifies the acquisition of calibration data using ONE or TWO sets of standards.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<boolean>`
  - **ON (1)** - TWO identical sets of standards will be used to simultaneously calibrate two ports (for both Forward and Reverse parameters).
  - **OFF (0)** - ONE set of standards will be used to perform a full 2-port calibration, one port at a time.

When specifying ON (use two sets), the `SENS:CORR:COLL:ACQuire` command uses the same standard index for each calibration class. To specify the calibration standard gender for each port, you must first ensure that the order of calibration class accurately reflects the configuration of your DUT. For example, for a DUT with a male connector on port 1 and a female connector on port 2, order the devices within the S11 classes (A, B, and C) such that the MALE standards are first in the list. Then order the S22 classes specifying the FEMALE standards as the first in the list.

**Examples**

```plaintext
SENS:CORR:TST 1
sense2:correction:tstandard:state 0
```

See an example using this command

**Query Syntax**

`SENSe<cnum>:CORRection:TSTandards[:STATe]?`

**Return Type**

Boolean

**Default**

ON

`SENSe:CORRection:TYPE:CATalog? <char>`
(Read-Write) Lists the Cal Types in the PNA by either GUID or registered name. Learn more about applying Cal Type using SCPI.

**Note:** Before using this command you must select a measurement using `CALC:PAR:SEL`. You can select one measurement for each channel.

**Parameters**

<char> Specifies the type of list. Choose from:

- **GUID** - the registered GUID of the Cal Type
- **NAME** - the registered name of the Cal Type

**Examples**

```
SENS:CORR:TYPE:CAT? GUID
```

**Query Syntax**

```
SENSe<cnum>:CORRection:TYPE:CATalog? <char>
```

**ReturnType**

Comma-separated string

**Default**

Not Applicable

---

Last modified:

- 23-Oct-2010 Added SENS:CORR:METH:MATC
- 15-Jun-2010 Updated confidence check to 12 User chars
- 26-Oct-2009 Added Noise commands and Normalization method. (9.1)
- 16-Sep-2009 Added Cal Pref FOM
- 13-Feb-2009 Added Acq, sync
- 23-Jan-2009 Added 'if meas deleted' to Swe:Chan
- 6-Mar-2008 Added Noise TCOLd
- 19-Sep-2007 Added missing <cnum> arguments
- July 30, 2007 Added ENR commands
- April 14, 2007 Addd ECal isolation commands
- Oct 30, 2006 Modified SavUser command
Sense:Correction:CKIT Commands

Manages the list of cal kits that are installed in the PNA.

SENSe:CORR:CKIT
  CLEar
  COUNt?
  ECAL
    | CHARacterize More commands
    | CLIS?
    | DMEMory
      | CLEar
      | IMPort
    | EXPort
    | INFormation
    | KNAMe
      | INFormation
    | LIST?
    | ORIent?
    | PATH
      | COUNt?
      | DATA?
  EXPort
  IMPort
  INITialize
  LOAD

- Click on a blue keyword to view the command details.
- Red is a superseded command
**SENSe:CORRection:CKIT:CLEar[:IMMediate] [ckit]**

(Write-only) Deletes installed cal kits.

**Parameters**

[ckit] Optional String. Cal Kit to delete. If not specified, all PNA Cal kits are deleted, including custom kits.

**Examples**

```
SENS:CORR:CKIT:CLE
sense:correction:kit:clear:immediate "85052B"
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe:CORRection:CKIT:COUNt?**

(Read-only) Returns the number of installed cal kits.

**Examples**

```
SENS:CORR:CKIT:COUNt?
```

**Query Syntax**

SENS:CORR:CKIT:COUNt?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:CKIT:ECAL<mod>:CLIS?**
(Read-only) Returns a list of characterizations stored in the specified ECal module.

**Parameters**

<mod> ECal module from which to read user characterization numbers. If unspecified, value is set to 1.

**Examples**

Module 1 contains User Characterizations 1 and 3.

SENSe:CORRection:CKIT:ECAL:CLIST?

Returns the following (0 always indicates the factory characterization):

0,1,3

**Return Type**

Numeric list, separated by commas.

**Default**

Not Applicable

---

SENSe:CORRection:CKIT:ECAL:DMEMory:CLEar <kitName>

(Write-only) Deletes user characterizations from PNA disk memory.

**Parameters**

<kitName> Optional String argument. ECal Model, User Characterization name + " ECal", and serial number of the ECal module, separated by spaces. See examples below.

If unspecified, ALL User Characterizations that are stored in PNA disk memory are deleted.

**Examples**

'These examples all use "MyUserChar" as the User characterization name.

'The "My User Char" characterization is deleted from disk memory.


'All User characterizations are deleted from disk memory.

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

SENSe:CORRection:CKIT:ECAL:DMEMory:IMPort <file>
After the PNA disk memory is Exported to a file, use this command to Import the file into PNA disk memory, which allows the User Characterization to be used with the PNA and ECal module.

**Note:** An ECal confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

**Parameters**

- `<file>` String. Full path and file name of file that was exported.

**Examples**

```plaintext
```

**Query Syntax** Not Applicable

**Default** Not Applicable

---

(SENSE:CORR:CKIT:ECAL:EXPort <kit>,<file>,<NewName>)

(Write-only) Saves an existing ECal characterization to a file. Use this command to archive the user characterization or to move the characterization to a different PNA for use with the specified ECal module. After exporting the user characterization, use **SENS:CORR:CKIT:ECAL:DMEM:IMPport** to make the user characterization available for use.

**Parameters**

- `<kit>` String. Not case sensitive. ECal Model, User char name + " ECal", and serial number of the ECal module used for the characterization, separated by spaces. See examples below.

  If the model and serial number of the module is not found, an error is returned.

- `<file>` Optional String argument. Path and filename of the user characterization. If not specified, the file is saved using characterization name + ".euc". If the path is not specified, it is stored in C:/Program Files/Agilent/Network Analyzer/ECal User Characterizations/. The extension ".euc" is appended if one is not specified.

- `<NewName>` Optional String argument. This allows you to change the name for the User Characterization. When specified, the new name is saved in the file with the characterization. If unspecified, the existing user characterization name is saved.

**Examples**

'**These examples all use "MyUserChar" as the User characterization name.**

'All parameters specified

(Read-only) This command is replaced with SENS:CORR:CKIT:ECAL:KNAM:INF?

Reads the user-characterization information from the specified ECal module. This command returns the same values as SENS:CORR:COLL:CKIT:INF?

**Parameters**

- **<mod>**  
  ECal module to read characterizations from. Choose from:
  
  1 through 8. If unspecified, value is set to 1.

- **<char>**  
  Optional argument. Specifies which characterization within the ECal module to read information from. If not specified, value is set to CHAR0.

Choose from:

- CHAR0  
  Factory characterization (data that was stored in the ECal module by Agilent)

- CHAR1  
  User characterization #1

- CHAR2  
  User characterization #2

- CHAR3  
  User characterization #3

- CHAR4  
  User characterization #4

- CHAR5  
  User characterization #5

Query Syntax  
**Default** Not Applicable
Examples

'Example return string:

ModelNumber: 85092-60007, SerialNumber: 01386, ConnectorType: N5FN5F, PortAConnector: Type N (50) female, PortBConnector: Type N (50) female, MinFreq: 30000, MaxFreq: 9100000000, NumberOfPoints: 250, Calibrated: July 4 2002

Return Type Character
Default Not Applicable

SENSe:CORRection:CKIT:ECAL:KNAMe:INFormation? <kitName>
(Read-only) This command replaces SENS:CORR:COLL:CKIT:INF?

Reads the user-characterization information from the specified ECal module or PNA disk memory.

Learn more about User Characterization in PNA Disk Memory.

Parameters

<kitName> String. ECal kit name and user characterization to read information from.

Optionally include the serial number in the <kit name>. Include the serial number when two or more ECal modules with same model number are attached to the PNA, or the PNA may not be referencing the intended ECal module.

Examples

'For a factory characterization in module memory:

'For user characterization in module memory:

'For user characterization in disk memory:

'Example return string:

ModelNumber: 85092-60007, SerialNumber: 01386, ConnectorType: N5FN5F, PortAConnector: Type N (50) female, PortBConnector: Type N (50) female, MinFreq: 30000, MaxFreq: 9100000000, NumberOfPoints: 250, Calibrated: July 4 2002

Return Type Character
Default Not Applicable
**SENSe:CORRection:CKIT:ECAL:LIST?**

(Read-only) Returns a list of index numbers to be used for referring to the ECal modules that are currently attached to the PNA.

**Examples**

```
SENS:CORR:CKIT:ECAL:LIST?

'If 2 modules are attached to the PNA
'then the returned list will be:

1,2
```

**Return Type** Numeric list, separated by commas.

**Default** Not Applicable

---

**SENSe<ch>:CORRection:CKIT:ECAL<n>:ORIent? <pnaPort>[,<charN>]**

(Read-only) Returns the ECal port that is connected to the specified PNA port. A calibration does not have to be in process.

- `<ch>` Channel number that contains the frequency range to be calibrated.
- `<n>` ECal module number. If unspecified (only one ECal module is connected to the USB), `<n>` is set to 1. If two or more modules are connected, use **SENS:CORR:CKIT:ECAL:LIST?** to determine how many, and **SENS:CORR:CKIT:ECAL:INF?** to verify their identities.
- `<pnaPort>` PNA port number.
- `<charN>` Optional argument. If unspecified, factory data (CHAR0) is used. User Characterization number that matches the physical adapters/fixtures that are on the ECal module. This aids in determining the orientation of the ECal module.

Choose from:

- **CHAR0** Factory characterization (data that was stored in the ECal module by Agilent)
- **CHAR1** User characterization #1
- **CHAR2** User characterization #2
  
  and so forth up to:

- **CHAR12** User characterization #12

Beginning with A.08.33, up to 12 User Characterizations can be stored in a single ECal module. Previous releases allowed up to 5. [Learn more](#).
Examples

SENSe:CORRection:CKIT:ECAL1:ORI? 2
sense2:correction:ckit,ecal1:orient? 2, char2

Return Type

The returned ECal port number is a 1-based number: 1 = Port A, 2 = Port B, 3 = Port C, 4 = Port D.

Zero (0) is returned when the auto-orientation routine is unable to resolve the orientation.

Default
Not Applicable

SENSe:CORRection:CKIT:ECAL<n>:PATH:COUNt? <path>

(Read-only) Returns the number of unique states that exist for the specified path name on the selected ECal module.

This command performs exactly the same function as CONT:ECAL:MOD:PATH:COUNt?

Use the CONT:ECAL:MOD:PATH:STAT command to set the module into one of those states.

Use SENS:CORR:CKIT:ECAL:PATH:DATA? to read the data for a state.

Parameters

<n> USB number of the ECal module. If unspecified (only one ECal module is connected to the USB), <n> is set to 1. If two or more modules are connected, use SENS:CORR:CKIT:ECAL:LIST? to determine how many, and SENS:CORR:CKIT:ECAL:INF? to verify their identities.

<path> Name of the path for which to read number of states. Choose from:

Reflection paths

- A
- B
- C (4-port modules)
- D (4-port modules)

Transmission paths

- AB
- AC (4-port modules)
Examples

```
CONT:ECAL:MOD:PATH:COUNT?
control:ecal:module2:path:count?
```

Return Type
Integer

Default
Not Applicable

**SENSe<ch>:CORRection:CKIT:ECAL<num>:PATH:DATA? <path>, <stateNum>[,<char>]**

(Read-only) Returns the data for a state from the memory of the selected ECAl module. The returned data is interpolated if necessary to have the same stimulus values as the specified channel `<ch>`.

- For a reflection path state, the data is reflection S-parameter data. The number of values equals the number of stimulus points on the channel multiplied by 2 (because they are complex numbers).
- For a transmission path state, the data is all 4 S-parameters of the state. The number of values returned is 4 times that of a reflection state.

The data is returned in the same format as **CALC:DATA:SNP?**

**Note:** This command returns SNP data without header information, and in columns, not in rows as .SnP files. This means that the data returned from this command sends all frequency data, then all Sx1 magnitude or real data, then all Sx1 phase or imaginary data, and so forth.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<num>` Optional argument. USB number of the ECAl module. If unspecified (only one ECAl module is connected to the USB), `<num>` is set to 1. If two or more modules are connected, use **SENS:CORR:CKIT:ECAL:LIST?** to determine how many, and **SENS:CORR:CKIT:ECAL:INF?** to verify their identities.
- `<path>` Name of the path for which to read number of states. Choose from:

  Reflection paths
  
  - A
  - B
  - C (4-port modules)
- D (4-port modules)

Transmission paths

- AB
- AC (4-port modules)
- AD (4-port modules)
- BC (4-port modules)
- BD (4-port modules)
- CD (4-port modules)

<stateNum> Number of the state to set. Refer to the following table to associate the <stateNum> with a state in your ECal module.

In addition, **CONT:ECAL:MOD:PATH:COUNt?** returns the number of states in the specified ECal module.

<table>
<thead>
<tr>
<th>&lt;stateNum&gt;</th>
<th>N4432A and N4433A States</th>
<th>N4431A States</th>
<th>N469x States**</th>
<th>8509x States</th>
</tr>
</thead>
<tbody>
<tr>
<td>One-Port Reflection States</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Open</td>
<td>Open</td>
<td>Impedance 1</td>
<td>Open</td>
</tr>
<tr>
<td>2</td>
<td>Short</td>
<td>Short</td>
<td>Impedance 2</td>
<td>Short</td>
</tr>
<tr>
<td>3</td>
<td>Impedance 1</td>
<td>Impedance 1</td>
<td>Impedance 3</td>
<td>Impedance 1</td>
</tr>
<tr>
<td>4</td>
<td>Impedance 2</td>
<td>Impedance 2</td>
<td>Impedance 4</td>
<td>Impedance 2</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td>Impedance 5</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td>Impedance 6</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td>Impedance 7</td>
<td></td>
</tr>
</tbody>
</table>

Two-Port Transmission States

| 1          | Thru                      | Thru          | Thru           | Thru         |
** The following modules have only FOUR Impedance states (1, 2, 3, 4): N4690B, N4691B, N4692A, N4696B.

<char> Optional argument. Specifies which characterization within the ECal module to read information from. If not specified, value is set to CHAR0.

Choose from:

- **CHAR0** Factory characterization (data that was stored in the ECal module by Agilent)
- **CHAR1** User characterization #1
- **CHAR2** User characterization #2
- and so forth up to:

- **CHAR12** User characterization #12

**Examples**

```
```

**Return Type** S1P or S2P

**Default** Not Applicable

---

**SENSe:CORRRection:CKIT:EXPort <kit>[,<file>]**

(Write-only) Saves an existing cal kit definitions to a file. Use this command to archive or move a user-defined or modified cal kit to a different PNA. After exporting the cal kit, use **SENS:CORR:CKIT:IMPort** to make the cal kit available for use on the PNA. This command provides the same behavior as the Installed Kits - Save As button on the Edit PNA Cal Kits dialog.

**Parameters**

- **<kit>** String. Not case sensitive. Name of the cal kit to export, as seen in the Cal Kits field of the Select DUT Connectors and Cal Kits dialog of a SMART Cal.
- **<file>** Optional String argument. Path and filename to where the Cal Kit file is to be saved. If not specified, the file is saved using <kit> + ".ckt". If the path is not specified, it is stored in C:/Program Files/Agilent/Network Analyzer/PNACalKits/User.

**Examples**

- 'File unspecified
  **SENS:CORR:CKIT:EXP "MyCalKit"**
- 'Both parameters are specified
SENSe:CORRection:CKIT:IMPort <string>

(Write-only) Imports the specified cal kit (.ckt file) and appends the imported kit to the end of the list of kits.

Parameters

<string> Path and cal kit name.

Examples

SENSe:CORRection:CKIT:IMPort "C:/Program Files/Agilent/Network Analyzer/Documents/85033D.ckt"

SENSe:CORRection:CKIT:INITialize[:IMMediate] [ckit]

(Write-only) Restores default factory installed cal kits.

Note: This command can also delete all existing User-defined Cal Kits. However, if saved using Save As, these kits can be restored in the same manner as after a PNA firmware upgrade. Learn more about saving modified Cal Kits.

Parameters

[ckit] Optional String. Cal Kit to restore. If not specified, all PNA factory Cal kits are restored.

Examples

SENSe:CORRection:CKIT:INITialize

SENSe:CORRection:ckit:initialize:immediate "85052B"

SENSe:CORRection:CKIT:LOAD <string>
(Write-only) Loads the specified collection of cal kits from a .wks file. You can make your own collection of cal kits from the Advanced Modify Cal Kit menu.

**Parameters**

- `<string>` Path and file name of the cal kit collection.

**Examples**

```
sense:correction:ckit:load "C:/Program Files/Agilent/Network Analyzer/PnaCalKits/factory/wMyCalKits.wks"
```

**Query Syntax** Not Applicable

**Default** Not Applicable

Last modified:

- **11-Apr-2011** Edited Orient? command
- **15-Jun-2010** Updated for 12 User Chars
- **19-May-2009** Added [ckit] argument to clear and init
- **24-Mar-2009** Edited Path SNP
- **6-Mar-2009** Added ECal orient?
- **31-Oct-2008** Added Characterizations (8.33)
- **16-Jun-2008** Added CKIT:INIT note
- **10/16/06** Modified Ecal:Data to include `<ch>`
**Sense:Correction:Collect:Kit Commands**

Use to change the definitions of calibration kit standards.

<table>
<thead>
<tr>
<th>SENSE:CORRection:COLLect:KIT:</th>
</tr>
</thead>
<tbody>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>CONNector</td>
</tr>
<tr>
<td>ADD</td>
</tr>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>DELete</td>
</tr>
<tr>
<td>FNAMe</td>
</tr>
<tr>
<td>SNAMe</td>
</tr>
<tr>
<td>DESCription</td>
</tr>
<tr>
<td>INFormation?</td>
</tr>
<tr>
<td>NAME</td>
</tr>
<tr>
<td>OLAB</td>
</tr>
<tr>
<td>OLIS?</td>
</tr>
<tr>
<td>ORDer</td>
</tr>
<tr>
<td>PORT[:SELect]</td>
</tr>
<tr>
<td>RESet</td>
</tr>
<tr>
<td>SELect</td>
</tr>
<tr>
<td>STANdard</td>
</tr>
<tr>
<td>CO, C1, C2, C3</td>
</tr>
<tr>
<td>CHARacter</td>
</tr>
<tr>
<td>DELay</td>
</tr>
<tr>
<td>FMAX</td>
</tr>
<tr>
<td>FMIN</td>
</tr>
<tr>
<td>IMPedance</td>
</tr>
<tr>
<td>LO, L1, L2, L3</td>
</tr>
</tbody>
</table>
Click on a **blue** keyword to view the command details.

**Red** keywords are superseded commands.

Most of these commands act on the currently selected standard from the currently selected calibration kit.

- To select a Calibration kit, use `SENS;CORR;COLL:CKIT:SEL`.
- To select a Calibration standard, use `SENS;CORR;COLL:CKIT:STAN:SEL`.
- See an **example** program that [CREATES a New Cal Kit](#).
- See an **example** program that [MODIFIES an Existing Cal Kit](#).
- Learn about Modifying Cal Kits.
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

**Note:** You should provide data for every definition field - for every standard in your calibration kit. If a field is not set, the default value may not be what you expect.

For more information, read [Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers (Application Note 1287-11)](#).

---

**SENSe:CORRection:COLLect:CKIT:CATalog?**
(Read-only) Returns the names of the first 50 mechanical cal kits in your PNA that can be used for unguided calibrations.

**Examples**
```
SENS:CORR:COLL:CKIT:CAT?
```

**Return Type**
A comma-separated string

**Default**
Not Applicable

---

**SENSe:CORRRection:COLLect:CKIT:CONNector:ADD**
```
<familiy>,<start>,<stop>,<z0>,<gender>,<media>,<cutoff>
```

(Write only) Creates a new connector. The connector is automatically added to the list of available connectors for the currently selected cal kit. If a connector includes both male and female connectors, each connector must be added separately.

**Parameters**
- `<family>` (String) Name of connector family. Limited to 50 characters.
- `<start>` Start frequency
- `<stop>` Stop frequency
- `<z0>` Characteristic Impedance of the connector in ohms.
- `<gender>` Connector gender. Choose from:
  - MALE
  - FEMALE
  - NONE
- `<media>` Media of the connector. Choose from:
  - COAX - coaxial
  - WAVE - waveguide
- `<cutoff>` Cutoff frequency of the connector (waveguide only).

**Examples**
```
SENS:CORR:COLL:CKIT:CONN:ADD "PSC 1.8 mm",0 HZ,999.9 GHZ,50,FEMALE,COAX,0.0
SENS:CORR:COLL:CKIT:CONN:ADD "PSC 1.8 mm",0 HZ,999.9 GHZ,50,MALE,COAX,0.0
```

**Query Syntax**
Not applicable

**Default**
Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:CONNector:CATalog?**
(Read-only) Returns a comma-separated list of all connectors defined within the currently selected calibration kit. The returned string includes the connector family name followed by the connector gender, if any. Kits may include a primary connector family name and additional connector family names.

Connector family names are case sensitive. A connector family named "PSC 2.4" is different from a connector family named "psc 2.4".

Learn more about Connector Family Name

**Examples**

```
SENS:CORR:COLL:CKIT:CONN:CAT?

'Returned string

"Type-N (50) male, Type-N (50) female"
```

**Default** Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:CONNector:DELete**

(Write-only) Deletes the primary connector family name from the selected kit. The PNA allows multiple connector families for each kit. If a kit includes multiple connector families, only the first listed (primary) connector family name is deleted.

Once the connector family is deleted, the connector may not be assigned to any new or existing standard within the kit.

The previously defined standards retain their association to the deleted connector name. To reassign standards to a new connector family name, use **SENS:CORR:COLL:CKIT:CONN:SNAMe**.

**Examples**

```
SENS:CORR:COLL:CKIT:CONN:DEL
```

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:CONNector:FNAME <name>**
(Read-Write) Replaces the primary connector family name from the selected kit with a new connector family name. The connector family name is replaced in all standards in the kit that share that name. The PNA allows multiple connector families for each kit. If a kit includes multiple connector families, only the first listed (primary) connector family name is replaced. Use the query form of this command to return the name of the primary connector family.

**Parameters**

- `<name>` New connector family name. Limited to 50 characters.

**Examples**

```
SENS:CORR:COLL:CKIT:CONN:FNAME 'MYPSC35'
```

```
Sense:correction:collect:ckit:connector:name 'My Type N'
```

**Query Syntax**

```
SENSe:CORRection:COLLect:CKIT:CONNector:FNAMe?
```

**Return Type**

String

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:CONNector:SNAMe <family>,<gender>,<port>**

(Read-Write) Assigns a family name to the currently selected standard from the currently selected kit. Specify each port of a 2-port standard individually. Use the query form of this command to read the connector family name assigned to the current standard. The name is not assigned unless the connector family name is previously defined within the selected kit.

**Parameters**

- `<family>` String. Connector family name.
- `<gender>` Connector gender. Choose from:
  - MALE
  - FEMALE
  - NONE
- `<port>` Number of the connector port to be assigned the connector family name. 2-port standards such as a thru line must be assigned separately. It is not relevant which connector is port 1 or port 2.
  - 1 Specifies a 1-port standard or the first port of a 2-port standard.
  - 2 Specifies the second port of a 2-port standard.

**Examples**

```
SENS:CORR:COLL:CKIT:CONN:SNAME "Type-N (50)",MALE,1
```

**Query Syntax**

```
SENSe:CORRection:COLLect:CKIT:CONNector:SNAME?
```

**Return Type**

String
SENSe:CORRection:COLLect:CKIT:DESCription <string>

(Read-Write) Modifies the cal kit description field of the selected kit. This description appears in the Edit PNA Cal Kit dialog box.

Parameters

<string> Description of the cal kit. Limited to 50 characters.

Examples

SENSe:CORRection:COLLect:CKIT:DESC "My New CalKit"

Query Syntax

SENSe:CORRection:COLLect:CKIT:DESCription?

Return Type

String

Default

Not Applicable

SENSe:CORRection:COLLect:CKIT:INFormation? <module>[,.char]

(Read Only) Reads characterization information from an ECal module.

Parameters

<string> Specifies which ECal module to read from. Choose from:

ECAL1

. through.

ECAL8

[char] Optional argument.

Specifies which characterization within the ECal module to read information from. If this argument is not used, the default is CHAR0. CHAR1 through CHAR5 are for user characterizations that may have been written to the module by the User Characterization feature on the PNA. Choose from:

CHAR0 Factory characterization (data that was stored in the ECal module by Agilent)

CHAR1 User characterization #1

CHAR2 User characterization #2

CHAR3 User characterization #3
SENSe:CORRection:COLLect:CKIT:NAME <name>

(Read-Write) Sets a name for the selected calibration kit.

Parameters

- `<name>` Calibration Kit name. Any string name, can include numerics, period, and spaces; any length (although the dialog box display is limited to about 30 characters).

Examples

SENSe:CORRection:COLLect:CKIT:NAME 'MYAPC35'
sense:correction:collect:ckit:name 'mytypen'

Query Syntax
SENSe:CORRection:COLLect:CKIT:NAME?

Return Type
String

Default
Not Applicable

SENSe:CORRection:COLLect:CKIT:OLABel<class> <name>

(Read-Write) Sets the label for the calibration class designated by `<class>`. The label is used in the prompts for connecting the calibration standards associated with that `<class>`.

Parameters

- `<class>` Number of the calibration class. Choose a number between: 1 and 18. The `<class>` numbers are associated with the following calibration Classes:
<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>S11A Reflection standard</td>
</tr>
<tr>
<td>2</td>
<td>S11B Reflection standard</td>
</tr>
<tr>
<td>3</td>
<td>S11C Reflection standard</td>
</tr>
<tr>
<td>4</td>
<td>S21T Thru/Delay standard</td>
</tr>
<tr>
<td>5</td>
<td>S22A Reflection standard</td>
</tr>
<tr>
<td>6</td>
<td>S22B Reflection standard</td>
</tr>
<tr>
<td>7</td>
<td>S22C Reflection standard</td>
</tr>
<tr>
<td>8</td>
<td>S12T Thru/Delay standard</td>
</tr>
</tbody>
</table>

### 3-port analyzers only

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>S33A Reflection standard</td>
</tr>
<tr>
<td>10</td>
<td>S33B Reflection standard</td>
</tr>
<tr>
<td>11</td>
<td>S33C Reflection standard</td>
</tr>
<tr>
<td>12</td>
<td>S32T Thru/Delay standard</td>
</tr>
<tr>
<td>13</td>
<td>S23T Thru/Delay standard</td>
</tr>
<tr>
<td>14</td>
<td>S31T Thru/Delay standard</td>
</tr>
<tr>
<td>15</td>
<td>S13T Thru/Delay standard</td>
</tr>
</tbody>
</table>

### TRL Calibrations

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>TRL &quot;T&quot; Thru standard</td>
</tr>
<tr>
<td>17</td>
<td>TRL &quot;R&quot; Reflect standard</td>
</tr>
<tr>
<td>18</td>
<td>TRL &quot;L&quot; Line standard</td>
</tr>
</tbody>
</table>

<name> Label for the calibration class. Must be enclosed in quotes. Any string between 1 and 12 characters long. Cannot begin with a numeric.

Examples

```
SENS:CORR:COLL:CKIT:OLAB3 'LOADS'
sense:correction:collect:ckit:olabel4 'Thru'
```

**Return Type** String

**Default** Not Applicable

**SENSe:CORRection:COLLect:CKIT:OLIST[class]?**

*(Read-only)* Returns seven values of standards that are assigned to the specified class.

**Parameters**

<class> Number of the calibration class to be queried. The <class> numbers are associated with the following calibration Classes:
<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>S11A Reflection standard</td>
</tr>
<tr>
<td>2</td>
<td>S11B Reflection standard</td>
</tr>
<tr>
<td>3</td>
<td>S11C Reflection standard</td>
</tr>
<tr>
<td>4</td>
<td>S21T Thru/Delay standard</td>
</tr>
<tr>
<td>5</td>
<td>S22A Reflection standard</td>
</tr>
<tr>
<td>6</td>
<td>S22B Reflection standard</td>
</tr>
<tr>
<td>7</td>
<td>S22C Reflection standard</td>
</tr>
<tr>
<td>8</td>
<td>S12T Thru/Delay standard</td>
</tr>
</tbody>
</table>

3-port analyzers ONLY

4-port analyzers use S11 and S22 classes *(see example program)*

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>S33A Reflection standard</td>
</tr>
<tr>
<td>10</td>
<td>S33B Reflection standard</td>
</tr>
<tr>
<td>11</td>
<td>S33C Reflection standard</td>
</tr>
<tr>
<td>12</td>
<td>S32T Thru/Delay standard</td>
</tr>
<tr>
<td>13</td>
<td>S23T Thru/Delay standard</td>
</tr>
<tr>
<td>14</td>
<td>S31T Thru/Delay standard</td>
</tr>
<tr>
<td>15</td>
<td>S13T Thru/Delay standard</td>
</tr>
</tbody>
</table>

**TRL Calibrations**

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>TRL &quot;T&quot; Thru standard</td>
</tr>
<tr>
<td>17</td>
<td>TRL &quot;R&quot; Reflect standard</td>
</tr>
<tr>
<td>18</td>
<td>TRL &quot;L&quot; Thru standard</td>
</tr>
</tbody>
</table>

**Examples**

<table>
<thead>
<tr>
<th>Examples</th>
<th>SENS:CORR:COLL:CKIT:OLIST8?</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Always returns 7 standard numbers. Unassigned standards return 0</td>
</tr>
</tbody>
</table>

**Return Type**

Numeric; returns the `<class>` number of the selected standard.

**Default**

Not Applicable

(Read-Write) Sets a standard number to a calibration class. Does NOT set or dictate the order for measuring the standards. For more information, see Assigning Standards to a Calibration Class

Parameters

$class$ Number of the calibration class that is assigned to $<standard>$. Choose a number between: 1 and 18. The $<class>$ numbers are associated with the following calibration Classes:

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>S11A Reflection standard</td>
</tr>
<tr>
<td>2</td>
<td>S11B Reflection standard</td>
</tr>
<tr>
<td>3</td>
<td>S11C Reflection standard</td>
</tr>
<tr>
<td>4</td>
<td>S21T Thru/Delay standard</td>
</tr>
<tr>
<td>5</td>
<td>S22A Reflection standard</td>
</tr>
<tr>
<td>6</td>
<td>S22B Reflection standard</td>
</tr>
<tr>
<td>7</td>
<td>S22C Reflection standard</td>
</tr>
<tr>
<td>8</td>
<td>S12T Thru/Delay standard</td>
</tr>
</tbody>
</table>

3-port analyzers ONLY

4-port analyzers use S11 and S22 classes (see example program)

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>S33A Reflection standard</td>
</tr>
<tr>
<td>10</td>
<td>S33B Reflection standard</td>
</tr>
<tr>
<td>11</td>
<td>S33C Reflection standard</td>
</tr>
<tr>
<td>12</td>
<td>S32T Thru/Delay standard</td>
</tr>
<tr>
<td>13</td>
<td>S23T Thru/Delay standard</td>
</tr>
<tr>
<td>14</td>
<td>S31T Thru/Delay standard</td>
</tr>
<tr>
<td>15</td>
<td>S13T Thru/Delay standard</td>
</tr>
</tbody>
</table>

TRL Calibration

<table>
<thead>
<tr>
<th>Class</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>TRL &quot;T&quot; Thru standard</td>
</tr>
<tr>
<td>17</td>
<td>TRL &quot;R&quot; Reflect standard</td>
</tr>
<tr>
<td>18</td>
<td>TRL &quot;L&quot; Line standard</td>
</tr>
</tbody>
</table>
Standard number to be assigned to the class; Choose a standard between 1 and 8. One standard is mandatory; up to six additional standards are optional.

**Examples**
- Assigns standard 3 to S11A class:
  ```
  SENS:CORR:COLL:CKIT:ORD1 3
  ```
- Assigns standard 2 and 5 to S21T class class:
  ```
  sense:correction:collect:ckit:order4 2,5
  ```

**Query Syntax**
- `SENSe:CORRection:COLLect:CKIT:ORDer<class>?`
  - 'Returns only the first standard assigned to the specified class. To query the remaining standards, use `SENSe:CORRection:COLLect:CKIT:OLIST[1-15]`?

**Return Type**
- Numeric
- **Default** Not Applicable

---

**SENSe<cnum>:CORRection:COLLect:CKIT:PORT<n>[:SELect] <string>**

(Read-Write) Sets and returns the name of the Cal Kit to use for unguided cal.

**Parameters**
- `<cnum>` Currently not used. The unguided cal kit selection is for all ports on all channels.
- `<n>` Currently not used. The unguided cal kit selection is for all ports on all channels.
- `<string>` Cal Kit name enclosed in quotes. Use `SENS:CORR:COLL:CKIT:CAT?` to read a list of all available Cal Kits in the PNA.

**Examples**
- `SENS:CORR:COLL:CKIT:PORT "85052B"
- `sense2:correction:collect:ckit:port:select "85052D"

**Query Syntax**
- `SENSe<cnum>:CORRection:COLLect:CKIT:PORT<n>:SELECT?`
- **Return Type** String
- **Default** Last kit selected

---

**SENSe:CORRection:COLLect:CKIT:RESet <num>** - Superseded
This command is replaced by Sens:Corr:Ckit:Init.

(Write-only) Resets the selected calibration kit to factory default definition values.

**Parameters**

- `<num>` The number of the calibration kit to be reset. Choose any integer between 1 and 8.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:CORR:COLL:CKIT:RESet 1</td>
</tr>
<tr>
<td>sense:correction:collect:ckit:reset 4</td>
</tr>
</tbody>
</table>

**Query Syntax**

- Not Applicable

**Default**

- Not Applicable

---

**SENSe<cnum>:CORRection:COLLect:CKIT[:SELect] <num>**

(Read-Write) Selects (makes active) a calibration kit for performing an UNGUIDED calibration or for modifying standards. All subsequent "CKIT" commands that are sent apply to this selected calibration kit. Select a calibration standard using SENS:CORR:COLL:CKIT:STAN <num>. Kits 1 to approximately kit 37 are factory installed Cal Kits.

Use the Edit PNA Cal Kits dialog to see the name that is associated with each Kit ID <num>.

Another method is to iterate through the cal kits, select the kit using this command, then query the name using SENS:CORR:COLL:CKIT:NAME.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<num>` The number of the calibration kit. Choose from:

<table>
<thead>
<tr>
<th>Kit ID</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cal Kit 1</td>
</tr>
<tr>
<td>2</td>
<td>Cal Kit 2</td>
</tr>
<tr>
<td>3</td>
<td>Cal Kit 3</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>Cal Kit 48</td>
</tr>
<tr>
<td>49</td>
<td>Cal Kit 49</td>
</tr>
<tr>
<td>50</td>
<td>Cal Kit 50</td>
</tr>
</tbody>
</table>

Use SENS:CORRection:COLLect:CKIT:RESet to restore Cal Kits to default values.
**SEnSe:CORRection:COLLect:CKIT:STANdard:C0 <num>**

*(Read-Write)* Sets the C0 value (the first capacitance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at [www.Agilent.com](http://www.Agilent.com).

**Parameters**

- `<num>` Value for C0 in femtofarads (1E-15)

**Examples**

The following commands set C0=15 femtofarads:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:C0?

**Return Type** Numeric

**Default** Not Applicable

---

**SEnSe:CORRection:COLLect:CKIT:STANdard:C1 <num>**

*(Read-Write)* Sets the C1 value (the second capacitance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at [www.Agilent.com](http://www.Agilent.com).

**Parameters**

- `<num>` Value for C1.

**Examples**

The following two commands set C1=15:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:C1?

**Return Type** Numeric

**Default** Not Applicable
SENSe:CORRection:COLLect:CKIT:STANdard:C2 <num>

(Read-Write) Sets the C2 value (the third capacitance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at www.Agilent.com.

**Parameters**

- `<num>` Value for C2.

**Examples**
The following two commands set C2:

```
SENS:CORR:COLL:CKIT:STAN:C2 15
sense:correction:collect:ckit:standard:c2 15
```

**Query Syntax**
SENSe:CORRection:COLLect:CKIT:STANdard:C2?

**Return Type**
Numeric

**Default**
Not Applicable

---

SENSe:CORRection:COLLect:CKIT:STANdard:C3 <num>

(Read-Write) Sets the C3 value (the fourth capacitance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at www.Agilent.com.

**Parameters**

- `<num>` Value for C3.

**Examples**
The following two commands set C3:

```
SENS:CORR:COLL:CKIT:STAN:C3 15
sense:correction:collect:ckit:standard:c3 15
```

**Query Syntax**
SENSe:CORRection:COLLect:CKIT:STANdard:C3?

**Return Type**
Numeric

**Default**
Not Applicable

---

SENSe:CORRection:COLLect:CKIT:STANdard:CHARacter <char>
(Read-Write) Sets the media type of the selected calibration standard.

**Parameters**

<char> Media type of the standard. Choose from:

- **Coax** - Coaxial Cable
- **Wave** - Waveguide

**Examples**

```
SENS:COR:COLL:CKIT:STAN:CHAR COAX
sense:correction:collect:ckit:standard:character wave
```

**Query Syntax**

```
SENSe:CORRection:COLLect:CKIT:STANdard:CHARacter?
```

**Return Type** Numeric

**Default** Coax

---

**SENSe:CORRection:COLLect:CKIT:STANdard:DELay <num>**

(Read-Write) Sets the electrical delay value for the selected standard.

**Parameters**

<num> Electrical delay in picoseconds

**Examples**

```
The following two commands set delay to 50 picoseconds

SENS:CORR:COLL:CKIT:STAN:DEL 50e-12
sense2:correction:collect:ckit:standard:delay 50ps
```

**Query Syntax**

```
SENSe:CORRection:COLLect:CKIT:STANdard:DELay?
```

**Return Type** Numeric

**Default** Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:FMAXimum <num>**
**SENSe:CORRection:COLLect:CKIT:STANdard:FMAXimum <num>**

(Read-Write) Sets the maximum frequency for the selected standard.

**Parameters**

<num>  Maximum frequency in Hertz.

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:FMAX 9e9  
sense:correction:collect:ckit:standard:fmaximum 9Ghz
```

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:FMAXimum?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:FMINimum <num>**

(Read-Write) Sets the minimum frequency for the selected standard.

**Parameters**

<num>  Minimum frequency in Hertz.

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:FMIN 1e3  
sense:correction:collect:ckit:standard:fminimum 1khz
```

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:FMINimum?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:IMPedance <num>**

(Read-Write) Sets the characteristic impedance for the selected standard.

**Parameters**

<num>  Impedance in Ohms

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:IMP 75  
sense:correction:collect:ckit:standard:impedance 50.3
```

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:IMPedance?

**Return Type**

Numeric

**Default**

50

---

**SENSe:CORRection:COLLect:CKIT:STANdard:L0 <num>**
**SENSe:CORRection:COLLect:CKIT:STANdard:L0**

*(Read-Write)* Sets the L0 value (the first inductance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at [www.Agilent.com](http://www.Agilent.com).

**Parameters**

- `<num>`: Value for L0 in femtohenries (1E-15)

**Examples**

The following two commands set L0=15 femtohenries:

```
SENS:CORR:COLL:CKIT:STAN:L0 15
sense:correction:collect:ckit:standard:l0 15
```

**Query Syntax**

`SENSe:CORRection:COLLect:CKIT:STANdard:L0?`

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:L1** `<num>`

*(Read-Write)* Sets the L1 value (the second inductance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at [www.Agilent.com](http://www.Agilent.com).

**Parameters**

- `<num>`: Value for L1.

**Examples**

The following two commands set L1=15:

```
SENS:CORR:COLL:CKIT:STAN:L1 15
sense:correction:collect:ckit:standard:l1 15
```

**Query Syntax**

`SENSe:CORRection:COLLect:CKIT:STANdard:L1?`

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:L2** `<num>`
(Read-Write) Sets the L2 value (the third inductance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at www.Agilent.com.

**Parameters**

<num> Value for L2.

**Examples**
The following two commands set L2=15:

```
SENS:CORR:COLL:CKIT:STAN:L2 15
sense:correction:collect:ckit:standard:l2 15
```

**Query Syntax**
SENSe:CORRection:COLLect:CKIT:STANdard:L2?

**Return Type**
Numeric

**Default**
Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:L3 <num>**

(Read-Write) Sets the L3 value (the fourth inductance value) for the selected standard. For a detailed discussion of this value, search for App Note 8510-5B at www.Agilent.com.

**Parameters**

<num> Value for L3.

**Examples**
The following two commands set L3=15:

```
SENS:CORR:COLL:CKIT:STAN:L3 15
sense:correction:collect:ckit:standard:l3 15
```

**Query Syntax**
SENSe:CORRection:COLLect:CKIT:STANdard:L3?

**Return Type**
Numeric

**Default**
Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:LABel <name>**
(Read-Write) Sets the label for the selected standard. The label is used to prompt the user to connect the specified standard.

**Parameters**

- `<name>` Label for the standard; Must be enclosed in quotes. Any string between 1 and 12 characters long. Cannot begin with a numeric.

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:LAB 'OPEN'
sense:correction:collect:ckit:standard:label 'Short2'
```

**Query Syntax**

`SENSe:CORRection:COLLect:CKIT:STANdard:LABel?`

**Return Type**

String

**Default**

Not Applicable

---

`SENSe:CORRection:COLLect:CKIT:STANdard:LOSS <num>`

(Read-Write) Sets the insertion loss for the selected standard.

**Parameters**

- `<num>` Insertion loss in Gohms / sec. (GigaOhms per second of electrical delay)

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:LOSS 3.5e9
sense:correction:collect:ckit:standard:loss 3
```

**Query Syntax**

`SENSe:CORRection:COLLect:CKIT:STANdard:LOSS?`

**Return Type**

Numeric

**Default**

Not Applicable

---

`SENSe:CORRection:COLLect:CKIT:STANdard:REMove`

(Write only) Deletes the selected standard from the selected cal kit.

**Examples**

```
SENS:CORR:COLL:CKIT:STAN:REMove
```

**Default**

Not Applicable

---

`SENSe:CORRection:COLLect:CKIT:STANdard:SDEscription <string>`
(Read-Write) Modifies the description of the selected standard of the selected kit. This description appears in the edit kit dialog box.

**Parameters**

<string> Description of the standard.

**Examples**

```
```

**Query Syntax**

```
SENS:CORR:COLL:CKIT:STAN:SDEScription?
```

**Return Type** String

**Default** Not Applicable

---

**SENS:CORR:COLL:CKIT:STANdard[:SELECT] <num>**

(Read-Write) Selects the calibration standard. All subsequent "CKIT" commands to modify a standard will apply to the selected standard. Select a calibration kit using SENS:CORR:COLL:CKIT:SEL

**Parameters**

<num> Number of the standard. Choose any number between: 1 and 30

**Examples**

```
SENS:CORR:COLL:CKIT:STAN 3
sense:correction:collect:ckit:standard:select 8
```

**Query Syntax**

```
SENS:CORR:COLL:CKIT:STANdard[:SELect]?
```

**Return Type** Numeric

**Default** 1

---

**SENS:CORR:COLL:CKIT:STANdard:TYPE <char>**
(Read-Write) Sets the type for the selected standard.

**Parameters**

<char> Choose from:
- OPEN
- SHORT
- LOAD
- SLOAD (sliding load)
- THRU (through)
- ARBI (arbitrary)

**Examples**

SENS:CORR:COLL:CKIT:STAN:TYPE LOAD
sense:correction:collect:ckit:standard:type short

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:TYPE?

**Return Type**

Character

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:TZReal <num>**

(Read-Write) Sets the TZReal component value of the Terminal Impedance for the selected standard.

**Note:** Only applicable when the Standard Type is set to **ARBI**

**Parameters**

<num> Value for TZReal in Ohms

**Examples**

The following commands set TZReal=15 Ohms:

SENS:CORR:COLL:CKIT:STAN:TZReal 15
sense:correction:collect:ckit:standard:TZReal 15

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:TZReal?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe:CORRection:COLLect:CKIT:STANdard:TZImag <num>**
(Read-Write) Sets the TZImag component value of the Terminal Impedance for the selected standard.

**Note:** Only applicable when the Standard Type is set to **ARBI**

**Parameters**

<num> Value for TZImag in Ohms

**Examples**

The following two commands set TZImag=15 Ohms:

```
SENS:CORR:COLL:CKIT:STAN:TZImag 15
sense:correction:collect:ckit:standard:TZImag 15
```

**Query Syntax**

SENSe:CORRection:COLLect:CKIT:STANdard:TZImag?

**Return Type**

Numeric

**Default**

Not Applicable

---

Last Modified:

<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11-Jan-2011</td>
<td>Minor edits</td>
</tr>
<tr>
<td>17-Mar-2010</td>
<td>Added Ckit:CAT and CKIT:Select by string (9.2)</td>
</tr>
<tr>
<td>10-Nov-2008</td>
<td>Clarified Select command</td>
</tr>
<tr>
<td>30-Oct-2008</td>
<td>Fixed SDES query per CN</td>
</tr>
<tr>
<td>14-Apr-2008</td>
<td>Added link to app note</td>
</tr>
<tr>
<td>17-Sep-2007</td>
<td>Fixed 'select' command</td>
</tr>
</tbody>
</table>
**Sense:Correction:Cset Commands**

Performs actions on calibration sets.

<table>
<thead>
<tr>
<th>SENSe:CORRection:CSET</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTivate</td>
</tr>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>COPY</td>
</tr>
<tr>
<td>CREate</td>
</tr>
<tr>
<td>DATA</td>
</tr>
<tr>
<td>DELete</td>
</tr>
<tr>
<td>DESCription</td>
</tr>
<tr>
<td>ETERm</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>FLATten</td>
</tr>
<tr>
<td>GUID</td>
</tr>
<tr>
<td>NAME</td>
</tr>
<tr>
<td>[SELect]</td>
</tr>
<tr>
<td>SAVE</td>
</tr>
<tr>
<td>STANdard</td>
</tr>
<tr>
<td>STIMulus?</td>
</tr>
<tr>
<td>TSET</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>TYPE</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
Click on a blue keyword to view the command details.

See Also

- Creating Cal Sets
- Example Programs
- Learn about Cal Sets
- Synchronizing the PNA and Controller

SENSe<cnum>:CORRection:CSET:ACTivate <string>, <bool>

This command replaces SENS:CORR:CSET:GUID

(Read-Write) Selects and applies a Cal Set to the specified channel.

Use SENS:CORR:CSET:CAT? to list the Cal Sets.

Parameters

- **<cnum>**  Any existing channel number. If unspecified, value is set to 1
- **<string>**  Cal Set to make active. Specify the Cal Set by GUID or Name. Use SENS:CORR:CSET:CAT? to list the available Cal Sets in either format.
- **<bool>**  Should the Cal Set stimulus values be applied to the channel. Choose from:
  - **ON (1)**  Apply the Cal Set stimulus values to the channel.
  - **OFF (0)**  Do NOT apply the Cal Set stimulus values. If the Cal Set stimulus values do not match the channel stimulus values, then the following will occur:
    - If interpolation is ON, then interpolation will be attempted. This may fail if the channel frequency is outside the range of the Cal Set.
    - If interpolation is OFF, the selection will be abandoned and an error is returned:

Examples

```
SENS:CORR:CSET:ACT "My2Port",1
sense:correction:cset:activate? name
returns
"My2Port"
```

Query Syntax

SENSe<cnum>:CORRection:CSET:ACTivate? [GUID|NAME]

Returns the name of the Cal Set that is applied to the specified channel. Choose from GUID or NAME to specify which string is returned. If unspecified, the
GUID of the Cal Set is returned. If no Cal Set is applied to the specified channel, then "No Calset Selected" is returned.

<table>
<thead>
<tr>
<th>Return Type</th>
<th>String</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**SENSe:CORRection:CSET:CATalog? [char]**

(Read-only) Returns a list of Cal Sets.

**Parameters**

- `<char>` Optional argument. The list is returned in one of the following formats. Both return comma-separated string lists.
  - **GUID** Cal Sets are listed by GUID (Default if unspecified).
  - **NAME** Cal Sets are listed by Name

**Examples**

```
SENS:CORR:CSET:CAT?

'Returns:
{FD6F863E-9719-11d5-8D6C-00108334AE96},{1B03B2CE-971A-11d5-8D6C-00108334AE96}

sense2:correction:cset:catalog? name
```

<table>
<thead>
<tr>
<th>Default</th>
<th>Not Applicable</th>
</tr>
</thead>
</table>

**SENSe<cnum>:CORRection:CSET:COPY <string>**

(Write-only) Creates a new Cal Set and copies the current Cal Set data into it. Use this command to manipulate data on a Cal Set without corrupting the original cal data.

**Parameters**

- `<cnum>` Channel number using the Cal Set to be copied. If unspecified, value is set to 1
- `<string>` Name of the new Cal Set.

**Examples**

```
SENS2:CORR:CSET:COPY 'My2Port'
```

**Query Syntax** Not Applicable

<table>
<thead>
<tr>
<th>Default</th>
<th>Not Applicable</th>
</tr>
</thead>
</table>
SENSe<cnum>:CORRection:CSET:CREate [name]

(Write-only) Creates an empty Cal Set and attaches it to the specified channel. This command is ONLY necessary before remotely filling the Cal Set with error term data. (For Advanced Users).

A Cal Set is automatically created, applied to the channel, and saved at the completion of a guided cal according to the preference setting SENS:CORR:PREF:CSET:SAVE.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `[name]` Optional argument. Name of the Cal Set. Spaces or punctuation are NOT allowed. If unspecified, a unique name is chosen in the form "Calset_N" where N is a unique number.

**Examples**

SENSe:CORR:CSET:CRE 'My2Port'

**Query Syntax**

Not Applicable

**Default**

Not Applicable

SENSe<cnum>:CORRection:CSET:DATA <eterm, portA, portB,>[<param>,] <block>

(Read-Write) Read or Write a specific error term from/to the Cal Set currently attached to the specified channel. (For Advanced Users). The command can be used only for the error terms listed. See SENS:CORR:CSET:ETERM to get and put error term data using a string argument for all error terms.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<eterm, portA, portB>` Error Term, Port pair of the specified error term.
  - Although not all error terms use two port numbers, two are required by the PNA in all cases. Each port number must be between 1 and the number of ports on the PNA.
  - **EDIR** - directivity
    - portA: the port at which directivity is measured.
    - portB: Not used, but must be a valid PNA port number.
  - **ESRM** - source match
    - portA: the port at which source match is measured.
    - portB: Not used, but must be a valid PNA port number.
**ERFT** - reflection tracking

portA: the port at which reflection tracking is measured.

portB: Not used, but must be a valid PNA port number.

**ELDM** - load match

portA: the port at which load match is measured.

portB: the source port.

Load match is measured with a cable connected between the measured port (portA) and the source port (portB).

The cal system requires that the complete matrix of loadmatch arrays be filled. In most cases you can measure loadmatch once at a port, driven by any other port. Then use that data for all variations of the receive port. (The exception is the 3-port PNA models, which requires the loadmatch-measured port to be driven by every other port.)

For example: Measure the loadmatch at port2 while driving port1. Then upload this same data to the following arrays:

```
ELDM,2,1,<data>
ELDM,2,3,<data>
ELDM,2,4,<data>
```

**ETRT** - transmission tracking

portA: the receive port

portB: the source port for this measurement

**EXTLK** - crosstalk

portA: the receive port

portB: the source port for this measurement

**ERSPT** - response tracking.

portA: Not used, but must be a valid PNA port number.

portB: Not used, but must be a valid PNA port number.
ERSPI - response isolation.

portA: Not used, but must be a valid PNA port number.

portB: Not used, but must be a valid PNA port number.

<string> - required ONLY when Eterm is response tracking (ERSPT) or response isolation (ERSPI). Specify the S-parameter, ratio, or unratioed measurement for which the Eterm applies.

Logical receiver notation is allowed. Ratioed measurements do not require source port to be specified.

A full 4-port calibration requires the following terms be uploaded:

<table>
<thead>
<tr>
<th>PORT B</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>EDIR,1,1</td>
<td>ELDM,1,2</td>
<td>ELDM,1,3</td>
<td>ELDM,1,4</td>
</tr>
<tr>
<td></td>
<td>ERFT,1,1</td>
<td>ETRT,1,2</td>
<td>ETRT,1,3</td>
<td>ETRT,1,4</td>
</tr>
<tr>
<td></td>
<td>ESRM,1,1</td>
<td>EXTLK,1,2</td>
<td>EXTLK,1,3</td>
<td>EXTLK,1,4</td>
</tr>
<tr>
<td>2</td>
<td>ELDM,2,1</td>
<td>EDIR,2,2</td>
<td>ELDM,2,3</td>
<td>ELDM,2,4</td>
</tr>
<tr>
<td></td>
<td>ETRT,2,1</td>
<td>ERFT,2,2</td>
<td>ETRT,2,3</td>
<td>ETRT,2,4</td>
</tr>
<tr>
<td></td>
<td>EXTLK,2,1</td>
<td>ESRM,2,2</td>
<td>EXTLK,2,3</td>
<td>EXTLK,2,4</td>
</tr>
<tr>
<td>3</td>
<td>ELDM,3,1</td>
<td>ELDM,3,2</td>
<td>EDIR,3,3</td>
<td>ELDM,3,4</td>
</tr>
<tr>
<td></td>
<td>ETRT,3,1</td>
<td>ETRT,3,2</td>
<td>ERFT,3,3</td>
<td>ETRT,3,4</td>
</tr>
<tr>
<td></td>
<td>EXTLK,3,1</td>
<td>EXTLK,3,2</td>
<td>ESRM,3,3</td>
<td>EXTLK,3,4</td>
</tr>
<tr>
<td>4</td>
<td>ELDM,4,1</td>
<td>ELDM,4,2</td>
<td>ELDM,4,3</td>
<td>EDIR,4,4</td>
</tr>
<tr>
<td></td>
<td>ETRT,4,1</td>
<td>ETRT,4,2</td>
<td>ETRT,4,3</td>
<td>ERFT,4,4</td>
</tr>
<tr>
<td></td>
<td>EXTLK,4,1</td>
<td>EXTLK,4,2</td>
<td>EXTLK,4,3</td>
<td>ESRM,4,4</td>
</tr>
</tbody>
</table>

Reflection terms

Transmission terms

<Block> (Block). Error term data. A Real / Imaginary data pair for each data point.

Format is set using FORM:DATA command.

For REAL binary formats, refer to Getting Data from the Analyzer using SCPI
Example

'Set the directivity term with a cal set using 5 points


Query Syntax
SENSe<cnum>:CORRection:CSET:DATA? <eterm,portA, portB>

Return Type
Block data

Default
Not Applicable

SENSe:CORRection:CSET:DELete <string>
(Write-only) Deletes a Cal Set from the set of available Cal Sets. This method immediately updates the Cal Set file on the hard drive. If the Cal Set is currently being used by a channel or does not exist, this request will be denied and an error is returned.

Parameters

<string> Cal Set to be deleted. Specify the Cal Set by GUID or Name. Use SENSe:CORRection:CSET:CAT? to list the available Cal Sets in either format.

Examples
sense2:correction:cset:delete 'MyCalSet'

Query Syntax
Not Applicable

Default
Not Applicable

SENSe<cnum>:CORRection:CSET:DESCription <string>
(Read-Write) Sets or returns the descriptive string assigned to the selected Cal Set. Change this string so that you can easily identify each Cal Set. Apply and select the Cal Set using SENSe:CORRection:CSET:ACT.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1
<string> The descriptive string associated with the currently-selected Cal Set

Examples
SENSe:CORRection:CSET:DESC 'MyCalSet'
sense2:correction:cset:description 'thisCalSet'

Query Syntax
SENSe<cnum>:CORRection:CSET:DESCription?

Return Type
String

Default
Not Applicable
SENSe<cnump>:CORRection:CSET:ETERm <string>,<r, i [r,i]...>

(Read-Write) Sets or returns error term data for all PNA measurements.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <string> (String) Error term to read or write. The error term is specified using the EXACT case-sensitive string displayed in the Cal Set Viewer utility. See SENS:CORR:CSET:DATA for a description of port numbers.

The following Noise figure error terms are listed for convenience:

- **RcvNoiseCorr_m_n** Noise correlation matrix of the noise receiver (a 2x2 complex matrix). The row and column indices m and n range from 1 to 2.
- **RcvT_m_n** T-matrix of the noise receiver (a 2x2 complex matrix). The row and column indices m and n range from 1 to 2.
- **GammaTuner_n** Reflection coefficient for impedance state n of the embedded noise tuner (Ecal module) in the port 1 source path. For the Agilent 4691 family of Ecal modules, n can range from 1 to 7.
- **<r, i>** (Block) Error term data. A Real / Imaginary data pair for each data point.

Format is set using FORM:DATA command.

For REAL binary formats, refer to Getting Data from the Analyzer using SCPI

Examples

```
SENS:CORR:CSET:ETERm "Directivity(1,1)", 0.237,-1.422, 0.513, 0.895  ' set directivity(source error term for 2 points
SENS:CORR:CSET:ETERm? "Directivity(1,1)"  ' read
```

Query Syntax

SENSe<cnump>:CORRection:CSET:ETERm? <string>

Return Type

Block data

Default Not Applicable

---

SENSe<cnump>:CORRection:CSET:ETERm:CATalog? <setNum>,<string>
(Read-only) Returns a list of error terms names found in the Cal Set containing the specified prefix.

**Parameters**

- `<setNum>` Set number of the calset. There can be more than one set of error terms in a Cal Set.
  
  - setNumber 0 contains the original set of error terms for a Cal Set.
  
  - setNumbers > 0 contain Interpolated error terms. Interpolated error terms are generated when interpolation is required and destroyed when no longer used. Learn about Interpolation.

To determine the Cal Set in use by a channel, see `SENS:CORR:CSET:NAME`?

- `<string>` (Optional argument) The string used to identify Cal Set data as belonging to a specific Cal Type. This string is used as a filter so that only the error term names of interest are returned. If the string is empty, all terms are returned.

**Examples**

```
SENS:CORR:CSET:ETER:CAT? 0,"Full 2 Port Cal (2,3)" 'Returns original error terms for a two port cal on ports 2 and 3
```

**Default** Not Applicable

---

**SENSe<cnum>:CORRection:CSET:FLATten <string>**

(Write-only) When a Cal Set that was produced by a calibration has been interpolated or otherwise modified (for example, by Fixturing operations) this command saves the modified Cal Set to the PNA hard drive so that it can be reused. There is no User Interface equivalent for this command.

**Background**

When a Cal Set is selected for use by a channel, the channel reads the Cal Set from disk (master Cal Set). If the channel aligns perfectly with the Cal Set, the master Cal Set is used directly. In this case, the active Cal Set is the master Cal Set.

When processing occurs on the error terms due to interpolation or modification due to the use of fixturing, the channel will generate a temporary "memory-resident" Cal Set. In this case, the active Cal Set is the memory-resident Cal Set. This FLATten command allows you to save the active Cal Set to disk.

Depending on the measurement conditions, this flattening of the Cal Set can improve performance, especially if the Cal Set is applied often (using multiple recall states) or used by many channels. Flattening a version of the Cal Set for each channel can avoid the interpolation or the fixturing processing that would otherwise occur when the Cal Set is selected or the instrument state is recalled.
You will have to manage the application of such a Cal Set as the PNA itself will have no way to
determine what processing had been done once the flatten command is used. For example, if fixture
de-embedding occurred prior to the flatten command, that Cal Set should then be applied WITHOUT
fixturing on, because fixturing is already embedded in that Cal Set. It is your responsibility to apply
the Cal Set properly.

If you want to repeatedly de-embed multiple networks (i.e. concatenate multiple 2-port de-embedding
files) you can use the flatten command to create a new master Cal Set after each de-embed, and
sequentially add additional de-embed networks.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Channel number on which the modified Cal Set resides. If unspecified, value is set to 1</td>
</tr>
<tr>
<td>&lt;string&gt;</td>
<td>Name of the new Cal Set. Spaces or punctuation NOT allowed.</td>
</tr>
</tbody>
</table>

**Examples**

```
SENS:CORR:CSET:FLAT "MyCalSet"
```

**Query Syntax**

```
SENSe<cnum>:CORRection:CSET:GUID?
```

Returns the GUID of the currently-selected Cal Set for the specified channel.

---

**SENS{<cnum>}:CORR{ection}:CSET{GUID} <string> Superseded**

This command is replaced by `SENS:CORR:CSET:ACTivate`.

(Read-Write) Selects the Cal Set identified by the string parameter (GUID) and applies it to the
specified channel.

- A Cal Set cannot be selected for a channel which is not ON.
- If the stimulus settings of the selected Cal Set differ from those of the selected channel, the instrument will automatically change the channel's settings to match the Cal Set.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnum&gt;</td>
<td>Any existing channel number. If unspecified, value is set to 1</td>
</tr>
<tr>
<td>&lt;string&gt;</td>
<td>GUID of the desired Cal Set. The curly brackets and hyphens must be included.</td>
</tr>
</tbody>
</table>

**Examples**

```
```

**Query Syntax**

```
SENSe<cnum>:CORRection:CSET:GUID?
```

Returns the GUID of the currently-selected Cal Set for the specified channel.
SENSe<cnum>:CORRection:CSET:NAME <string>

(Read-Write) Sets or queries the name of the Cal Set currently applied to the specified channel.

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<string>`: Name of the Cal Set. Spaces or punctuation NOT allowed.

**Examples**
```
SENSe:CORR:CSET:NAME 'MyCalSet'
sense2:correction:cset:name 'thisCalSet'
```

**Query Syntax**
SENSe<cnum>:CORRection:CSET:NAME?

**Return Type** String

**Default** Not Applicable

---

SENSe<cnum>:CORRection:CSET[:SElect] <char>  **Superseded**

This command is replaced by **MMEM:LOAD**

(Read-Write) Recalls a *.cst file from memory. The file name is "CSETx.cst" where x is the user number assigned to `<char>`. Learn more about .cst files

For more information on the file naming syntax, see the **MMEMory** subsystem.

**Note:** This command does NOT select a Cal Set for a channel. To select a Cal Set, use **SENSe:CORR:CSET:ACTivate**

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<char>`: Choose from:
  - **DEF**: Presets the analyzer
  - **USER01**: Restores User01 calibration data
  - **USER02**: Restores User02 calibration data
  - through...
  - **USER10**: Restores User10 calibration data
**SENSe<cnum>:CORRection:CSET:SAVE <char>**

This command is NOT necessary after completion of a calibration. A Cal Set is automatically created, applied to the channel, and saved at the completion of a guided cal according to the preference setting `SENSe:CORR:PREF:CSET:SAVE`. (Read Write)

Saves the channel's Cal Set to the PNA hard drive. For example, use this command after writing data to a Cal Set using `SENSe:CORR:CSET:DATA` (For Advanced Users).

The file name is saved as "CSET:x.cst" where x is the user number assigned to <char>, and .cst specifies a Cal Set and instrument state. This is not the same syntax as a file saved through the default choices from the front panel, which is "at00x.cst". For more information on the file naming syntax, see the MMEMory subsystem. Learn more about Instrument/Cal States.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Choose from:
  - USER01
  - USER02...
  - USER10

**Examples**

`SENSe:CORR:CSET:SAVE USER03`

`sense2:correction:cset:save user09`

**Query Syntax**

`SENSe<cnum>:CORRection:CSET:SAVE?`

Queries the last correction set saved.

**Return Type**

Character

**Default** Not applicable

**SENSe<cnum>:CORRection:CSET:STANdard <string>,<data>**
(Read-Write) Sets or returns standard data. Standard data is available for Unguided Cals ONLY.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<string>` (String) Cal standard to read or write. The standard is specified using the EXACT case-sensitive string displayed in the [Cal Set Viewer](#) utility. See [SENS:CORR:CSET:DATA](#) for a description of port numbers.
- `<data>` (Block). Acquisition data. A Real / Imaginary data pair for each data point.

Format is set using [FORM:DATA](#) command.

For REAL binary formats, refer to [Getting Data from the Analyzer using SCPI](#)

**Examples**

```plaintext
SENS:CORR:CSET:STAN 'S11C(1,1), 0.237,-1.422, 0.513, 0.895 '  
Set acquisition data for two points.

SENS:CORR:CSET:STAN? "S11C(1,1)"  
'Read data
```

**Query Syntax**

`SENSe<cnum>:CORRection:CSET:STANdard? (string)`

**Return Type**

Block data

**Default** Not Applicable

---

**SENSe<ch>:CORRection:CSET:STIMulus? [num]**

(Read-only) Returns the source or response stimulus values for the Cal Set that is currently used by channel `<ch>`. Values are returned in the format specified by [FORM:DATA](#) (Block or ASCII).

**Parameters**

- `<ch>` Channel number to query Cal Set stimulus values. If unspecified, value is set to 1
- `[num]` Optional argument. Range of frequencies to return. These values would be different when FOM (Opt 080) is enabled.
  - `0` - returns source frequencies. Default setting if not specified.
  - `1` - returns response frequencies.
  - `2` - returns primary frequencies.

**Examples**

```plaintext
SENSe:CORR:CSET:STIM?

sense:correction:cset:stimulus 1
```

**Return Type** Numeric
SENSe:CORRection:CSET:TSET:ALLPorts? <cset>

(Read-only) Reads the port mapping used for the specified Cal Set. The returned values are the physical ports. The POSITION of the returned values corresponds to the logical ports.

For example, with an N44xx test set, if the returned string is "PNA 1,TS 2,PNA 2, TS 4" this means:

- PNA 1 is assigned to logical port 1
- TS 2 is assigned to logical port 2
- PNA 2 is assigned to logical port 3
- TS 4 is assigned to logical port 4

**Parameters**

- **<cset>** (String) Name or GUID of the Cal Set. Use SENS:CORR:CSET:CAT? to read the list of available Cal Set names or GUIDs.

**Examples**

```
SENS:CORR:CSET:TSET:ALLP? "MyCalSet"
```

**Return Type** String

**Default** Not Applicable

---

SENSe:CORRection:CSET:TSET:TYPE? <cset>

(Read-only) Reads the test set type (model) used for the specified Cal Set.

**Parameters**

- **<cset>** (String) Name or GUID of the Cal Set. Use SENS:CORR:CSET:CAT? to read the list of available Cal Set names or GUIDs.

**Examples**

```
SENS:CORR:CSET:TSET:TYPE? "MyCalSet"
returns "N44xx"
```

**Return Type** String

**Default** Not Applicable
SENSe<ch>:CORRrection:CSET:TYPE:CATalog? [format]

(Read-only) Query the Cal Types available in the selected Cal Set. The output is a comma separated list of Guids or a Cal Type names. Learn more about applying Cal Types using SCPI.

Use CALC:CORR:TYPE to apply a Cal Type.

Parameters

[ch] Any existing channel number. If unspecified, value is set to 1
[format] (Optional) Format of the output of cal types. choose from:

NAME - (default) returns a list of cal type string names.

GUID - returns a list of cal type GUIDs

Examples

SENS:CORR:CSET:TYPE:CAT? NAME
SENS2:CORR:CSET:TYPE:CAT?

Return Type String

Default Not Applicable

Last modified:

9-Mar-2011 Added last paragraph to Flatten
4-Nov-2010 Added CSET:ETerm:CAT?
19-Nov-2009 Added Flatten command
28-Sep-2009 Added <block> desc to cset:Stan command
23-Mar-2009 Added <block> desc to cset:data command
12-Nov-2008 Added Stimulus? (8.33)
6-Mar-2008 Added CSET Delete by Name (8.0)
5-Mar-2008 Added Noise ETerm and Std commands (8.0)
9/12/06 Added TSET commands for multiport.
## Sense:Correction:Extension Commands

Performs and applies Port Extensions.

<table>
<thead>
<tr>
<th>SENSE:CORREction:EXTension:</th>
</tr>
</thead>
<tbody>
<tr>
<td>AUTO</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>PORT</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
SENSe<cnum>:CORRection:EXTension:AUTO:CONFig <char>

(Read-Write) Sets the frequencies used to calculate Automatic Port Extension. Learn more about calculating Automatic Port Extension.

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Frequencies to be used:
  - **CSPN** Use current frequency span.
  - **AMKR** - Use active marker frequency.

Examples

   SENS:CORR:EXT:CONF CSPN
   sense2:correction:extension:auto:config amkr

Query Syntax

SENSe<cnum>:CORRection:EXTension:AUTO:CONFig ?

Return Type

Character

Default

CSPN
SENSe<cnum>:CORRection:EXTension:AUTO:DCOFfset <bool>

(Read-Write) Specifies whether or not to include DC Offset as part of automatic port extension. Learn more about Automatic DC Offset. Only allowed when SENS:CORR:EXT:AUTO:LOSS is set to ON.

**Parameters**

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <bool> ON (or 1) - Includes DC Offset correction.
  
  **Examples**
  
  ```
  SENS:CORR:EXT:AUTO:DCOF 1
  sense2:correction:extension:auto:dcoffset off
  ```

**Query Syntax**

SENSe<cnum>:CORRection:EXTension:AUTO:DCOFfset?

**Return Type**

Boolean

**Default**

OFF (0)

SENSe<cnum>:CORRection:EXTension:AUTO:LOSS <bool>

(Read-Write) Specifies whether or not to include loss correction as part of automatic port extension. Learn more about Loss Compensation in port extension.

**Parameters**

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <bool> ON (or 1) - Includes Loss correction.
  
  **Examples**
  
  ```
  SENS:CORR:EXT:AUTO:LOSS 1
  sense2:correction:extension:auto:loss off
  ```

**Query Syntax**

SENSe<cnum>:CORRection:EXTension:AUTO:LOSS?

**Return Type**

Boolean

**Default**

OFF (0)

SENSe<cnum>:CORRection:EXTension:AUTO:MEASure <char>
(Write-only) Measures either an OPEN or SHORT standard. When this command is sent, the PNA acquires the measurement with which to set automatic port extensions. Learn more about which standard to measure.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Standard to be measured. Choose from:
  - **OPEN**: Measure OPEN standard
  - **SHORT**: Measure SHORT standard

**Examples**

- `SENS:CORR:EXT:AUTO:MEAS OPEN`
- `sense2:correction:extension:auto:measure short`

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<cnum>:CORRection:EXTension:AUTO:PORT<n> <bool>**

(Read-Write) Enables and disables automatic port extensions on the specified port.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<n>` PNA Port number to enable or disable for automatic port extensions.
- `<bool>` ON (or 1) - Enable
  - OFF (or 0) - Disable

**Examples**

- `SENS:CORR:EXT:AUTO:PORT2 0`
- `sense2:correction:extension:auto:port4 on`

**Query Syntax**

`SENSe<cnum>:CORRection:EXTension:AUTO:PORT<n>?`

**Return Type**

Boolean

**Default**

All ports ON (enabled)

---

**SENSe<cnum>:CORRection:EXTension:AUTO:RESet**
(Write-only) Clears old port extension delay and loss data in preparation for acquiring new data. Send this command prior to sending a new series of SENS:CORR:EXT:AUTO:MEAS. If acquiring both OPEN and SHORT standards, do not send this command between those acquisitions.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SENS:CORR:EXT:AUTO:RES
sense2:correction:extension:auto:reset
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<cnm>:CORRection:EXTension:AUTO:STARt <value>**

(Read-Write) Set the start frequency for custom user span. [Learn more about User Span.](#)

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<value>` User span start value. Must be within the frequency range of the active channel and less than the value set by SENS:CORR:EXT:AUTO:STOP.

**Examples**

```
SENS:CORR:EXT:AUTO:STAR 1E9
sense2:correction:extension:auto:start 200e6
```

**Query Syntax**

```
SENSe<cnm>:CORRection:EXTension:AUTO:STARt <value>?
```

**Return Type**

Numeric

**Default**

Start frequency of the current active channel.

---

**SENSe<cnm>:CORRection:EXTension:AUTO:STOP <value>**
**SENSe<cnum>:CORRection:EXTension:PORT<pnum>:DISTance <value>**

(Read-Write) Sets and returns the port extension delay in physical length (distance).

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<pnum>`: Port Number that will receive the delay setting. If unspecified, value is set to 1.

**Examples**
- `SENSe:CORR:EXT:PORT1:DIST 12`
- `sense2:correction:extension:port2:distance .003`

**Query Syntax**
`SENSe<cnum>:CORRection:EXTension:PORT<pnum>:DISTance?`

**Return Type**
Numeric

**Default**
0

---

**SENSe<cnum>:CORRection:EXTension:PORT<pnum>:FREQuency<n> <value>**
avadoc

(Read-Write) Sets and returns the frequency for the Freq and Loss pair number and for the specified port number.

Learn about Loss Compensation values.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<pnum>`: Port Number that will receive the freq/loss settings. If unspecified, value is set to 1.
- `<n>`: Freq and Loss pair number. Choose from 1 or 2. If unspecified, value is set to 1.
- `<value>`: Frequency value. Choose a frequency within the frequency span of the PNA.

**Examples**

```
SENS:CORR:EXT:PORT1:FREQ1 10E9
sense2:correction:extension:port2:freq2 2E10
```

**Query Syntax**

```
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:FREQuency<n>?
```

**Return Type**

Numeric

**Default**

1 GHz

---

SENSe<cnum>:CORR:EXT:PORT<pnum>:INCLude<n>[;STATE] <bool>

(Read-Write) Sets and returns the ON/OFF state for the Freq and Loss pair number and for the specified port number.

Learn about Loss Compensation values.

**Note:** This command affects ALL measurements on the specified channel.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<pnum>`: Port Number that will receive the Freq/Loss settings. If unspecified, value is set to 1.
- `<n>`: Freq and Loss pair. Choose from 1 or 2. If unspecified, value is set to 1.
- `<value>`: State of Freq and Loss values for port extension.

0 or OFF Specified Freq and Loss values are OFF

1 or ON Specified Freq and Loss values are ON

**Examples**

```
SENS:CORR:EXT:PORT:INCL 0
sense2:correction:extension:port2:include2:state on
```
Query Syntax  
```
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:INCLude[:STATe]?
```

Return Type  
Boolean

Default  
OFF

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:LDC <value>
(Read-Write) Sets and returns the Port Loss at DC value for the specified port number.

Learn about Loss Compensation values.

Note: This command affects ALL measurements on the specified channel.

Parameters
- `<cnum>`  
  Any existing channel number. If unspecified, value is set to 1
- `<pnum>`  
  Port number to receive Loss value. If unspecified, value is set to 1.
- `<value>`  
  Loss in dB. Choose a value between -90 and 90

Examples
```
SENSe:CORR:EXT:PORT:LDC 1.5
SENSe:CORR:EXT:PORT:LDC 1.5
sense2:correction:extension:port2:ldc .1
```

Query Syntax  
```
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:LDC?
```

Return Type  
Numeric

Default  
0

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:LOSS<n> <value>
(Read-Write) Sets and returns the Loss value for the specified port number.

Learn about Loss Compensation values.

Note: This command affects ALL measurements on the specified channel.

Parameters
- `<cnum>`  
  Any existing channel number. If unspecified, value is set to 1
- `<pnum>`  
  Port Number that will receive the Freq/Loss settings. If unspecified, value is set to 1.
- `<n>`  
  Loss "Use" number. Choose from 1 or 2. If unspecified, value is set to 1.
- `<value>`  
  Loss in dB. Choose a value between -90 and 90
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:MEDium <char>
(Read-Write) Sets and returns the media type of the added fixture or transmission line.

See also SENS:CORR:EXT:PORT:SYSMedia

**Note:** This command affects ALL measurements on the specified channel.

**Parameters**
- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<pnum>` Port Number for which media type is being set. If unspecified, value is set to 1.
- `<char>` Medium type. Choose from:
  - COAX
  - WAVEguide

**Examples**
SENSe:CORR:EXT:PORT:MEAd COAX
SENSe:CORR:EXT:PORT:MEAd waveguide

**Query Syntax**
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:MEDium?

**Return Type** Character

**Default** COAX
(Read-Write) Sets and returns the state of coupling with the system Media type. Learn more.

Note: This command potentially affects ALL measurements on the PNA.

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<pnum>` Port Number for which system Velocity Factor coupling is being set. If unspecified, value is set to 1.
- `<bool>` Coupling state. Choose from:
  - **ON** (or 1) - Media type is coupled with the system setting.
  - **OFF** (or 0) - Media type is NOT coupled with the system setting.

Examples

```
SENS:CORR:EXT:PORT:SYSM 1
sense2:correction:extension:port2:sysmedia off
```

Query Syntax

```
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:SYSMedia?
```

Return Type

boolean

Default

1 or ON (Coupled)

---

**SENSe<cnum>:CORRection:EXTension:PORT<pnum>:SYSVelocity <bool>**

(Read-Write) Sets and returns the state of coupling with the system Velocity Factor value. Learn more.

Note: This command potentially affects ALL measurements on the PNA.

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<pnum>` Port Number for which system Velocity Factor coupling is being set. If unspecified, value is set to 1.
- `<bool>` Coupling state. Choose from:
  - **ON** (or 1) - Velocity Factor is coupled with the system setting.
  - **OFF** (or 0) - Velocity Factor is NOT coupled with the system setting.

Examples

```
SENS:CORR:EXT:PORT:SYSV 1
sense2:correction:extension:port2:sysvelocity off
```
### SENSe<cnum>:CORRection:EXTension:PORT<pnum>:SYSVelocity?

**Query Syntax**: SENSe<cnum>:CORRection:EXTension:PORT<pnum>:SYSVelocity?

**Return Type**: Boolean

**Default**: 1 or ON (Coupled)

---

### SENSe<cnum>:CORRection:EXTension:PORT<pnum>[:TIME] <num>

*(Read-Write)* Sets the extension delay value in time at the specified port. Must also set SENS:CORR:EXT ON.

**Note**: This command affects ALL measurements on the specified channel.

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<pnum>`: Port Number that will receive the extension. If unspecified, value is set to 1.
- `<num>`: The port extension in seconds; may include suffix. Choose a number between -1E18 and 1E18.

**Examples**
- SENS:CORR:EXT:PORT 2MS
- sense2:correction:extension:port2 .00025

### SENSe<cnum>:CORRection:EXTension:PORT:UNIT <char>

*(Read-Write)* Sets and returns the units for specifying port extension delay in physical length (distance).

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<char>`: Units for delay in distance. Choose from:
  - METer
  - FEET
  - INCH

**Examples**
- SENS:CORR:EXT:PORT:UNIT MET
- sense2:correction:extension:port:unit inch
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:VELFactor <value>

(Read-Write) Sets and returns the velocity factor of the fixture or added transmission line.

Parameters
- <cnum>  Any existing channel number. If unspecified, value is set to 1
- <pnum>  Port Number for which velocity factor is being set. If unspecified, value is set to 1.
- <value> Velocity Factor.

Set SENS:CORR:EXT:PORT:SYSV to use the system velocity factor.

Examples
SENSe<cnm>:CORRection:EXTension:PORT:VELF .6
sense2:correction:extension:port2:velfactor 1

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:WGCutoff <value>

(Read-Write) Sets and returns the cutoff (minimum) frequency of the added waveguide fixture or transmission line.

Parameters
- <cnum>  Any existing channel number. If unspecified, value is set to 1
- <pnum>  Port Number for which media type is being set. If unspecified, value is set to 1.
- <value> Cutoff frequency in Hz.

This value is ignored when SENS:CORR:EXT:PORT:MED is set to COAX for the same port.

Examples
SENS:CORR:EXT:PORT:WGC 1e8
sense2:correction:extension:port2:wgcutoff 100Mhz

Query Syntax
SENSe<cnum>:CORRection:EXTension:PORT<pnum>:VELFactor?

Return Type  Character
Default  METer
SENSe<cnm>:CORRection:EXTension:RECeiver<Rnum>[<TIME>] <num> OBSOLETE
(Read-Write) This command has NO replacement and no longer works.

Sets the extension value at the specified receiver. Must also set SENS:CORR:EXT ON.

**Note:** Before using this command you must select a measurement using CALC:PAR:SEL. You can select one measurement for each channel.

**Parameters**
- `<cnm>` Any existing channel number. If unspecified, value is set to 1
- `<Rnum>` Number of the receiver that will receive the extension. If unspecified, value is set to 1
  - Choose from:
    - 1 for Receiver A
    - 2 for Receiver B
- `<num>` The electrical length in seconds; may include suffix. Choose a number between:
  - -10 and 10

**Examples**
- SENS:CORR:EXT:REC 2MS
- sense2:correction:extension:receiver2:time .00025

**Query Syntax**
SENSe<cnm>:CORRection:EXTension:RECeiver<Rnum>[<TIME>]?

**Return Type** Numeric
- **Default** 0

SENSe<cnm>:CORRection:EXTension[:STATe] <ON | OFF>
(Read-Write) Turns port extensions ON or OFF.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>` **ON** (or 1) - turns port extensions ON.
  - **OFF** (or 0) - turns port extensions is OFF.

**Examples**

```
SENS:CORR:EXT ON
sense2:correction:extension:state off
```

**Query Syntax**

```
SENSe<cnum>:CORRection:EXTension[:STATe]?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default** OFF

---

Last Modified:

- 11-Jan-2011 Minor edits
- 28-Sep-2009 Fixed Port:Frequency command syntax
- 6-Feb-2009 Added new commands (A.08.50)
**Sense:Correction:Collect:Guided Commands**

Performs and applies a SmartCal (Guided) calibration and other error correction features.

To perform a **Guided Calibration**, use ONLY `Sens:Corr:Coll:Guided` commands. See the "Guided" example programs for clarification.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSE:CORRectiON:COLLect:GUIDed:</td>
<td></td>
</tr>
<tr>
<td>ABORt</td>
<td>ABORT</td>
</tr>
<tr>
<td>ACQuire</td>
<td>ACQUIRE</td>
</tr>
<tr>
<td>ADAPter</td>
<td></td>
</tr>
<tr>
<td>COUNt</td>
<td></td>
</tr>
<tr>
<td>ZERO</td>
<td></td>
</tr>
<tr>
<td>CReate?</td>
<td></td>
</tr>
<tr>
<td>DELay</td>
<td></td>
</tr>
<tr>
<td>DESCription</td>
<td></td>
</tr>
<tr>
<td>PATHs</td>
<td></td>
</tr>
<tr>
<td>CHANnel:MODE</td>
<td></td>
</tr>
<tr>
<td>CKIT</td>
<td></td>
</tr>
<tr>
<td>CATalog?</td>
<td></td>
</tr>
<tr>
<td>PORT</td>
<td></td>
</tr>
<tr>
<td>CATalog?</td>
<td></td>
</tr>
<tr>
<td>[SELect]</td>
<td></td>
</tr>
<tr>
<td>CONNector</td>
<td></td>
</tr>
<tr>
<td>CATalog?</td>
<td></td>
</tr>
<tr>
<td>PORT</td>
<td></td>
</tr>
<tr>
<td>[SELect]</td>
<td></td>
</tr>
<tr>
<td>DESCription</td>
<td></td>
</tr>
<tr>
<td>DMATch</td>
<td></td>
</tr>
<tr>
<td>APPLy</td>
<td></td>
</tr>
<tr>
<td>[IMMediate]</td>
<td></td>
</tr>
<tr>
<td>PORTs?</td>
<td></td>
</tr>
<tr>
<td>[INITiate]</td>
<td></td>
</tr>
<tr>
<td>ETERms:LOAD</td>
<td></td>
</tr>
<tr>
<td>[CSET]</td>
<td></td>
</tr>
<tr>
<td>INITiate</td>
<td></td>
</tr>
<tr>
<td>ISOLation</td>
<td></td>
</tr>
<tr>
<td>AVERage</td>
<td></td>
</tr>
<tr>
<td>INCRement</td>
<td></td>
</tr>
</tbody>
</table>
SENSe<ch>:CORRection:COLLect:GUIDed:ABORt

(Write-only) Aborts the acquiring of a guided calibration that has been INITialized but has not yet been concluded using the SAVE command. If at least one Cal standard has already been measured, and the Calibration Window is being displayed, this command also closes the Calibration Window and re-tiles the other measurement windows.

Parameters

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

Examples

SENs:CORr:COLL:GUID:ABOr
sense2:correction:collect:guided:abort

Query Syntax

Not Applicable

Default

Not Applicable
**SENSe:<ch>CORRection:COLLect:GUIDed[:ACQuire] <std>[,sync]**

*(Write-only)* Initiates the measurement of the specified calibration standard. Executing this command with an unnecessary standard has no affect.

The measured data is stored and used for subsequent calculations of error correction coefficients. All standards must be measured before a calibration can be completed. Any measurement can be repeated until the **SENS:CORR:COLL:GUID:SAVE** command is executed.

Query the user prompt description using **SENS:CORR:COLL:GUID:DESC**?

Query the required calibration steps using **SENS:CORR:COLL:GUID:STEP**?

**Parameters**

- `<ch>` Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- `<std>` Choose from: STAN1, STAN2, STAN3, through STAN40
- `[sync]` Optional argument. Choose from:
  - **SYNChronous** - blocks SCPI commands during standard measurement (default behavior).
  - **ASYNchrous** - does NOT block SCPI commands during standard measurement.

**Learn more.**

**Examples**

- **SENS:CORR:COLL:GUID STAN1**
- **sense2:correction:collect:guided:acquire stan1**

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:ADAPter:CREate? <conn1>, <conn2>**
Specifies the use of a THRU adapter to be used during the Guided Cal Unknown THRU and Adapter Removal Cal. Returns an adapter index <n> which is used to refer to the adapter in several related commands. See Cal Thru Methods. While the choice of which end of the adapter is <conn1> and <conn2> is arbitrary, it is necessary to remember which will be used on each test port.

The settings for this command remain until Preset, or the command is sent using a different setting, or until the ZERO command is sent.

**Parameters**

- `<ch>` Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.
- `<conn1>` Adapter port 1 connector type. Use SENS:CORR:COLL:GUID:CONN:CAT? to return a list of valid connector types.
- `<conn2>` Adapter port 2 connector type.

**Examples** See example using this command.

**Return Type** Numeric

**Default** Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:ADAPter:COUNt?

(Read-Only) Returns the number of THRU adapters that have been created for this calibration using SENS:CORR:COLL:GUID:ADAP:CREate.

**Parameters**

- `<ch>` Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

**Examples** See example using this command.

**Return Type** Numeric

**Default** Not Applicable

SENSe<ch> CORRection:COLLect:GUIDed:ADAPter:COUNT:ZERO

2532
(Write-only) Removes all adapters that have been defined for calibrations on the specified channel using **SENS:CORR:COLL:GUID:ADAP:CREate**.

**Parameters**

- `<ch>` Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.

**Examples**

```
SENS:CORR:COLL:GUID:ADAP:COUNT:ZERO
```

```
SENSe:CORRection:COLLect:GUIDed:ADAPter<n>:DElay <coax>, [w phase, wdelay]
```

(Write-only) Specifies the adapter delay, and optionally waveguide delay and optional phase offset (degrees) of adapter `<n>`.

The settings for this command remain until Preset, or the command is sent using a different setting, or until the **ZERO** command is sent.

**Parameters**

- `<ch>` Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- `<n>` Adapter index number that was returned from **SENS:CORR:COLL:GUID:ADAP:CREate?**
- `<coax>` Delay value of coax adapter `<n>` in seconds. If the adapter has no coax connector, enter 0.
- `<wphase>` Waveguide phase offset in degrees. If the adapter has no waveguide connector, do not enter a value.
- `<wdelay>` Waveguide delay in seconds. If the adapter has no waveguide connector, do not enter a value.

**Examples**

See example using this command.

**Default** Not Applicable

**SENS**<ch>:CORRection:COLLect:GUIDed:ADAPter<n>:DEscription <string>
(Write-only) Specifies the adapter description for use as the guided cal connection prompts.

The settings for this command remain until Preset, or the command is sent using a different setting, or until the **ZERO** command is sent.

**Parameters**

- `<ch>` Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- `<n>` Adapter index number that was returned from **SENS:CORR:COLL:GUID:ADAP:CREate?**
- `<string>` Adapter description.

**Examples**

See example using this command.

**Query Syntax**

Not Applicable

**Default**

Not Applicable

**SENSe<ch>:CORRection:COLLect:GUIDed:ADAPter<n>:PATHs <port pairs>**

(Write-only) Specifies the port pairs for which the adapter will be used for a THRU connection.

For example, for a 3-port cal on channel 1 using ports 1,2, and 3), to use adapter 1 between the ports (1 to 2) and (1 to 3) the following command is used: SENS1:CORR:COLL:GUID:ADAP1:PATH 1,2,1,3.

The adapter must have the same DUT connectors as the ports that are already specified for these ports.

The settings for this command remain until Preset, or the command is sent using a different setting, or until the **ZERO** command is sent.

**Parameters**

- `<ch>` Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- `<n>` Adapter index number that was returned from **SENS:CORR:COLL:GUID:ADAP:CREate?**
- `<port pair>` Ports for which the adapter will be used. The orientation is not critical, as the PNA will align the connector types as necessary. The minimum number of Thru connections required is the number of ports to calibrated -1.

**Examples**

See example using this command.

**Query Syntax**

Not Applicable
**SENSe:CORRection:COLLect:GUIDed:CHANnel:MODE <bool>**

(Read-Write) Determines whether or not to honor the channel `<ch>` argument in guided calibration SCPI commands.

**Parameters**

- **<bool> OFF (0)**: Honor all `<ch>` arguments. This means the `<ch>` channel is calibrated regardless of which channel is currently active.

- **<bool> ON (1)**: Default (legacy) behavior. Behavior is specified by the following table:

<table>
<thead>
<tr>
<th><code>&lt;ch&gt;</code> channel type</th>
<th>Active channel type</th>
<th>Behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td>Std or App</td>
<td>Std or App</td>
<td></td>
</tr>
<tr>
<td>Std</td>
<td>Std</td>
<td>Active chan cal'd</td>
</tr>
<tr>
<td>Std</td>
<td>App</td>
<td>&quot;Channel not found&quot; error</td>
</tr>
<tr>
<td>App</td>
<td>Std</td>
<td><code>&lt;ch&gt;</code> chan cal'd</td>
</tr>
<tr>
<td>App</td>
<td>App</td>
<td><code>&lt;ch&gt;</code> chan cal'd</td>
</tr>
</tbody>
</table>

Learn about [Standard vs Application](#) channels.

**Examples**

SENSe:CORR:COLL:GUID:CHAN:MODE 0

sense:correction:collect:guided:channel:mode ON

**Query Syntax**

SENSe:CORRection:COLLect:GUIDed:CHANnel:MODE?

**Return Type**

Boolean

**Default**

ON (1)

Returns a comma-separated list of valid kits that use the specified connector type. This includes mechanical cal kits, applicable characterizations found within ECAL modules currently connected to the PNA, and all user characterizations stored in PNA disk memory. For ECAL modules, the returned list includes the serial numbers. See ECAL User Characterization commands.

Use items in the list to select the kit to be used with the `SENS:CORR:COLL:GUID:CKIT:PORT` and `SENSe<ch>:CORRection:COLLect:GUIDed:PSENsor<pnum>:CKIT` commands.

### Parameters

- **<conn>**
  

- **Examples**
  
  `SENSe:CORR:COLL:GUID:CKIT:CAT? "Type N (50) male"`

- **Return Type**
  
  String

- **Default**
  
  Not Applicable

---


(Read-only) This command is replaced by `SENS:CORR:COLL:GUID:CKIT:CAT?`.

Returns a comma-separated list of valid kits for the specified PNA port. In addition to mechanical calibration kits, this will include applicable characterizations found within ECAL modules currently connected to the PNA.

Use items in the list to select the kit to be used with the `SENS:CORR:COLL:GUID:CKIT:PORT` command.

**Note:** Beginning with PNA Rev 9.1, the serial number is returned for ALL ECAL modules that are connected with the connector type of the specified port. Previously, the returned list would include the serial numbers to distinguish the ECAL modules only when two or more identical ECAL models were connected to the PNA.

### Parameters

- **<pnum>**
  
  Any existing port number. If unspecified, value is set to 1

- **Examples**
  

  'When "Type N (50) male" is specified for connector type, returns:

  "85054D, 85032F"

  'When two identical ECAL modules are connected for the connector type,'

(Read-Write) Specifies the calibration kit (mechanical or ECal) for each port to be used during a guided calibration. An unused port does NOT need to have a specified Cal Kit.

**Note**: Sliding loads are not fully supported from Sens:Corr:Coll:Guided... The **Measure** button must be manually pressed.

1. Specify the connector type for the port with SENS:CORR:COLL:GUID:CONN:PORT.
2. Query the valid available kits for the connector on each port with SENS:CORR:COLL:GUID:CKIT:PORT:CAT?
3. Specify the kit using this command.
4. Perform a query of this command. If the <kit> parameter was incorrectly entered, an error will be returned.

**Parameters**

- **<ch>** Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- **<pnum>** Any existing port number. If unspecified, value is set to 1
- **<kit>** Calibration kit to be used for the specified port. **Case-sensitive.**

When using an ECal module, include the characterization name in the <kit> string. Use SENSE:CORR:COLL:GUID:CKIT:CAT? to read the list of characterizations available in the module and in PNA disk memory.

If two or more identical ECal modules are connected to the PNA, the serial number must be included to distinguish the ECal modules.

**Examples**

'Note: All of the following examples specify port 1 only

' Mechanical Cal kit

SENS:CORR:COLL:GUID:CKIT:PORT1 '85055A'

' Standard ECal modules

---

Return Type: String
Default: Not Applicable
Non-factory ECal characterizations are specified as follows:

SENS:CORR:COLL:GUID:CKIT:PORT1 "N4691-60004 User 1 ECal"

When two or more ECal modules with the same model number are connected, also specify the serial number as follows:

SENS:CORR:COLL:GUID:CKIT:PORT1 "N4691-60004 ECal 01234"

When Disk Memory ECal user characterizations are used, specify both the User char and the serial number as follows:

SENS:CORR:COLL:GUID:CKIT:PORT1 "N4691-60004 MyDskChar ECal 01234"


Return Type: String - If the <kit> parameter was incorrectly entered while writing, an error will be returned.

Default: Not Applicable

SENSe:CORRection:COLlECT:GUIDed:CONNector:CATalog?

(Read only) Returns a list of valid connectors based on the connector descriptions of the available cal kits. Use an item from the returned list to specify a connector for SENSE:CORR:COL:GUID:CONN:PORT

Here are the more common connector types:

- W-band waveguide
- V-band waveguide
- U-band waveguide
- R-band waveguide
- Q-band waveguide
- K-band waveguide
- P-band waveguide
- X-band waveguide
- 7-16 female

Type B: 1.00 mm female

Type A (50): 1.00 mm male

Type A (50): 1.85 mm male

Type F (75): 1.85 mm female

Type F (75): 2.92 mm female

Type N (75): 2.92 mm female

Type N (75): APC 2.4 female

Type N (50): APC 2.4 male

Type N (50): APC 3.5 female
Examples

```
SENS:CORR:COLL:GUID:CONN:CAT?
```

**Returns:**

Type N (50) female, Type N (50) male, APC 7 (50), 3.5 mm (50) male, 3.5 mm (50) female, User Connector A

**Return Type** Comma separated string values

**Default** Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:CONNECTor:PORT<pnum>[:SElect] <conn>

(Read-Write) Specifies a connector type for every port during the Guided Calibration procedure. Valid connector names are stored within calibration kits. Some cal kits may include both male and female connectors. Therefore, specifying connector gender may be required.

The PNA remembers previous Guided Cal settings. Therefore, for completeness, unused ports should be defined as "Not used". See Guided Cal examples.

- A single port with a valid <conn> name indicates a 1-Port calibration will be performed.
- Two ports with valid <conn> names indicate either a 2-Port SOLT or TRL calibration will be performed depending on the standards definition found within the cal kit and the capability of the PNA.
- Three ports with valid <conn> names indicate a 3-Port calibration will be performed, and so forth.

Follow these steps to ensure port connectors are specified correctly:

2. Set a connector type for each port using this command.
3. Perform a query of this command. If the connector type was incorrectly entered, an error will be returned.
4. Specify the cal kit to use for each port with SENS:CORR:COLL:GUID:CKIT:PORT.

**Parameters**

- `<ch>` Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.
<pnum> Any existing port number. If unspecified, value is set to 1.
<conn> String - DUT connector type to connect with PNA port <pnum>. Case-sensitive.

**Examples**

'Specifying a 2-port cal (1 & 2) on a 4-port PNA

SENS:CORR:COLL:GUID:CONN:PORT1 'Type N (50) female'
SENS:CORR:COLL:GUID:CONN:PORT2 'Type N (50) male'
SENS:CORR:COLL:GUID:CONN:PORT3 'Not used'
SENS:CORR:COLL:GUID:CONN:PORT4 'Not used'

**Query Syntax**

SENSe:CORRection:COLLect:GUIDed:CONNector:PORT<pnum>[[:SELect]?]

**Return Type**

String

**Default**

Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:DESCription? <step>**

(Read-only) Returns the connection description for the specified calibration step.

**Parameters**

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

<step> A number from 1 to the number of steps required to complete the calibration (Use SENS:CORR:COLL:GUID:STEP? to query the number of steps)

**Examples**

SENS:CORR:COLL:GUID:DESC? 10

'Returns:
Connect APC 7 Open to port3

**Return Type**

String

**Default**

Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:DMATch:APPLY[:IMMediate] [<CalSetGUID>]**
(Write-only) Specifies a Cal Set as a source of delta match correction.

If CalSetGUID is not specified, then the Global Delta Match Cal Set is assumed. An error is returned if the specified Cal Set does not meet the following Delta Match criteria. The Global Delta Match Cal can ALWAYS be applied.

- Must have been performed using ECal or as a guided mechanical cal (not Unguided).
- Must have the same start freq, stop freq, and number of points as the channel being calibrated.
- Must calibrate the ports that are required by the TRL or Unknown Thru cal as indicated by SENS:CORR:COLL:GUID:DMATch:APPLy:PORTs?.

Learn more about Delta match calibration.

See example of a complete Global Delta Match calibration.

See example where Delta Match is applied to a calibration.

**Parameters**

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

<CalSetGUID> Optional. GUID of the Cal Set to use. If unspecified, the Global Delta Match Cal Set is used.

**Examples**

SENS:CORR:COLL:GUID:DMAT:APPL
sense:correction:collect:guided:dmatch:apply:immediate
"{2B893E7A-971A-11d5-8D6C-00108334AE96}"

**Query Syntax** Not Applicable

**Default** Not Applicable

`SENSe<ch>:CORRection:COLLect:GUIDed:DMATch:APPLy:PORTs?`
(Read-only) Returns the port numbers for which delta match correction is required. 0 (zero) is returned if the Cal does NOT require Delta Match correction for one of the following reasons:

- The Cal does NOT involve Unknown THRU or TRL. You specify this using `SENS:CORR:COLL:GUID:METH <UNKN | TRL>`.
- The Cal DOES involve Unknown THRU or TRL, but the delta match data can be calculated by the Unknown Thru or TRL Cal. Learn how this is possible. However, you can force the Cal to use the Delta Match data from a Cal Set.

Learn more about Delta match calibration.

See example of a complete Delta Match calibration.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>&lt;ch&gt;</code></td>
<td>Channel being calibrated, depending on the <code>CHAN:MODE</code> setting. If unspecified, value is set to 1.</td>
</tr>
</tbody>
</table>

**Examples**

```
SENS:CORR:COLL:GUID:APPL:PORT?
'Returns:
1, 2, 3
```

**Return Type** Numeric

**Default** Not Applicable

**SENS<ch>:CORRection:COLLect:GUIDed:DMATch[:INITiate] <conn>,<cKit>**

(Write-only) Initiates a global delta match calibration.

Learn more about Global Delta Match calibration.

See example of a complete Delta Match calibration.

**Parameters**

- `<ch>` Channel being calibrated, depending on the `CHAN:MODE` setting. If unspecified, value is set to 1.
- `<conn>` **String**. Connector type for port 1. All other ports are set automatically.
- `<cKit>` **String** Cal Kit for all ports. If incorrectly entered while writing, an error is returned.

**Examples**

```
SENS:CORR:COLL:GUID:DMAT APC 3.5 female,"85052B"
```

**Query Syntax** Not Applicable

**Default** Not Applicable

(Write-only) Loads 1-port error terms from a Cal Set into the current Guided Cal sequence. When the Cal steps are recomputed, connection steps are removed due to the loading of the error terms.

See example of how to use this command.

Parameters

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

<cset> String Name of User Cal Set in which the error terms reside.

<pnum> Integer Port number of the current cal to receive error terms.

[csPort] Integer Optional argument. Port number associated with the error terms in the Cal Set. If unspecified, the same port number as <calPort> is used.

Examples See example

Query Syntax Not Applicable

Default Not Applicable

---

SENSe<ch>:CORRection:COLLect:GUIDed:INITiate[:IMMediate] [string[, bool]]

(Write-only) Initiates a guided calibration.


- After this command is executed, subsequent commands can be used to query the number of measurement steps, issue the acquisition commands, query the connection description strings, and subsequently complete a guided calibration.

Parameters

<ch> Channel to be calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

[string] Optional argument. Cal set name or GUID enclosed in quotes.

If specified, choose an existing Cal Set, either by name or by GUID.

If NOT specified, behavior depends on the SENS:CORR:PREFerence:CSET:SAVE setting.
By Cal Set name: include quotes.

By Calset GUID in the form: " {GUID} " ; including quotes and curly brackets.

Query all Cal Set GUIDs with SENS:CORR:CSET:CAT?

An error is reported if the Cal Set is not found.

The Cal Set is either supplemented or overwritten depending on the method, connectors, and ports selected. Learn more about Cal Sets.

[bool] Optional argument.

**OFF (0)** If Cal Set stimulus settings differ from the existing channel, do not change channel stimulus settings. The Cal Set is save to the current setting of the SENS:CORR:PREF:CSET:SAVE command. This is the default setting if not specified.

**ON (1)** If cal set stimulus settings differ from the existing channel, change the channel stimulus settings to match the Cal Set settings.

**Examples**


sense2:correction:collect:guided:initiate:immediate 'myCalSet'

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:ISOLation:AVERage:INCRement <num>**

*(Read-Write)* Specifies amount to increment (increase) the channels averaging factor during measurement of isolation standards in a guided calibration.

**Note:** If the channel has averaging turned OFF and the value of <num> is greater than 1, averaging will be turned ON only during the isolation measurements and with the averaging factor equal to <num>.

**Parameters**

*<ch>* Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

*<num>* Amount to increment the averaging factor for the isolation measurement. The maximum averaging factor for the channel is 65536 ($2^{16}$).

**Examples**

'Measure isolation on all paths for the cal

SENS:CORR:COLL:GUID:ISOL ALL
Remove the port pairs 1-to-2 and 1-to-3 from the list of paths on which to measure isolation.

Query Syntax

Return Type
Numeric

Default
8 - If this command is NOT sent, but isolation is measured, then averaging will be turned ON with factor set to 8 during the isolation measurements.

SENSe<ch>:CORRection:COLLect:GUIDed:ISOLation[:PATHs] <char>[,<p1a, p1b, p2a, p2b]

(Read-Write) Specifies the paths (port pairs) to make isolation measurements on during a guided calibration.

Parameters

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

<char> ALL Measure isolation on all pairings of the ports that are to be calibrated.

NONE Do not measure isolation on any pairing of the ports to be calibrated. (Default behavior).

ADD Add one or more specific pairings of ports to the list of port pairings for which isolation will be measured.

REMove Remove one or more specific pairings of ports from the list of port pairings for which isolation will be measured. If many paths are to be measured, it may be easier to first send ALL, then REMove and specify the paths to remove.

<p1a, p2a...> For use when <char> is ADD or REMove.

Specify Port numbers in pairs:

- For 3-port cals, specify up to 3 pairs.
- For 4-port cals, specify up to 6 pairs.

p1a, p1b (Path1 - port A and port B)

p2a, p2b (Path2 - port A and port B)

p3a, p3b (Path3 - port A and port B)
Examples

'Measure isolation on all paths for the cal
SENS:CORR:COLL:GUID:ISOL ALL

'Remove the port pairs 1-to-2 and 1-to-3 from the list of paths on which to measure isolation
sense:correction:collect:guided:isolation:paths REMove,1,2,1,3

Query Syntax
SENS<ch>:CORRection:COLLect:GUIDed:ISOLation:PATHs?

Note: if isolation is not be measured on any of the paths, the query returns 0

Return Type
Numeric

Default
0 - Isolation not measured on any paths.

SENSe:CORRection:COLLect:GUIDed:METHod <char> Superseded


(Read-Write) Selects from one of several algorithms available for performing the THRU portion of a guided calibration. Learn more about THRU methods.

Parameters

<char> DEFAULT - Informs guided calibrations to use the default algorithm when computing the number of needed standards acquisition steps. (default selection if omitted.)

ADAP - Use the adapter removal algorithm

FLUSH - Use with insertable devices.

UNKN - Use the Unknown THRU algorithm with calibrations for non-insertable devices.

DEFINED - Use the THRU definition that you stored in the cal kit file, or ECal module.

TRL - Select TRL Cal Type for guided cals. Valid for "TRL ready" Cal Kits with properly assigned TRL cal classes.

SOLT - Select SOLT Cal Type for guided cals. Valid for any kit with properly assigned SOLT cal classes.

Examples

SENS:CORR:COLL:GUID:METH ADAP
sense:correction:collect:guided:method unkn
SENSe<ch>:CORRection:COLLect:GUIDed:PACQuire <std>

(Read-Write) Show the Cal Window, and optionally one or more other specific windows before acquiring a Cal standard. This command will cause the Cal Window to display the specific measurements that are to be made for that particular Cal standard.

Parameters

- `<ch>`: Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.
- `<std>`: Choose from: STAN1, STAN2, STAN3, through STAN40.

Examples

```
SENS:CORR:COLL:GUID:PACEquire STAN2
sense:correction:collect:guided:pacquire STAN5
```

See an example that uses this command.

Query Syntax

SENSe<ch>:CORRection:COLLect:GUIDed:PACQuire?

Return Type: Character

Default: Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:PATH:CMEThod <port1>,<port2>,<caltype1[,caltype2]>

Note: This command replaces SENS:CORR:COLL:GUID:METH.

(Read-Write) Specifies the calibration method for each port pair.

Note: Do NOT send this command to rely on SmartCal to determine the most accurate cal method for your connector settings and Cal Kits. You can send the query form of this command to learn the cal method determined by SmartCal.

Before sending this command, first do the following:

1. Set the connector types: SENS:CORR:COLL:GUID:CONN:PORTn
3. Set or query the thru path pairs: SENS:CORR:COLL:GUID:THRU:PORT

5. **Send this command** `(SENS:CORR:COLL:GUID:PATH:CMET)`

After sending this command, send the query form to be sure that the command was accepted. If not, then the chosen Cal method is not compatible with the specified Thru method. For example, if the specified Thru method is Unknown Thru, an attempt to set Enhanced Response Cal should be rejected.

Learn more about Thru Methods.

**Parameters**

- `<ch>` Channel being calibrated, depending on the `CHAN:MODE` setting. If unspecified, value is set to 1.
- `<port1>` First port of the pair to be calibrated.
- `<port2>` Second port of the pair to be calibrated.
- `<caltype1[caltype2]>` (String) Cal types for 1st and 2nd ports of the pair, enclosed in a single pair of quotes. NOT case-sensitive.

  - **caltype1** Cal type for the pair if caltype2 is not specified. Otherwise, Cal type for port 1. Choose from:
    - TRL
    - SOLT
    - QSOLTN
    - EnhRespN

  For the last two arguments, replace N with the port to be used as the source port, which MUST be one of the port pair.

  - **caltype2** Optional argument. Use only when performing an adapter removal cal on the pair. This argument specifies the Cal type on the second port. Caltype1 then specifies the Cal type of the first port.

Examples

```
SENS:CORR:COLL:GUID:PATH:CMEThod 2,3,"QSOLTN"
```

```
sense:correction:collect:guided:path:cmethod 2,3,"solt,trl"
```
Query Syntax

SENSe<ch>:CORRection:COLLect:GUIDed:PATH:CMETHod?
<port1>,<port2>

If only one caltype is returned then its NOT adapter removal.

Return Type
String

Default
The most accurate Cal method for the current cal.

SENSe<ch>:CORRection:COLLect:GUIDed:PATH:TMETHod
<port1>,<port2>,<thruType1[,thruType2]>

Note: This command replaces SENS:CORR:COLL:GUID:METH.

(Read-Write) Specifies the calibration THRU method for each port pair.

Note: Do NOT send this command to rely on SmartCal to determine the most accurate cal for your connector settings and Cal Kits. You can send the query form of this command to learn the THRU method determined by SmartCal.

Before sending this command, first do the following:

1. Set the connector types: SENS:CORR:COLL:GUID:CONN:PORTn
3. Set or query the thru path pairs: SENS:CORR:COLL:GUID:THRU:PORT
4. Send this command (SENS:CORR:COLL:GUID:PATH:TMETHod)

To determine the default Thru method:

1. Set the connector types: SENS:CORR:COLL:GUID:CONN:PORTn
4. Read the proposed thru path pairs: SENS:CORR:COLL:GUID:THRU:PORT?
5. Read this command:(SENS:CORR:COLL:GUID:PATH:TMETHod)

After sending this command, set and query the Cal Type: SENS:CORR:COLL:GUID:PATH:CMETHod

Learn more about Thru methods.
Parameters

<ch> Channel being calibrated, depending on the CHAN:MODE setting. If unspecified, value is set to 1.

<port1> First port of the port pair to be calibrated.

<port2> Second port of the port pair to be calibrated.

<thruType1,[thruType2]> (String) Thru methods for 1st and 2nd ports of the pair, enclosed in a single pair of quotes. NOT case-sensitive.

thruType1 Calibration thru method for the pair if thruType2 is not specified. Otherwise, thru method for port 1.

Choose from:

- **Defined Thru** A thru type for which there is a stored definition in the Cal Kit.
- **Zero Thru** Zero length thru, also known as flush-thru.
- **Undefined Thru** A thru type for which there is NOT a stored definition in the Cal Kit. Also known as Unknown Thru. Valid ONLY for SOLT cal type.
- **Undefined Thru using a Defined Thru** Using an ECal module, measure the internal thru using the "Undefined Thru" method.

thruType2 Optional argument. Use only when performing an adapter removal cal on the pair as determined by SENS:CORR:COLL:GUID:PATH:CMETHod.

The only valid arguments for ThruType1&2 (when specifying ThruType2) "Defined Thru, Defined Thru".

Examples

SENSe<ch>:CORRection:COLLect:GUIDed:PATH:TMEThod?

Query Syntax

SENSe<ch>:CORRection:COLLect:GUIDed:PATH:TMEThod?

Always returns two parts:

If the second part of the string is empty, adapter removal is NOT being performed.

If the string is "Defined Thru, Defined Thru", adapter removal IS being performed.
Return Type  String

Default  The most accurate Thru method for the current cal.

SENSe<ch>:CORRection:COLLect:GUIDed:SAVE[:IMMediate] [bool]

(Write-only) Completes the guided cal by computing the error correction terms, turning Correction ON, and saving the calibration to a Cal Set.

If all of the required standards have not been measured, the calibration will not complete properly.

**Parameters**

- `<ch>` Channel being calibrated, depending on the `CHAN:MODE` setting. If unspecified, value is set to 1.
- `[bool]` Optional argument. If unspecified, the default behavior is the current PNA preference setting of `SENSe:CORRection:PREFerence:CSET:SAVE

  **OFF** (0)  Save cal data ONLY to a Cal Register.

  **ON** (1)  Save cal data to a Cal Register and a User Cal Set. The filename is automatically generated.

**Examples**

```
SENS:CORR:COLL:GUID:SAVE
sense2:correction:collect:guided:save:immediate 0
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:SAVE:CSET <cal set name or guid>
(Write-only) Completes the guided cal by computing the error correction terms, turning Correction ON, and saving the calibration to the specified Cal Set. This command performs the same function as `SENSe:CORRection:COLLect:GUIDed:SAVE`, but this command allows the name or GUID of the Cal Set to be specified.

- Use this command instead of the optional name or GUID argument in `SENSe:CORR:COLL:GUID:INIT`.
- Use `SENSe:CORR:CSET` commands to get names or GUIDs of existing Cal Sets.
- The cal data is also saved to the channel Cal Register.
- If all of the required standards have not been measured, the calibration will not complete properly.

**Parameters**

- `<ch>` Channel being calibrated, depending on the `CHAN:MODE` setting. If unspecified, value is set to 1.
- `<cal set name or guid>` String - Name or GUID of an existing Cal Set to be overwritten. If specifying a GUID, curly brackets must be included.

**Examples**

```
```

```
sense:correction:collect:guided:save:cset "MyCalSet"
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:STEPs?**

(Read-only) Returns the number of measurement steps required to complete the current guided calibration. This command is sent after the `SENSe:CORR:COLL:GUID:INIT`, `SENSe:CORR:COLL:GUID:CONN:PORT` and `SENSe:CORR:COLL:GUID:CKIT:PORT` commands.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.

**Examples**

```
SENS:CORR:COLL:GUID:STEP?
sense2:correction:collect:guided:steps?
```

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<ch>:CORRection:COLLect:GUIDed:THRU:PORTs <t1a, t1b, t2a, t2b, t3a, t3b...>**
(Read-Write) For calibrating more than 2-ports ONLY. Specifies the port pairs for the thru connections of the calibration.

**Note:** Do NOT send this command to rely on SmartCal to determine the most accurate cal for your connector settings and Cal Kits. You can send the query form of this command to learn the Thru pairs determined by SmartCal.

**Parameters**

- **<ch>** Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- **<t1a,...>** Specify Port numbers in pairs:
  - For 3-port cals, specify two or three pairs.
  - For 4-port cals, specify from three up to six pairs.

  - t1a, t1b (Thru1 - port A and port B)
  - t2a, t2b (Thru2 - port A and port B)
  - t3a, t3b (Thru3 - port A and port B)

  ...and so forth up to six pairs.

**Examples**

```plaintext
SENS:CORR:COLL:GUID:THRU:PORT 1,2,1,3,1,4 '4-port measurement
SENSe:CORR:COL:GUID:THRU:PORTs 1,2,2,3 '3-port measurement
```

**Query Syntax**

`SENSe<ch>:CORRection:COLLect:GUIDed:THRU:PORTs?`

**Return Type**

Numeric

**Default**

Port pairings that were used in the previous cal.

---

Last modified:
5-Apr-2011  Modified ECal Cal Kit examples
11-Jan-2011  Several edits
29-Nov-2010  Moved Psensor commands to their own page (9.33)
22-Oct-2010  Linked Save
30-Jun-2010  Added chan:mode and Guided power cal (9.30)
16-Nov-2009  Added Note to GUID:CKIT:PORT<num>:CAT?
24-Aug-2009  Added ckitCat (9.0)
17-Jul-2009  Fixed order in which CMET and TMET are set.
27-Mar-2009  Fixed Isolation:Incr query
24-Feb-2009  Replaced True/False
13-Feb-2009  Added sync to ACQuire
26-Sep-2008  Added link to apply Delta Match example
10-Mar-2008  Added Abort command (A.07.50.27)
1-Nov-2007  Added PAcquire command
14-Apr-2007  Added Cal Set by name and isolation commands
 8-Mar-2007  Added CMethod and TMethod
23-Oct-2006  Fixed wording in Conn:Port:Sel
Sense:Correction:IMD Commands

Controls an IMD and IMDx calibration. These commands supplement the Guided Cal commands.

SENSe:CORRection:IMD

| CALibration
| FREQuencies
| METHod
| LO
| PCAL
| MPRoduct
| POWer
| SENSor
| CKIT
| CONNector
| SORDer:INCLude

Click on a blue keyword to view the command details.

Other IMD (Opt 087) commands

- CALC:CUSTom:DEFine - creates an IMD measurement.
- Swept IMD
- IMSSpectrum

See Also

- Example - Create and Cal an IMD measurement
- Learn about IMD Calibration
- Learn about Measurement Class
- Synchronizing the PNA and Controller
- SCPI Command Tree
SENSe<cnm>:CORRection:IMD:CALibration:FREQuencies <char>
(Read-Write) Sets and returns the frequencies at which an IMD source power cal is performed.

Parameters

<cnm> Any existing channel number; if unspecified, value is set to 1.
<char> Choose from:

- CENTer - Perform source power calibration at only the center frequencies, midway between the main tones.
- ALL - Perform source power calibration at all main tone frequencies.

Examples

SENS:CORR:IMD:CAL:FREQ ALL
sense2:correction:imd:calibration:frequencies center

Query Syntax
SENSe<cnm>:CORRection:IMD:CALibration:FREQuencies?

Return Type
Character

Default
CENTer

SENSe<cnm>:CORRection:IMD:CALibration:METHod <char>
(Read-Write) Sets and returns the method by which the match-correction portion of an IMD calibration is performed.

Parameters

<cnm> Any existing channel number; if unspecified, value is set to 1.
<char> Choose from:

- MATCH - Performs a full 2-port cal for full match-correction.
- RESPONSE - Performs only a response (normalization) cal instead of a full 2-port cal.

Examples

SENS:CORR:IMD:CAL:METH MATCH
sense2:correction:imd:calibration:method response

Query Syntax
SENSe<cnm>:CORRection:IMD:CALibration:METHod?

Return Type
Character

Default
MATCH

SENSe<cnm>:CORRection:IMD:LO<n>:PCAL[:STATe] <bool>
(Read-Write) Sets and returns whether or not the LO power cal step is included in the cal steps when an IMDX cal is performed. Learn more.

**Parameters**

- **<cnum>** Any existing channel number; if unspecified, value is set to 1.
- **<n>** LO Stage. Choose 1.
- **<char>** LO Power Cal state. Choose from:
  - O or OFF - Skips over the LO Power Cal when calibrating.
  - 1 or ON - Includes a step for LO Power Cal when calibrating

**Examples**

```
SENS:CORR:IMD:LO1:PCAL 0
sense2:correction:imd:lo1:pcal:state on
```

**Query Syntax**

SENSe<cnum>:CORRection:IMD:LO<n>:PCAL[:STATe]?

**Return Type**

Boolean

**Default**

PNA Rev. 9.1 and above: 0 or OFF

Before Rev 9.1: 1 or ON

---

**SENSe<cnum>:CORR:IMD:MPR <num>**

(Read-Write) Sets and returns the maximum intermod product frequencies to be calibrated. All lower product frequencies are also calibrated.

**Parameters**

- **<cnum>** Any existing channel number; if unspecified, value is set to 1.
- **<char>** Maximum IM products to calibrate. Choose from:
  - 2 - second order products
  - 3 - third order products
  - 5 - fifth order products
  - 7 - seventh order products
  - 9 - ninth order products

**Examples**

```
SENS:CORR:IMD:MPR 5
sense2:correction:imd:mproduct 9
```

2557
SENSe<cnum>::CORRection:IMD:POWer <num>

(Read-Write) Sets and returns the power level at which to perform the source power calibration using a power sensor.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<char>`: Power level. Choose a value between the min and max power level of the PNA.

**Examples**

```
SENS:CORR:IMD:POW -5
sense2:correction:imd:power 5
```

**Query Syntax**

SENSe<cnum>::CORRection:IMD:POWer?

**Return Type**

Numeric

**Default**

0

SENSe<cnum>::CORRection:IMD:SENSor:CKIT <string>

(Read-Write) Sets and returns the cal kit to be used for calibrating at the port 1 reference plane when the power sensor connector is different from the DUT port 1. This effectively removes the effects of an adapter that is used to connect the power sensor.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<string>`: Cal Kit enclosed in quotes. First set the DUT connector for port 1 and the connector of the power sensor. Then use SENS:CORR:COLL:GUID:CKIT:PORT1:CAT? to return a list of valid cal kits.

**Examples**

```
SENS:CORR:IMD:SENS:CKIT "85052B"
```

**Query Syntax**

SENSe<cnum>::CORRection:IMD:SENSor:CKIT?

**Return Type**

String

**Default**

Depends on the specified connectors
SENSe<cnum>::CORRection:IMD:SENSor:CONNector <string>

(Read-Write) Sets and returns the power sensor connector type which is used to perform the Source Power Cal portion of an IMD calibration.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.

Select "Ignored" to NOT compensate for the adapter.

**Examples**

```
SENS:CORR:IMD:SENS:CONN "APC 3.5 male"
```

```
sense2:correction:imd:sensor:connector "Ignored"
```

**Query Syntax**

SENSe<cnum>::CORRection:IMD:SENSor:CONNector?

**Return Type** String

**Default** "Ignored"

---

SENSe<cnum>::CORRection:IMD:SORDer:INCLude <bool>

(Read-Write) Sets and returns whether to include the second order products in the calibration. These frequencies of these products can be far from the main tones.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<bool>` Choose from:

  - **ON** (or 1) - Include 2nd order products
  - **OFF** (or 0) - Do NOT include 2nd order products

**Examples**

```
SENS:CORR:IMD:SORD:INCL ON
```

**Query Syntax**

SENSe<cnum>::CORRection:IMD:SORDer:INCLude?

**Return Type** Boolean

**Default** OFF or 0

Last Modified:

- 11-Jan-2011 Minor edit
- 27-Mar-2009 Edited for IMDx
- 18-Sep-2008 MX New topic
Sense:Correction:Collect:Session Commands - Superseded

Note: These commands are replaced with Sense:Corr:Guided commands.

The commands in this topic are common to perform both SMC and VMC calibrations.

A calibration session is a term used to describe an instance of a SMC or VMC calibration. The session number is chosen in the SENS:CORR:COLL:SESS:INITiate command. All other commands refer to that session number. For more commands, see SESS:SMC and SESS:VMC.

Commands to read (STEP?) and describe (DESC?) each step are provided to facilitate a remote user interface.

Click on a blue keyword to view the command details.

See Also

- Learn about SMC and VMC calibrations
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe<ch>:CORRection:COLLect:SESSion<n>:ACQuire <step>[,sync]
(Write only) Acquire a calibration measurement.

**Parameters**

- `<ch>`  Any existing channel number. If unspecified, value is set to 1.
- `<n>`  Session number of the calibration.  
  Learn about Cal sessions.
- `<step>`  Step number to acquire. Use `SENS:CORR:COLL:SESS:STEPS?` to find the number of steps required for the calibration.
- `[sync]`  Optional argument. Choose from:  
  SYNChronous - blocks SCPI commands during standard measurement (default behavior).
  ASYNChronous - does NOT block SCPI commands during standard measurement.
  Learn more.

**Examples**

```
SENS2:CORR:COLL:SESS6:ACQ 5, ASYN; *OPC?
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:CKIT:PORT<p>:CATalog?**

(Read only) Returns a list of cal kits that are compatible with the connector on port `<p>`. The port connector type is set with `SENS:CORR:COLL:SESS:CONN:PORT`.

**Parameters**

- `<ch>`  Any existing channel number. If unspecified, value is set to 1.
- `<n>`  Session number of the calibration.  
  Learn about Cal sessions.
- `<p>`  PNA port number connector to query for compatible cal kits.  
  For VMC, output port of the calibration mixer, specify 3 unless already used for the output of the mixer. Otherwise, specify 4.

**Examples**

```
```

**Return Type**  Comma separated string values

**Default**  Not Applicable

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:CKIT:PORT<p>[:SELect] <calkit>**
(Read-Write) Set or return the Cal Kit for the specified port. Use `SENS:CORR:COLL:SESS:CKIT:PORT:CAT?` to list compatible Cal Kits.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<p>` PNA port number connector for which to specify a cal kit.
  For VMC, output port of the calibration mixer, specify 3 unless already used for the output of the mixer. Otherwise, specify 4.
- `<calkit>` Cal Kit Name

**Examples**

```
SENS:CORR:COLL:SESS:CKIT:PORT:SEL 85091A
```

**Query Syntax**


**Return Type** String

**Default** Not Applicable

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:CONNector:PORT<p>[:SELect] <conn>**

(Read-Write) Set the connector type and sex for the specified port number. Catalog valid connector types using `SENS:CORR:COLL:GUID:CONN:CAT?`

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<p>` PNA port number connector for which to specify a connector type.
- `<conn>` Name of the connector type

**Examples**

```
SENS2:CORR:COLL:SESS6:CONN:PORT1:SEL "N Type"
```

**Query Syntax**

`SENSe<ch>:CORR:COLL:SESS<n>:CONN:PORT<p>[:SELect]?`

**Return Type** String

**Default** Not Applicable
SENSe<ch>:CORRection:COLLect:SESSion<n>:DESCription? <step>

(Read-only) Returns the connection prompt for the step. List the number of steps in the calibration using SENS:CORR:COLL:SESS:STEPS?.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1
<n> Session number of the calibration.
Learn about Cal sessions.
<step> Step number


Return Type Numeric
Default Not Applicable

SENSe<ch>:CORRection:COLLect:SESSion<n>:DONE

(Write only) Ends the calibration session. First use SAVE? to calculate error terms and save the CalSet.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1
<n> Session number of the calibration.
Learn about Cal sessions.

Examples SENS1:CORR:COLL:SESS6:SAVE?
SENS1:CORR:COLL:SESS6:DONE

Query Syntax Not Applicable
Default Not Applicable

SENSe<ch>:CORRection:COLLect:SESSion<n>:INITiate <string>
(Write only) Initiates an SMC or VMC calibration session. Use the session number for subsequent SMC or VMC commands.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number. Choose from 1 to the max number of channels. If the session number already exists it will be terminated and a new session initiated.
- `<string>` Name of the calibration. Choose from:
  - "VMC" or "VectorMixerCal.VMCTYPE"
  - "SMC" or "ScalarMixerCal.SMCType"

**Examples**

SENS1:CORR:COLL:SESS6:INITiate "VectorMixerCal.VMCTYPE"

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENS<ch>:CORR:COLL:SESS<n>:SAVE?**

(Read only) Finish the SMC or VMC calibration, compute error terms, populate and save the Cal Set, and return the GUID of the Cal Set.

*Note:* The destination (Cal Register or User Cal Set) is determined by the setting of the SENS:CORR:PREFERENCE:CSET:SAVE command.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

**Examples**

SENS1:CORR:COLL:SESS6:SAVE?

**Return Type**

String specifying the GUID of the Cal Set produced by this session.

**Default**

Not Applicable

---

**SENS<ch>:CORR:COLL:SESS<n>:STEPS?**
(Read-only) Returns the number of steps required by the Calibration.

To ensure this query always completes successfully, first send the write command:
SENS:CORR:COLL:SESS:STEP, then send the query.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

[Learn about Cal sessions.](#)

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS1:CORR:COLL:SESS6:STEPS?</td>
</tr>
</tbody>
</table>

**Return Type** Numeric

**Default** Not Applicable

---

Last Modified:

- 4-Jun-2009 Edits to DONE for clarification.
- 9-Apr-2009 Fixed commands for mixer port numbers.
- 13-Feb-2009 Added sync to ACQuire
- 5-Aug-2008 Edited sessions descriptions
Perform scalar (SMC) calibration on a frequency converting device.

<table>
<thead>
<tr>
<th>SENSE:CORRectionCOLLect:SESSion:SMC</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECAL</td>
</tr>
<tr>
<td>CHARacterza</td>
</tr>
<tr>
<td>FSIMulator</td>
</tr>
<tr>
<td>NETWork</td>
</tr>
<tr>
<td>FILEname</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>IMPort</td>
</tr>
<tr>
<td>PHASE</td>
</tr>
<tr>
<td>DELay</td>
</tr>
<tr>
<td>METHod</td>
</tr>
<tr>
<td>MIXer</td>
</tr>
<tr>
<td>PWRCal</td>
</tr>
<tr>
<td>SEParate</td>
</tr>
<tr>
<td>SRCPort</td>
</tr>
<tr>
<td>TWOPort</td>
</tr>
<tr>
<td>ECAL</td>
</tr>
<tr>
<td>STATE</td>
</tr>
<tr>
<td>PORTmap</td>
</tr>
<tr>
<td>METHod</td>
</tr>
<tr>
<td>OMITisolat</td>
</tr>
<tr>
<td>OPTion</td>
</tr>
</tbody>
</table>
Click on a blue keyword to view the command details.

Red keywords are obsolete.

See Also

- Example Programs
- Learn about SMC Calibrations
- Synchronizing the PNA and Controller
- SCPI Command Tree

**NOTE:** To configure a power meter and sensor see SOURce:POWer: commands.

```plaintext
SENS<ch>:CORRection:COLLect:SESSION<n>:SMC:ECAL:CHARacteriza <mod>,<char> (Read-Write) Specifies the ECal module and characterization to be used for the SMC calibration.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

Learn about Cal sessions.

- `<mod>` 1 - Electronic Calibration Module
- `<char>` Specifies which characterization within the ECal module from which to read the confidence data.
  - 0  Factory characterization (data that was stored in the ECal module by Agilent). Default if not specified.
  - 1  User characterization #1
  - 2  User characterization #2
  - 3  User characterization #3
  - 4  User characterization #4
  - 5  User characterization #5

**Examples**

SENS:CORR:COLL:SESS:SMC:ECAL:CHAR 1, 2

**Query Syntax**


**Return Type** Numeric
```

(Read-Write) Specifies the S2P filename to embed or de-embed on the input or output of your mixer measurement. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<p>` Apply network to input or output of mixer. Choose from:
  1 - Input of mixer
  2 - Output of mixer
- `<string>` Filename of the S2P used for embedding or de-embedding. Use the full path name, file name, and .S2P suffix, enclosed in quotes.

**Examples**

**Query Syntax**

**Return Type** String

**Default** Not Applicable

(Read-Write) Allows you to embed (add) or de-embed (remove) circuit network effects on the input and output of your mixer measurement. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<p>` Apply network to input or output of mixer. Choose from:
  1 - Input of mixer
  2 - Output of mixer
- `<char>` Choose from:
  - **NONE** - Do nothing with effects of S2P file.
  - **EMBed** - Add effects of S2P file from the measurement results.
  - **DEEMbed** - Remove effects of S2P file from the measurement results.

**Examples**

```
```

**Query Syntax**

```
SENS<ch>:CORRection:COLLect:SESSion<n>:SMC:IMPort <calName>, <dataset>
```

**Return Type** Character

**Default** NONE

SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:IMPort <calName>, <dataset>
(Write-only) Imports existing Source Power Cal data into the SMC calibration.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

Learn about Cal sessions.

- `<calName>` (String) Name of existing Cal Set from which power meter data is imported.
- `<dataset>` (String) Name of the data set. Use POWER_STEP

**Examples**


**Query Syntax**

Not Applicable

**Default**

NONE

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PHASe:DELay <num>**

(Read-Write) Set and return the known delay through the calibration mixer. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

Learn about Cal sessions.

- `<char>` Known delay through the calibration mixer in seconds.

**Examples**


**Query Syntax**

SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PHASe:DELay?

**Return Type**

Numeric

**Default**

0 seconds

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PHASe:METHod <char>**
(Read-Write) Set and return the method of setting the delay through the calibration mixer. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

Learn about Cal sessions.

- `<char>` Choose from:

  - **MIXer** - use the S2P file set with `SENS:CORR:COLL:SESS:SMC:PHAS:MIX`

**Examples**

```
```

**Query Syntax**

```
SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PHASe:METHod?
```

**Return Type** Character

**Default** FIXed

---

`SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PHASe:MIXer <string>`

(Write-only) Set the filename of the S2P file used to characterize the calibration mixer. Learn more.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

Learn about Cal sessions.

- `<string>` Calibration mixer filename. Use the following rules to specify path names:
  - The default folder is "C:/Program Files/Agilent/Network Analyzer/Documents"
  - You can change the active directory using `MMEMory:CDIRectory`
  - Specify only the file name if using the active directory.
  - You can also use an absolute path name to specify the folder and file.
SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PWRCal:SEParate <bool>

(Read-Write) Specifies whether to use a Thru standard or to use two power sensor connections during the power cal of an SMC calibration.

This command must be sent BEFORE the INITiate command and all the other calibration commands.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Session number of the calibration.

<bool> OFF or 0 - Perform Cal with Thru standard.

ON or 1 - Do NOT use a Thru, but instead perform separate power cals on Input and Output reference planes.

Examples

SENS:CORR:COLL:SESS:INIT "SMC"
SENS:CORR:COLL:SESS:SMC:PWRC:SEP 1

Query Syntax


Return Type

Boolean

Default

0 or OFF

SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:PWRCal:SRCPort <string> Obsolete
Specifies which port to calibrate.

**Note:** Beginning with Rev 6.0, this command is no longer necessary. Because of improved calibration techniques, **Both** is always selected although a power meter measurement is performed only on port 1.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

*Learn about Cal sessions.*

- `<char>`
  - '1' Source port 1 (SMC forward direction)
  - '2' Source port 2 (SMC reverse direction)
  - 'BOTH' (both forward and reverse directions)

**Examples**

```
SENS:CORR:COLL:SESS:SMC:PWRC:SRCP 'both'
SENS2:CORRection:COLLection:SESSion6:SMC:PWRCal:SRCPort '2'
```

**Query Syntax**


**Return Type** String

**Default**

1

---


*(Read-Write)* Sets ECAL Auto-Orientation ON or OFF. If setting auto-orientation OFF, you must manually specify the orientation of the ECAL module with `SENS:CORR:COLL:SESS:SMC:TWOP:ECAL:PORTmap`.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.

*Learn about Cal sessions.*

- `<bool>`
  - **OFF or 0** = Orientation OFF
  - **ON or 1** = Orientation ON

**Examples**

```
```

**Query Syntax**


**Return Type** Boolean
SENSe<ch>:CORRection:COLLect:SESSION<n>:SMC:TWOPort:ECAL:PORTmap <mod>,<string>

(Read-Write) Specifies the manual orientation (which ports of the module are connected to which ports of the PNA) when auto-orientation is OFF.

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1
- **<n>** Session number of the calibration.
  
  Learn about Cal sessions.

- **<mod>** 1 - Electronic Calibration Module

- **<string>** Format in the following manner:

  Aw,Bx,Cy,Dz

  where

  - A, B, C, and D are literal ports on the ECAL module
  - w, x, y, and z are substituted for PNA port numbers to which the ECAL module port is connected.

  Ports of the module which are not used are omitted from the string.

**For example,** on a 4-port ECal module with

- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1

the string would be: A2,B3,D1

If either the receive port or source port (or load port for 2-port cal) of the
CALC:PAR:SELelected measurement is not in this string and orientation is OFF, an attempt to perform an ECal calibration will fail.

**Examples**


**Query Syntax**


**Return Type**

String

**Default**

"A1,B2"

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:TWOPort:METHod <string>**

(Read-Write) Specifies the guided ECal method for performing the thru portion of the calibration.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Session number of the calibration.

Learn about Cal sessions.

<string> ECAL Method: Choose from:

'DEFAULT' - Default

'ADAP' - Adapter removal

'FLUSH' - Flush Through

'UNKN' - Unknown Thru

**Examples**

SENS:CORR:COLL:SESS:SMC:TWOPort:METH 'default'

**Query Syntax**


**Return Type**

String

**Default**

DEFAULT

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:SMC:TWOPort:OMITisolat <bool>**
(Read-Write) Select to omit or perform the isolation portion of the ECAL.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.
  
  Learn about Cal sessions.
- `<bool>` **ON or 1** - Omit isolation
  
  **OFF or 0** - Perform isolation

**Examples**

```
SENS:CORR:COLL:SESS:SMC:TWOPort:OMIT 1
```

**Query Syntax**


**Return Type** Boolean

**Default** 1

---

**SENSe<ch>:CORRection:COLLect:SESSion<n >:SMC:TWOPort:OPTion <string>**

(Read-Write) Sets the SMC calibration to ECAL or MEChanical

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.
  
  Learn about Cal sessions.
- `<char>` Choose from:
  
  'ECAL' Electronic Calibration Module -
  
  **Note:** This selection assumes there is only one ECal module on the USB and so it selects the first enumerated module on the bus, and the factory characterization on that ECal module, to be used for the cal.
  
  'MECH' Mechanical Calibration Kit

**Examples**

```
SENS:CORR:COLL:SESS:SMC:TWOPort:OPTion 'ECAL'
```

**Query Syntax**

`SENS:CORR:COLL:SESS:SMC:TWOPort:OPTion?`

**Return Type** String

**Default** ECAL
<table>
<thead>
<tr>
<th>Date</th>
<th>Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>9-Feb-2011</td>
<td>Added note to SMC:TWOPort:OPTion 'ECAL'</td>
</tr>
<tr>
<td>1-Dec-2010</td>
<td>Superseded (A.09.33)</td>
</tr>
<tr>
<td>22-Dec-2009</td>
<td>Added Import</td>
</tr>
<tr>
<td>24-Feb-2009</td>
<td>Replaced True/False</td>
</tr>
<tr>
<td>5-Aug-2008</td>
<td>Edited sessions description</td>
</tr>
</tbody>
</table>
**Sense:Correction:Collect:Session:VMC Commands - Superseded**

**Note:** Beginning with PNA release A..09.33, these commands (commonly known as "Session" commands) were replaced with Sens:Corr:Coll:Guid:VMC commands.

Performs a vector (VMC) calibration on a frequency converting device.

```
SENS:CORR:COLL:SESS:VMC
```

Click on a blue keyword to view the command details.

**See Also**

- Example Programs
- Learn about VMC Calibration
- Synchronizing the PNA and Controller

```
SENSe<ch>:CORRection:COLLect:SESSion<n> :VMC:FSIMulator:NETWork<x>:MODE <char>
```
(Read-Write) Allows you to embed (add) or de-embed (remove) circuit network effects on the input and output of your mixer measurement. Learn more.

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<x>` Apply network to input or output of mixer. Choose from:
  1 - Input of mixer
  2 - Output of mixer
- `<char>` Choose from:
  - NONE - Do nothing with effects of S2P file.
  - EMBED - Add effects of S2P file from the measurement results.
  - DEEMBED - Remove effects of S2P file from the measurement results.

Examples

```
```

Query Syntax

```
SENS<ch>:CORRection:COLLect:SESSion<n> :VMC:FSIMulator:NETWork<x>:MODE?
```

Return Type Character

Default NONE

---

**SENSe<ch>:CORRection:COLLect:SESSion<n> :VMC:FSIMulator:NETWork<x>:FILENAME <string>**

(Read-Write) Specifies the S2P filename to embed or de-embed on the input or output of your mixer measurement. Learn more.

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<x>` Apply network to input or output of mixer. Choose from:
  1 - Input of mixer
  2 - Output of mixer

(Read-Write) Specifies the .S2P filename used for mixer characterization. Use the VMC:MIXer:CHARacterize:CAL:OPTion command to load the file for mixer characterization. Once loaded, use this command to query the current filename or set a new filename.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Session number of the calibration.

Learn about Cal sessions.

<string> Filename of the S2P used for mixer characterization. Use the full path name, file name, and .S2P suffix, enclosed in quotes.

Examples


Query Syntax


Return Type String

Default C:/Program Files/Agilent/Network Analyzer/Documents/default.s2p
(Read-Write) Sets the mixer characterization method to ECal, Mechanical, or read from a file.

Parameters

<ch>  Any existing channel number. If unspecified, value is set to 1

<n>  Session number of the calibration.

Learn about Cal sessions.

<char>  ECAL - Electronic Calibration Module

MECH - Mechanical Calibration Kit

FILE, <filename> - Retrieve a mixer characterization file. Also specify the filename of the S2P used for mixer characterization. Use the full path name, file name, and .S2P suffix. Use the VMC:CHARacterize:CAL:FILENAME command to query the filename.

Examples


'or


Return Type  String

Default  MECH


(Read-Write) Specifies the direction in which to characterize the calibration mixer. Learn more about the calibration mixer.

Parameters

<ch>  Any existing channel number. If unspecified, value is set to 1

<n>  Session number of the calibration.

Learn about Cal sessions.

<bool>  OFF (0) - Characterize the calibration mixer in the SAME direction as that specified in the mixer setup.

ON (1) - Characterize the calibration mixer in the REVERSE direction as that specified in the mixer setup.
**Examples**


**Query Syntax**


**Return Type**

Boolean

Default OFF

---

**SENSe<ch>:CORRection:COLLect:SESSion<n >:VMC:MIXer:ECAL:CHARacteriza <mod> ,<char>**

*(Read-Write)* Specifies the ECAL module and characterization to be used for the mixer characterization portion of the calibration.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<mod>` 1 - Electronic Calibration Module
- `<char>` Characterization number in the specified ECAL module. Choose from:
  - 0 Factory characterization (data that was stored in the ECAL module by Agilent). Default if not specified.
  - 1 User characterization #1
  - 2 User characterization #2
  - 3 User characterization #3
  - 4 User characterization #4
  - 5 User characterization #5

**Examples**

SENS:CORR:COLL:SESS:VMC:MIX:ECAL:CHAR 1,0

**Query Syntax**


**Return Type**

Numeric

Default 1,0

---

**SENSe<ch>:CORRection:COLLect:SESSion<n >:VMC:MIXer:ECAL:PORTmap <mod>,<string>**
(Read-Write) Sets the port mapping for the mixer characterization with ECal. This command is required if SENS:CORR:COLL:SESS:VMC:MIX:CHAR:CAL:OPT ECAL is specified.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<mod>` 1 - Electronic Calibration Module
- `<string>` Choose from:
  - "A1" - ECAL module port A is connected to PNA port 1
  - "B1" - ECAL module port B is connected to PNA port 1

**Examples**

```
```

**Query Syntax**


**Return Type** String

**Default** "A1"

---

**SENSe<ch>:CORRection:COLLect:SESSion<n>:VMC:OPERation <string>**

(Read-Write) Perform either full VMC calibration or mixer characterization only.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. Learn about Cal sessions.
- `<char>` 'CAL' - full calibration and mixer characterization
  - 'CHAR' - mixer characterization only (no reference mixer required) - Saves an .S2P file with the filename specified in SENS<ch>:CORR:COLL:SESSion<n>:VMC:CHARacterize:CAL:FILENAME <filename> . If none is specified, a filename is automatically generated and can be queried using the filename command.

**Examples**

```
SENS:CORR:COLL:SESS:VMC:OPER 'CAL'
```

**Query Syntax**

SENS:CORR:COLL:SESS:VMC:OPER?

(Read-Write) Specifies the ECAL module and characterization to be used for the VMC calibration.

Parameters

<ch>  Any existing channel number. If unspecified, value is set to 1

<n>  Session number of the calibration.

<mod>  1 - Electronic Calibration Module

<char>  Characterization number in the specified ECAL module. Choose from:

0  Factory characterization (data that was stored in the ECAL module by Agilent). Default if not specified.
1  User characterization #1
2  User characterization #2
3  User characterization #3
4  User characterization #4
5  User characterization #5

Examples  SENS:CORR:COLL:SESSION:VMC:TWOP:ECAL:CHAR 1, 1


Return Type  Integer

Default  1,0
(Read-Write) Sets ECAL orientation for the VMC ECal.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.
  - Learn about Cal sessions.
- `<bool>` **ON or 1** - Perform orientation
  - **OFF or 0** - Do NOT perform orientation

**Examples**

```
```

**Query Syntax**


**Return Type** Integer

**Default** ON

---

SENS<ch>:CORRection:COLLect:SESSION<n>:VMC:TWOPort:ECAL:PORTmap <mod>,
<string>

(Read-Write) Specifies the manual orientation (which ports of the module are connected to which ports of the PNA) when orientation is turned off.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration.
  - Learn about Cal sessions.
- `<mod>` 1 - Electronic Calibration Module
- `<string>` Port Map, formatted in the following manner:
  - Aw,Bx,Cy,Dz
  where:

  - A, B, C, and D are literal ports on the ECAL module.
  - w,x,y, z are substituted for PNA port numbers to which the ECAL module port is connected.
  - Ports of the module which are not used are omitted from the string.
For example, on a 4-port ECal module with:

- port A connected to PNA port 2
- port B connected to PNA port 3
- port C not connected
- port D connected to PNA port 1

the string would be: A2,B3,D1

If either the receive port or source port (or load port for 2-port cal) of the measurement is not in this string and orientation is OFF, an attempt to perform an ECal will fail.

**Examples**

```
```

**Query Syntax**

```
```

**Return Type** string

**Default** "A1,B2"

---

**SENSe<ch>:CORRection:COLLect:SESSion<n> :VMC:TWOPort:METHod <string>**

*(Read-Write)* Specifies the guided ECal method for performing the thru portion of the calibration.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Session number of the calibration. [Learn about Cal sessions.](#)
- `<char>`  
  - `'DEFAULT'` - Default
  - `'ADAP'` - Adapter removal
  - `'FLUSH'` - Flush Through
  - `'UNKN'` - Unknown Thru

**Examples**

```
SENS:CORR:COLL:SESS:VMC:TWOP:METH 'ADAP'
SENSe2:CORR:COLL:SESSion6:VMC:TWOPort:METHod 'FLUSH'
```

**Query Syntax**

```
SENS:CORR:COLL:SESS:VMC:TWOP:METH?
```

**Return Type** String

**Default** DEFAULT
SENSe<ch>:CORRection:COLLect:SESSion<n> :VMC:TWOPort:OMITisolat <bool>

(Read-Write) Select to omit or perform the isolation portion of the ECAL.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Session number of the calibration.

<bool> ON or 1 - omit isolation

OFF or 0 - perform isolation

Examples

SENS:CORR:COLL:SESS:VMC:TWOP:OMIT 1

Query Syntax

SENS:CORR:COLL:SESS:VMC:TWOP:OMIT?

Return Type Boolean

Default ON

SENSe<ch>:CORRection:COLLect:SESSion<n> :VMC:TWOPort:OPTion <string>

(Read-Write) Sets the 2-port calibration option to ECAL or MEChanical

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Session number of the calibration.

<char> Choose from:

'ECAL' Electronic Calibration Module

'MECH' Mechanical Calibration Kit

Examples

SENS:CORR:COLL:SESS:VMC:TWOP:OPT 'MECH'

SENSe2:CORR:COLL:SESSion6:VMC:TWOPort:OPTion 'ECAL'

Query Syntax

SENS:CORR:COLL:SESS:VMC:TWOP:OPT?

Return Type String

Default "MECH"
<table>
<thead>
<tr>
<th>Date</th>
<th>Change Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-Dec-2010</td>
<td>Superseded (A.09.33)</td>
</tr>
<tr>
<td>24-Feb-2009</td>
<td>Replaced True/False</td>
</tr>
<tr>
<td>5-Aug-2008</td>
<td>Edited session description</td>
</tr>
</tbody>
</table>
Sense:Couple Commands

SENSe:COUPlE

| PARameter | [STATe] |

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe<cnum>:COUPlE <ALL | NONE>

(Read-Write) Sets the sweep mode as Chopped or Alternate.

Learn about Alternate Sweep

Parameters

< <cnum> > Any existing channel number; if unspecified, value is set to 1.
< <ALL | NONE> > ALL - Sweep mode set to Chopped - reflection and transmission measured on the same sweep.
NONE - Sweep mode set to Alternate - reflection and transmission measured on separate sweeps. Improves Mixer bounce and Isolation measurements.
Increases sweep time

Examples

SENSe:COUPlE ALL
sense2:couple none

Query Syntax

SENSe<cnum>:COUPlE?

Return Type

Character

Default

ALL

SENSe<cnum>:COUPlE:PARameter[::STATe] <bool>
(Read-Write) Turns ON and OFF Time Domain Trace Coupling. All of the measurements in the specified channel are coupled.

- To select Transform parameters to couple, use **CALC:TRAN:COUP:PAR**
- To select Gating parameters to couple, use **CALC:FILT:COUP:PAR**

Learn more about **Time Domain Trace Coupling**.

**Parameters**

- **<cnum>**  
  Any existing channel number; if unspecified, value is set to 1.
- **<bool>**  
  **ON (or 1)** - Turns ON Time Domain Trace Coupling.

  **OFF (or 0)** - Turns OFF Time Domain Trace Coupling.

**Examples**

```
SENS:COUP:PAR 0  
sense2:couple:parameter:state on
```

**Query Syntax**  
SENS<enum>:COUPE:PARame[ter[:STATe]]?

**Return Type**  
Boolean

**Default**  
ON (or 1)
**Sense:FOM (Frequency Offset) Commands**

Controls the frequency offset settings which cause stimulus and response frequencies to be different.

**Note:** These commands replace the previous FOM commands. Although the old commands will continue to work, they can NOT be mixed with these new commands.

Click on a **blue** keyword to view the command details.

**See Also**

- FOM Example Program
- Learn about Frequency Offset
- Synchronizing the PNA and Controller
- SCPI Command Tree

\[
\text{SENSe}<\text{cnum}>:\text{FOM[:STATe]} <\text{bool}>
\]
(Read-Write) Turns Frequency Offset ON and OFF. Frequency offset settings are not enabled until this setting is ON.

Send this command (FOM ON) AFTER sending other FOM settings to avoid 'out-of-range' errors.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<bool>` ON (or 1) - turns FOM ON.
  
  OFF (or 0) - turns FOM OFF.

**Examples**

```
SENS:FOM 1
sense2:fom:state on
```

**Query Syntax**

`SENSe<cnum>:FOM:STATe?`

**Return Type** Boolean

**Default** OFF

---

**SENSe<cnum>:FOM:CATalog?**

(Read-only) Returns a comma-separated list of available range names in the PNA.

Use `SENS:FOM:CAT?` to see a list of available range names.

Use `SENS:FOM:COUNT?` to see a list of available range numbers.

Use `SENS:FOM:RNUM?` to see the range number for a specific name.

Use `SENS:FOM:RANG:NAME?` to see the range name for a specific number.

External devices can appear in the list of range names. Learn more.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.

**Examples**

```
SENS:FOM:CAT?
'returns
"Primary, Source, Receivers"
```

**Return Type** String

**Default** Not Applicable

---

**SENSe<cnum>:FOM:COUNT?**
(Read-only) Returns the number of valid range numbers in the PNA.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.

**Examples**

```
SENS:FOM:COUN?
sense2:fom:count?
```

**Query Syntax**

SENSe<cnum>:FOM:COUNt?

**Return Type** Numeric

**Default** Not Applicable

### SENSe<cnum>:FOM:DISPlay:SELect <string>

(Read-Write) Select the range to be displayed on the PNA x-axis. All traces in the channel have this same x-axis scaling.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<string>` Range name. Case insensitive. Use SENSe:FOM:CAT? to see a list of available frequency range names.

**Examples**

```
SENS:FOM:DISPLAY:SELECT "source2"
sense2:fom:display:select "source"
```

**Query Syntax**

SENSe<cnum>:FOM:DISPlay:SELect?

**Return Type** String

**Default** Receivers

### SENSe<cnum>:FOM:RNUM? <string>
(Read-only) Returns the number of a specified range name.

The FOM range items are typically numbered as follows:

1. Primary
2. Source
3. Receivers
4. Source2 (if present)

Use SENS:FOM:CAT? to see a list of available range names.
Use SENS:FOM:COUNt? to see a list of available range numbers.
Use SENS:FOM:RANG:NAME? to see the range name for a specific number.

External devices can appear in the list of range names. Learn more.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<string>` Range name for which a number is being queried. Case insensitive.

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:FOM:RNUM?</td>
<td>Returns the range number.</td>
</tr>
<tr>
<td>sense2:fom:rnum?</td>
<td>Returns the range number for a specific name.</td>
</tr>
</tbody>
</table>

Default Not Applicable

SENSe<cnum>:FOM:RANGE<n>:COUPled <bool>

(Read-Write) Sets and returns the state of coupling (ON or OFF) of the specified range to the primary range.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number to couple to primary range. An error is returned when attempting to couple to the Primary range (1).

Use SENS:FOM:CAT? to see a list of available range names.
Use SENS:FOM:COUNt? to see a list of available range numbers.
Use SENS:FOM:RNUM? to see the range number for a specific name.
Use SENS:FOM:RANG:NAME? to see the range name for a specific number.
<bool> ON (or 1) - Couple range to primary range.
OFF (or 0) - Do NOT couple to primary range.

Examples
SENS:FOM:RANG:COUP 1
sense2:fom:range2:coupled 0

Query Syntax SENSE<cnum>:FOM:RANGE<n>:COUPled?
Return Type Boolean

Default ON (or 1) Coupled

SENSe<cnum>:FOM:RANGe<n>:FREQuency:CW <num>

(Read-Write) Sets and returns the CW frequency.
This setting is valid for the primary range, or if the specified range is already uncoupled from the primary range and if the sweep type is CW.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.
<n> Range number. If unspecified, value is set to 1.
Use SENS:FOM:CAT? to see a list of available range names.
Use SENS:FOM:COUNt? to see a list of available range numbers.
Use SENS:FOM:RNUM? to see the range number for a specific name.
Use SENS:FOM:RANG:NAME? to see the range name for a specific number.
<br>
<num> CW frequency value in Hz. Choose any frequency within the range of the PNA.

Examples SENS:FOM:RANG:FREQ:CW 1e9
sense2:fom:range2:frequency:cw 10000000

Query Syntax SENSE<cnum>:FOM:RANGE<n>:FREQuency:CW?
Return Type Numeric

Default Center frequency of the PNA.

SENSe<cnum>:FOM:RANGe<n>:FREQuency:DIVisor <num>
(Read-Write) Sets and returns the divisor value.
This setting is valid only if the specified range is coupled to the primary range.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.
<n> Range number. If unspecified, value is set to 1.
Use SENS:FOM:CAT? to see a list of available range names.
Use SENS:FOM:COUNT? to see a list of available range numbers.
Use SENS:FOM:RNUM? to see the range number for a specific name.
Use SENS:FOM:RANG:NAME? to see the range name for a specific number.
<num> Divisor value (unitless).

Examples
SENS:FOM:RANG:FREQ:DIV 3
sense2:fom:range2:frequency:divisor 0

Query Syntax
SENSe<cnum>:FOM:RANGe<n>:FREQuency:DIVisor?

Return Type
Numeric

Default 1

SENSe<cnum>:FOM:RANGe<n>:FREQuency:MULTiplier <num>
(Read-Write) Sets and returns the multiplier value to be used when coupling this range to the primary range.
This setting is valid only if the specified range is coupled to the primary range.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.
<n> Range number. If unspecified, value is set to 1.
Use SENS:FOM:CAT? to see a list of available range names.
Use SENS:FOM:COUNT? to see a list of available range numbers.
Use SENS:FOM:RNUM? to see the range number for a specific name.
Use SENS:FOM:RANG:NAME? to see the range name for a specific number.
<num> Multiplier value. (Unitless)

Examples
SENS:FOM:RANG:FREQ:MULT 1
sense2:fom:range2:frequency:multiplier 2
### Query Syntax

```
SENSe<cnum>:FOM:RANGe<n>:FREQuency:OFFSet <num>
```

(Read-Write) Sets and returns the offset value to be used when coupling this range to the primary range. Learn more about offset value.

This setting is valid only if the specified range is coupled to the primary range.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
  - Use `SENSe:FOM:CAT?` to see a list of available range names.
  - Use `SENSe:FOM:COUNT?` to see a list of available range numbers.
  - Use `SENSe:FOM:RNUM?` to see the range number for a specific name.
  - Use `SENSe:FOM:RANG:NAME?` to see the range name for a specific number.
- `<num>` Offset value. (Unitless)

**Examples**

```
SENSe:FOM:RANG:FREQ:OFFS 1E9
sense2:fom:range2:frequency:offset 10000000
```

### Query Syntax

```
SENSe<cnum>:FOM:RANGe<n>:FREQuency:STARt <num>
```

**Return Type** Numeric

**Default** 1

**Return Type** Numeric

**Default** 0
(Read-Write) Sets and returns the Start value of frequency range. Also specify Stop frequency.
This setting is valid for the primary range, or if the specified range is already uncoupled from the primary range and if the sweep type is LOG or LINear.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
  - Use `SENS:FOM:CAT?` to see a list of available range names.
  - Use `SENS:FOM:COUNt?` to see a list of available range numbers.
  - Use `SENS:FOM:RNUM?` to see the range number for a specific name.
  - Use `SENS:FOM:RANG:NAME?` to see the range name for a specific number.
- `<num>` Start value in Hz. Choose any frequency within the range of the PNA.

Examples

`SENS:FOM:RANG:FREQ:STAR 1GHz`
`sense2:fom:range2:frequency:start 100000000`

Query Syntax

`SENSe<cnum>:FOM:RANGe<n>:FREQuency:STARt?`

Return Type

Numeric

Default

Minimum frequency of the PNA.

---

`SENSe<cnum>:FOM:RANGe<n>:FREQuency:STOP <num>`

(Read-Write) Sets and returns the Stop value of frequency range. Also specify Start frequency.
This setting is valid for the primary range, or if the specified range is already uncoupled from the primary range and if the sweep type is LOG or LINear.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
  - Use `SENS:FOM:CAT?` to see a list of available range names.
  - Use `SENS:FOM:COUNt?` to see a list of available range numbers.
  - Use `SENS:FOM:RNUM?` to see the range number for a specific name.
  - Use `SENS:FOM:RANG:NAME?` to see the range name for a specific number.
- `<num>` Stop value in Hz. Choose any frequency within the range of the PNA.


**SENSe<cnum>:FOM:RANGe<n>:NAME?**

*Read-only* Returns the name of range<n>.
The FOM range items are typically named as follows:

1. Primary
2. Source
3. Receivers
4. Source2 (if present)

Use **SENSe:FOM:CAT?** to see a list of available range names.
Use **SENSe:FOM:COUNt?** to see a list of available range numbers.
Use **SENSe:FOM:RNUM?** to see the range number for a specific name.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.

**Examples**

```sql
SENSe:FOM:RANG:NAME?
sense2:fom:range2:name?
```

**Return Type** String

**Default** Not Applicable

---

**SENSe<cnum>:FOM:RANGe<n>:SWEep:TYPE <char>**

---

2600
(Read-Write) Sets and returns the sweep type to be used with the specified range.
This setting is valid only if the specified range is already uncoupled from the primary range.
Learn about Unsupported Sweep Type combinations.

Parameters

- `<cnum>`  Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
  Use SENS:FOM:CAT? to see a list of available range names.
  Use SENS:FOM:COUNt? to see a list of available range numbers.
  Use SENS:FOM:RNUM? to see the range number for a specific name.
  Use SENS:FOM:RANG:NAME? to see the range name for a specific number.
- `<char>` Sweep type. Choose from:
  - CW - Also specify CW frequency.
  - LINear - Also specify frequency Start/Stop or Center/Span
  - LOG - Also specify frequency Start/Stop or Center/Span
  - PHASE - See all Phase sweep settings.
  - POWER - Also specify power Start/Stop or Center/Span
  - SEGMENT - Also specify segment sweep settings.

Examples

SENS:FOM:RANG:SWE:TYPE LOG
sense2:fom:range2:sweep:type linear:

Query Syntax
SENSe<cnum>:FOM:RANGe<n>:SWEep:TYPE?

Return Type
Character

Default
Linear

Last Modified:
7-Feb-2011 Added phase sweep
24-Sep-2008 Updated with range queries
5-Jul-2007 Fixed state link
8-Mar-2007 MQX New topic
**Sense:FOM:Range:Segment Commands**

Constructs a segment table for a specified **UNCOUpled** FOM range.

**Note:** Do NOT use **Sens:Segment** commands for FOM segment sweep.

Click on a blue keyword to view the command details.

**See Also**

- Other **SENSe:FOM Commands**
- **Example Programs**
- **Synchronizing the PNA and Controller**
- **SCPI Command Tree**

**SENSe<cnum>:FOM:RANGE<n>:SEGMENT<s>:ADD**
(Write-only) Adds a segment.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<n>`: Range number. If unspecified, value is set to 1.
- `<s>`: Segment number to add. If unspecified, value is set to 1. Segment numbers must be sequential. If a new number is added where one currently exists, the existing segment and those following are incremented by one.

**Examples**

Two Segments exist (1 and 2). The following command will add a new segment (1). The existing (1 and 2) will become (2 and 3) respectively.

```
sense2:fom:range2:segment:add
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<cnum>:FOM:RANGe<n>:SEGment<s>:BWIDth[:RESolution] <num>**

(Read-Write) Sets the IF Bandwidth for the specified segment. First set

```
SENSe:FOM:RANGe:SEGm:BWIDth:CONTrol ON
```

All subsequent segments that are added assume the new IF Bandwidth value.

Valid either for Receiver range or for Primary range when coupled to Receiver.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<n>`: Range number. If unspecified, value is set to 1.
- `<s>`: Segment number for which to set independent IF Bandwidth.
- `<num>`: IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. See the list of valid IFBW values. If an invalid number is specified, the analyzer will round up to the closest valid number.

**Examples**

```
SENSe:FOM:RANGe:SEGm:BWIDth 100
sense2:fom:range2:segment4:bwidth:resolution 1e3
```

**Query Syntax**

SENSe<cnum>:FOM:RANGe<n>:SEGment<s>:BWIDth[:RESolution]?

**Return Type**

Numeric

**Default**

See Preset IFBW for your PNA model.
SENSe<cnum>:FOM:RANGe<n>:SEGment:BWIDth[:RESolution]:CONTrol <bool>

(Read-Write) Specifies whether the IF Bandwidth resolution can be set independently for each segment. When set, each segment added after this will be set to ON automatically.

Valid either for Receiver range or for Primary range. Primary range value is ignored unless Receiver is coupled to Primary.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<bool>`
  - ON (or 1) - turns Bandwidth control ON. Bandwidth can be set for each segment
  - OFF (or 0) - turns Bandwidth control OFF. Use the channel IF bandwidth setting instead.

Examples

```
SENSe:FOM:RANG:SEG:BWIDth:CONT 0
sense2:fom:range2:segment:bwidth:resolution:control 1
```

Query Syntax

```
SENSe<cnum>:FOM:RANGe<n>:SEGMe nt:BWIDth[:RESolution]:CONTrol?
```

Return Type

Boolean

Default

OFF

SENSe<cnum>:FOM:RANGe<n>:SEGment:COUNt?

(Read-only) Returns the number of segments that exist for the specified range.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.

Examples

```
SENSe:FOM:RANG:SEG:COUNT?
sense2:fom:range2:segment:count?
```

Return Type

Numeric

Default

Not Applicable

SENSe<cnum>:FOM:RANGe<n>:SEGment<s>:DELe te
(Write-only) Deletes the specified sweep segment.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.
<n> Range number. If unspecified, value is set to 1.
<s> Number of the segment to delete. If unspecified, value is set to 1.

Examples

SENS:FOM:RANG:SEGM3:DEL
sense2:fom:range2:segment4:delete

Query Syntax

Not Applicable

Default

Not Applicable

SENSe<cnum>:FOM:RANGe<n>:SEGMent:DELete:ALL

(Write-only) Deletes all sweep segments in the specified range.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.
<n> Range number. If unspecified, value is set to 1.

Examples

SENS:FOM:RANG:SEGM:DEL:ALL
sense2:fom:range2:segment:delete:all

Query Syntax

Not Applicable

Default

Not Applicable

SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:FREQuency:CENTer <num>
(Read-Write) Sets and returns the center frequency for the specified sweep segment. Also specify segment frequency span.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<s>` Segment number to modify. Choose any existing segment number.
- `<num>` Center Frequency in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

**Examples**

```
SENS: FOM: RANG: SEGM: FREQ: CENT 1GHz
sense2: fom: range2: segment4: frequency: center 1e9
```

**Query Syntax**

```
SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:FREQuency:CENTer?
```

**Return Type** Numeric

**Default** Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

---

SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:FREQuency:SPAN <num>

(Read-Write) Sets and returns the span frequency for the specified sweep segment. Also specify segment center frequency.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<s>` Segment number to modify. Choose any existing segment number.
- `<num>` Frequency span in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

**Examples**

```
SENS: FOM: RANG: SEGM: FREQ: SPAN 1GHz
sense2: fom: range2: segment4: frequency: span 1e9
```

**Query Syntax**

```
SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:FREQuency:SPAN?
```

**Return Type** Numeric
**SENSe\(<cnum>\):FOM:RANGe\(<n>\):SEGMent\(<s>\):FREQuency:STARt \(<num>\)**

*(Read-Write)* Sets and returns the start frequency for the specified sweep segment. Also specify segment stop frequency.

All other segment Start and Stop Frequency values that are larger than this frequency are changed to this frequency.

**Parameters**

- \(<cnum>\) Any existing channel number; if unspecified, value is set to 1.
- \(<n>\) Range number. If unspecified, value is set to 1.
- \(<s>\) Segment number to modify. Choose any existing segment number.
- \(<num>\) Start frequency in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

**Examples**

```
SENS:FOM:RANG:SEGM:FREQ:STAR 1GHz
sense2:fom:range2:segment4:frequency:start 1e9
```

**Query Syntax**

SENSe\(<cnum>\):FOM:RANGe\(<n>\):SEGMent\(<s>\):FREQuency:STARt?

**Return Type** Numeric

**Default** If first segment, frequency span of the analyzer. Otherwise 0.

---

**SENSe\(<cnum>\):FOM:RANGe\(<n>\):SEGMent\(<s>\):FREQuency:STOP \(<num>\)**

Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.
Sets and returns the stop frequency for the specified sweep segment. Also specify segment start frequency.

All other segment Start and Stop Frequency values that are larger than this frequency are changed to this frequency.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<n>`: Range number. If unspecified, value is set to 1.
- `<s>`: Segment number to modify. Choose any existing segment number.
- `<num>`: Stop frequency in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

**Examples**

```
SENS:FOM:RANG:SEGM:FREQ:STOP 1GHz
sense2:fom:range2:segment4:frequency:stop 1e9
```

**Query Syntax**

```
SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:FREQuency:STOP?
```

**Return Type**

Numeric

**Default**

Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

---

Sets the Port Power level for the specified sweep segment. First set SENS:FOM:RANG:SEGM:POW:CONTrol ON.

When port power is Coupled, setting port power for one port will apply port power for all source ports.

All subsequent segments that are added assume the new Power Level value.

Valid either for Source ranges or for Primary range when coupled to the source.

**Parameters**

- `<cnum>`: Any existing channel number; if unspecified, value is set to 1.
- `<n>`: Range number. If unspecified, value is set to 1.
- `<s>`: Segment number to modify. Choose any existing segment number.
- `<p>`: Port number of the source. If unspecified, value is set to 1.
- `<num>`: Power level in dBm.
Note: The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, send SOUR:POW? MAX and SOUR:POW? MIN. (SOUR:POW:ATT:AUTO must be set to ON).

Actual achievable leveled power depends on frequency.

Examples

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:FOM:RANGe&lt;n&gt;:SEGMent&lt;s&gt;:POWer&lt;p&gt;[:LEVel]?</td>
<td>Numeric</td>
<td>0</td>
</tr>
</tbody>
</table>

SENSe<cnum>:FOM:RANGe<n>:SEGMent:POWer[:LEVel]:CONTrol <bool>

(Read-Write) Specifies whether Power Level is to be set independently for each segment.
Valid either for Source ranges or for Primary range. Primary range value is ignored unless Source is coupled to Primary.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<bool>` ON (or 1) - Power level will be set for each segment.
  OFF (or 0) - Use the channel power level setting.

Examples

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:FOM:RANGe&lt;n&gt;:SEGMent:POWer[:LEVel]:CONTrol?</td>
<td>Boolean</td>
<td>OFF (or 0)</td>
</tr>
</tbody>
</table>

SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>[:STATe] <bool>
(Read-Write) Turns the specified sweep segment ON or OFF.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<s>` Segment number to be turned ON or OFF. Choose any existing segment number.
- `<bool>` ON (or 1) - turns segment ON.
  OFF (or 0) - turns segment OFF.

Examples

```plaintext
SENS:FOM:RANG:SEGM 0
sense2:fom:range2:segment4:state on
```

Query Syntax

```
SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>[STATe]?
```

Return Type

Boolean

Default

OFF (or 0)

---

**SENS<cnue>:FOM:RANG<n>:SEGM<s>:SWEep:POINts <num>**

(Read-Write) Sets the number of data points for the specified sweep segment.

Parameters

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Range number. If unspecified, value is set to 1.
- `<s>` Segment number to modify. Choose any existing segment number.
- `<num>` Number of points in the segment. The total number of points in all segments cannot exceed 20001. A segment can have as few as 1 point.

Examples

```plaintext
SENS:FOM:RANG:SEGM:SWE:POIN 101
sense2:fom:range2:segment4:sweep:points 201
```

Query Syntax

```
SENSe<cnum>:FOM:RANGe<n>:SEGMent<s>:SWEep:POINts?
```

Return Type

Numeric

Default

21
SENSe\textless cnum\textgreater :FOM:RANGE\textless n\textgreater :SEGMENT\textless s\textgreater :SWEep:TIME \textless num\textgreater

\textit{(Read-Write)} Sets the time the PNA takes to sweep the specified segment.
Valid ONLY for receiver ranges.

\textbf{Parameters}

\begin{itemize}
  \item \texttt{\textless cnum\textgreater} Any existing channel number; if unspecified, value is set to 1.
  \item \texttt{\textless n\textgreater} Range number. If unspecified, value is set to 1.
  \item \texttt{\textless s\textgreater} Segment number for which to set sweep time.
  \item \texttt{\textless num\textgreater} Sweep time in seconds. Choose a number between 0 and 100
\end{itemize}

\textbf{Examples}

\begin{verbatim}
SENS:FOM:RANGE:SEGMENT:SWE:TIME 1
sense2:fom:range2:segment3:sweep:time 1
\end{verbatim}

\textbf{Query Syntax}

SENSe\textless cnum\textgreater :FOM:RANGE\textless n\textgreater :SEGMENT\textless s\textgreater :SWEep:TIME?

\textbf{Return Type} Numeric

\textbf{Default} Not Applicable

SENSe\textless cnum\textgreater :FOM:RANGE\textless n\textgreater :SEGMENT:SWEep:TIME:CONTrol \textless bo\textgreater

\textit{(Read-Write)} Specifies whether Sweep Time can be set independently for each sweep segment.
Valid either for Receiver ranges or for Primary range. Primary range value is ignored unless Receiver is \texttt{coupled} to Primary.

\textbf{Parameters}

\begin{itemize}
  \item \texttt{\textless cnum\textgreater} Any existing channel number; if unspecified, value is set to 1.
  \item \texttt{\textless n\textgreater} Range number. If unspecified, value is set to 1.
  \item \texttt{\textless bo\textgreater} \texttt{ON} (or 1) - Sweep time will be set for each segment.
  \texttt{OFF} (or 0) - Use the channel sweep time setting.
\end{itemize}

\textbf{Examples}

\begin{verbatim}
SENS:FOM:RANGE:SEGMENT:SWE:TIME:CONT 1
sense2:fom:range2:segment:sweep:time:control off
\end{verbatim}

\textbf{Query Syntax}

SENSe\textless cnum\textgreater :FOM:RANGE\textless n\textgreater :SEGMENT:SWEep:TIME:CONTrol?

\textbf{Return Type} Boolean
Last Modified:

21-Jun-2007 Increased max number of points
Sense:Frequency Commands

Sets the frequency sweep functions of the analyzer.

SENSe: FREQuency

CENTER  [CW | FIXED]  SPAN  START  STOP

Click on a blue keyword to view the command details.

See Also

- Example using some of these commands.
- Learn about Frequency Sweep
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe<cnum>:FREQuency:CENTer <num>

(Read-Write) Sets the center frequency of the analyzer.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Center frequency. Choose any number between the minimum and maximum frequency limits of the analyzer. Units are Hz.

This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

`SENS:FREQ:CENT 1000000`

`sense2:frequency:center 1mhz`

`sense2:frequency:center 1e6`

**Query Syntax**

SENSe<cnum>:FREQuency:CENTer?

**Return Type**

Numeric

**Default**

Center of the analyzer's frequency span

SENSe<cnum>:FREQuency[:CW | :FIXed] <num>
(Read-Write) Sets the Continuous Wave (or Fixed) frequency. Must also send SENS:SWEEP:TYPE CW to put the analyzer into CW sweep mode.

### Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` CW frequency. Choose any number between the **minimum** and **maximum** frequency limits of the analyzer. Units are Hz.

This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

### Examples

```
SENS:FREQ 1000000
SENS:FREQ:CW MIN
sense2:frequency:fixed 1mhz
```

### Query Syntax

`SENSe<cnum>:FREQuency[:CW | :FIXed]?`

### Return Type

Numeric

**Default**

1 GHz

---

**SENSe<cnum>:FREQuency:SPAN <num>**

(Read-Write) Sets the frequency span of the analyzer.

### Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Frequency span in Hz. Choose any number between 0 (minimum) and the **maximum** frequency span of the analyzer.

This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

### Examples

```
SENS:FREQ:SPAN 1000000
sense2:frequency:span max
```

### Query Syntax

`SENSe<cnum>:FREQuency:SPAN?`

### Return Type

Numeric

**Default**

Maximum frequency span of the analyzer

---

**SENSe<cnum>:FREQuency:STARt <num>**
(Read-Write) Sets the start frequency of the analyzer.

**Parameters**

- `<cnun>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Start frequency. Choose any number between the **MIN** and **MAX** frequency limits of the analyzer. Units are Hz.

  If `FREQ:START` is set greater than `FREQ:STOP`, then `STOP` is set equal to `START`.

  This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
SENS:FREQ:STAR 1000000
sense2:frequency:start MIN
```

**Query Syntax**

`SENSe<cnun>:FREQuency:STARt?`

If **Sweep type is segment**, this query returns the start frequency of the first segment.

**Return Type** Numeric

**Default** Minimum frequency of the analyzer

---

**SENSe<cnun>:FREQuency:STOP <num>**

(Read-Write) Sets the stop frequency of the analyzer.

**Parameters**

- `<cnun>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Stop frequency. Choose any number between: the **minimum** and **maximum** frequency limits of the analyzer. Units are Hz.

  If `FREQ:STOP` is set less than `FREQ:START`, then `START` will be set equal to `STOP`.

  This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
SENS:FREQ:STOP 1000000
sense2:frequency:stop max
```

**Query Syntax**

`SENSe<cnun>:FREQuency:STOP?`

If **Sweep type is segment**, this query returns the stop frequency of the last segment.

**Return Type** Numeric

**Default** Maximum frequency of the analyzer

---

2615
Last Modified:

5-Feb-2008    Modified Max and Min notes
SENe:GCSetup Commands

Controls the Gain Compression configuration.

SENe:GCSetup:

AMODe <char>

COMPression:

| ALGorithm <char>
| BACKoff:LEVel <num>
| DELTa:X <num>
| DELTa:Y <num>
| INTerpolation <bool>
| LEVel <num>
| SATuration:LEVel

EOSoperation <string>

PMAP

| INPut?
| OUTPut?

PWer:

| LINear:INPut:LEVel <num>
| REVerse:LEVel <num>
| STARt:LEVel <num>
| STOP:LEVel <num>

SAFE:

| CPADjustment <num>
| ENABle <bool>
| FPADjustment <num>
| FTHReshold <num>
| MLIMit <num>
Click on a blue keyword to view the command details.

Other Gain Compression commands

The calibration commands listed in this topic are supplemental to the Guided Cal commands.

- **CALC:CUSTom:DEFine** - creates a gain compression measurement.
- **CALC:GCMeas:ANAL** - Gain Compression Analysis settings
- **GCA Calibration** uses the Guided Calibration commands, except for the following:
  - **Sens:Corr:GCS:Power** - sets power level for Source Power Cal

See Also

- **Example Programs**
  - [Create and Cal a Gain Compression Measurement](#)
  - [Create and Cal a GCX Measurement](#)
- **Learn about Gain Compression Application**
- **Synchronizing the PNA and Controller**
- **SCPI Command Tree**

**SENSe<ch>:GCSetup:AMODe <char>**
(Read-Write) Set and read the method by which gain compression data is acquired.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<char>` Choose from:
  - PFREQuency - 2D Power Per Frequency
  - FPOWer - 2D Frequency Per Power
  - SMARtsweep - Smart Sweep

**Examples**

```
SENS:GCS:AMOD SHAR
sense:gcsetup:amode pfrequency
```

**Query Syntax** `SENS<ch>:GCSetup:AMODe ?`

**Return Type** Character

**Default** SMARtsweep

---

**SENSe<ch>:GCSetup:COMPression:ALGorithm <char>**

(Read-Write) Set and read the algorithm method used to compute gain compression.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<char>` Algorithm method. Choose from:
  - CFLG - compression from linear gain
  - CFMG - compression from maximum gain
  - BACKoff - compression from BackOff
  - XYCOM - X/Y Compression
  - SAT - compression from saturation

**Examples**

```
SENS:GCS:COMP:ALG BACK
sense:gcsetup:compression:algorithm XYcom
```

**Query Syntax** `SENS<ch>:GCSetup:COMPression:ALGorithm?`

**Return Type** Character

**Default** CFLG
SENSe<ch>:GCSetup:COMPression:BACKoff:LEVel <num>
(Read-Write) Set and read value for the BackOff compression algorithm.

Parameters
- <ch> Any existing GCA channel. If unspecified, value is set to 1.
- <num> Backoff value in dB. Choose from 30 to (-30)

Examples
- SENS:GCS:COMP:BACK:LEV 10
- sense:gcsetup:compression:backoff:level 5

Query Syntax
SENSe<ch>:GCSetup:COMPression:BACKoff:LEVel?

Return Type Numeric

Default 10

SENSe<ch>:GCSetup:COMPression:DELTa:X <num>
(Read-Write) Set and read the 'X'' value in the delta X/Y compression algorithm.

Parameters
- <ch> Any existing GCA channel. If unspecified, value is set to 1.
- <num> X value in dB. Choose from 30 to (-30)

Examples
- SENS:GCS:COMP:DELT:X 9
- sense:gcsetup:compression:delta:X 8

Query Syntax
SENSe<ch>:GCSetup:COMPression:DELTa:X?

Return Type Numeric

Default 10

SENSe<ch>:GCSetup:COMPression:DELTa:Y <num>
(Read-Write) Set and read the "Y" value in the delta X/Y compression algorithm.

**Parameters**
- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.
- `<num>`: Y value in dB. Choose from 30 to (-30)

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:GCS:COMP:DELT:Y</td>
<td>9</td>
</tr>
<tr>
<td>sense:gcsetup:compression:delta:Y</td>
<td>8</td>
</tr>
</tbody>
</table>

**Query Syntax**
SENSe<ch>:GCSetup:COMPression:DELTa:Y?

**Return Type**
Numeric

**Default**
9

---

**SENSe<ch>:GCSetup:COMPres:sion:INTerpоляtion <bool>**

(Read-Write) Sets whether or not interpolation should be performed on 2D measured compression data. Applies ONLY to 2D acquisition modes.

**Parameters**
- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.
- `<bool>`: Choose from:
  - **ON** or (1) Interpolate the results
  - **OFF** or (0) Do NOT interpolate the results but return the value closest to compression.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:GCS:COMP:INT</td>
<td>1</td>
</tr>
<tr>
<td>sense:gcsetup:compression:interpolation</td>
<td>off</td>
</tr>
</tbody>
</table>

**Query Syntax**
SENSe<ch>:GCSetup:COMPres:sion:INTerpоляtion?

**Return Type**
Boolean

**Default**
OFF

---

**SENSe<ch>:GCSetup:COMPression:LEVel <num>**

2621
(Read-Write) Set and read the desired gain reduction (from reference gain).

This value is used for Compression from Linear Gain and Compression from Maximum Gain.

Use \texttt{SENS:GCS:COMP:ALG CFLG} to set this compression algorithm.

\textbf{Parameters}

- \texttt{<ch>}: Any existing GCA channel. If unspecified, value is set to 1.
- \texttt{<num>}: Compression level in dB. Choose a value greater than 0.1 dB.

\textbf{Examples}

\begin{verbatim}
SENS:GCS:COMP:LEV 1
sense:gcsetup:compression:level 3
\end{verbatim}

\textbf{Query Syntax}

\texttt{SENSe<ch>:GCSetup:COMPression:LEVel?}

\textbf{Return Type}

Numeric

\textbf{Default}

1

\begin{verbatim}
SENSe<ch>:GCSetup:COMPression:SATuration:LEVel <num>
\end{verbatim}

(Read-Write) Set and read the deviation dB from the maximum Pout. This is the point of saturation.

Use \texttt{SENS:GCS:COMP:ALG CFLG} to set this compression algorithm.

\textbf{Parameters}

- \texttt{<ch>}: Any existing GCA channel. If unspecified, value is set to 1.
- \texttt{<num>}: Saturation level in dB. Choose a value greater than 0.01 dB.

\textbf{Examples}

\begin{verbatim}
SENS:GCS:COMP:SAT:LEV 1
sense:gcsetup:compression:saturation:level 3
\end{verbatim}

\textbf{Query Syntax}

\texttt{SENSe<ch>:GCSetup:COMPressionSATuration:LEVel?}

\textbf{Return Type}

Numeric

\textbf{Default}

.1 dB

\begin{verbatim}
SENSe<ch>:GCSetup:EOSoperation <char>
\end{verbatim}
(Read-Write) Set and read the This setting is used to protect a sensitive device from too much power during the sweep retrace. Other instrument settings or channels may over-ride this setting. Learn more.

Parameters

<ch> Any existing GCA channel. If unspecified, value is set to 1.
<char> End Of Sweep operation. Choose from:

- STANdard Use the default PNA method. Learn more.
- POFF Always turn power OFF while waiting.
- PSTArt Sweep Start power
- PSTOp Sweep Stop power.

Examples

```
SENS:GCS:EOS PSTA
```

```
sense:gcsetup:eosoperation standard
```

Query Syntax
SENSe<ch>:GCSetup:EOSperation?

Return Type
Character

Default
STANdard

SENSe<ch>:GCSetup:PMAP <in>,<out>

(Write-only) Set the DUT-to-PNA port mapping for the Gain Compression measurement.

Parameters

<ch> Any existing GCA channel. If unspecified, value is set to 1.
<in> PNA port which is connected to the DUT input.
<out> PNA port which is connected to the DUT output.

Examples

```
SENS:GCS:PMAP 1,2
```

```
sense:gcsetup:pmap 2,1
```

Query Syntax
Not Applicable

Default
1,2

SENSe<ch>:GCSetup:PMAP:INPut?
(Read-only) Read the PNA port number to be connected to the DUT Input.

Use `SENS:GCS:PORTMap` to set the port mapping.

### Parameters

- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.

### Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>SENS:GCS:PMAP:INP?</code></td>
<td>Get the input port mapping</td>
</tr>
<tr>
<td><code>sense:gcsetup:pmap:input?</code></td>
<td>Get the input port mapping</td>
</tr>
</tbody>
</table>

### Return Type

- **Numeric**

### Default

- **1**

---

`SENS<ch>:GCSetup:PMAP:OUTPut?`

(Read-only) Read the PNA port number to be connected to the DUT Output.

### Parameters

- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.

### Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>SENS:GCS:PMAP:OUTP?</code></td>
<td>Get the output port mapping</td>
</tr>
<tr>
<td><code>sense:gcsetup:pmap:output?</code></td>
<td>Get the output port mapping</td>
</tr>
</tbody>
</table>

### Return Type

- **Numeric**

### Default

- **2**

---

`SENS<ch>:GCSetup:POWer:LINear:INPut:LEVel <num>`

(Read-Write) Set and read the input power at which Linear Gain and all S-parameters are measured.

### Parameters

- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.
- `<num>`: Input power level in dBm. Choose a value from +30 to (-30).

### Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>SENS:GCS:POW:LIN:INP:LEV 0</code></td>
<td>Set the input power to 0 dBm</td>
</tr>
<tr>
<td><code>sense:gcsetup:power:linear:input:level -10</code></td>
<td>Set the input power to -10 dBm</td>
</tr>
</tbody>
</table>

### Query Syntax

`SENS<ch>:GCSetup:POWer:LINear:INPut:LEVel?`

### Return Type

- **Numeric**

### Default

- **-25 dBm**
SENSe<ch>:GCSetup:POWer:REVerse:LEVel <num>

(Read-Write) Set and read the reverse power level to the DUT. This is applied to the DUT output port when making reverse measurements like S22.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Reverse power level in dBm. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:POW:REV:LEV 0
sense:gcsetup:power:reverse:level -5
```

**Query Syntax**

`SENSe<ch>:GCSetup:POWer:REVerse:LEVel?`

**Return Type**

Numeric

**Default**

-5

---

SENSe<ch>:GCSetup:POWer:STARt:LEVel <num>

(Read-Write) Set and read the start power level.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Start power level in dBm. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:POW:STAR:LEV 0
sense:gcsetup:power:start:level -5
```

**Query Syntax**

`SENSe<ch>:GCSetup:POWer:STARt:LEVel?`

**Return Type**

Numeric

**Default**

-30

---

SENSe<ch>:GCSetup:POWer:STOP:LEVel <num>
(Read-Write) Set and read the stop power level.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Stop power level in dBm. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:POW:STOP:LEV 0
sense:gcsetup:power:stop:level -5
```

**Query Syntax**

```
SENSe<ch>:GCSetup:POWer:STOP:LEVel?
```

**Return Type** Numeric

**Default** -5

---

**SENSe<ch>:GCSetup:SAFE:CPADjustment <num>**

(Read-Write) Set and read the Safe Sweep COARSE power adjustment. [Learn more](#).

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Coarse power adjustment setting in dBm. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:SAFE:CPAD 2
sense:gcsetup:safe:cpadjustment 3.5
```

**Query Syntax**

```
SENSe<ch>:GCSetup:SAFE:CPADjustment?
```

**Return Type** Numeric

**Default** 3.0

---

**SENSe<ch>:GCSetup:SAFE:ENABle <bool>**
(Read-Write) Set and read the (ON | OFF) state of Safe Sweep mode. Learn more

**Parameters**

- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.
- `<num>`: (Boolean) - Safe Sweep state. Choose from:
  - OFF (or 0) - Disable Safe Sweep
  - ON (or 1) - Enable Safe Sweep

**Examples**

```
SENS:GCS:SAFE:ENAB 0
sense:gcsetup:safe:enable 1
```

**Query Syntax**

SENSe<ch>:GCSetup:SAFE:ENABle?

**Return Type**

Boolean

**Default**

0

---

**SENSe<ch>:GCSetup:SAFE:FPADjustment <num>**

(Read-Write) Set and read the Safe Sweep FINE power adjustment. Learn more

**Parameters**

- `<ch>`: Any existing GCA channel. If unspecified, value is set to 1.
- `<num>`: Fine power adjustment setting in dBm. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:SAFE:FPAD 2
sense:gcsetup:safe:fpadjustment .5
```

**Query Syntax**

SENSe<ch>:GCSetup:SAFE:FPADjustment?

**Return Type**

Numeric

**Default**

1.0 dBm

---

**SENSe<ch>:GCSetup:SAFE:FTHReshold <num>**
(Read-Write) Set and read the compression level in which Safe Sweep changes from the COARSE power adjustment to the FINE power adjustment. [Learn more]

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Threshold setting in dB. Choose a value from +30 to (-30).

**Examples**

```
SENS:GCS:SAFE:FTHR .1
```

```
sense:gcsetup:safe:fthreshold .75
```

**Query Syntax**

```
SENSe<ch>:GCSetup:SAFE:FTHReshold?
```

**Return Type**

Numeric

**Default**

0.5 dB

---

`SENSe<ch>:GCSetup:SAFE:MLimit <num>`

(Read-Write) When the PNA port that is connected to the DUT Output measures this value, the input power to the DUT is no longer incremented at that frequency. Safe Mode must be enabled with `SENS:GCS:SAFE:ENAB ON` [Learn more]

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Maximum power limit in dBm. Choose a value from -100 to +100

**Examples**

```
SENS:GCS:SAFE:MLIM 20
```

```
sense:gcsetup:safe:mlimit 30
```

**Query Syntax**

```
SENSe<ch>:GCSetup:SAFE:MLIMIT?
```

**Return Type**

Numeric

**Default**

30

---

`SENSe<ch>:GCSetup:SFAilures?`
(Read-only) Returns a comma-separated list of the frequency indexes that were out of tolerance for SMART Sweep mode, or at the power limit for 2D acquisition modes. Zero (0) is the first frequency data point.

Must be Single triggered. Invalid results occur if the GCA channel is continuously sweeping.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.

**Examples**

```
SENS : GCS : SFA?
sense : gcsetup : sfailures?
```

**Return Type** Comma-separated list of frequency indexes.

**Default** Not Applicable

**SENSe<ch>:GCSetup:SMARt:MIterations <num>**

(Read-Write) Set and read the maximum permitted number of iterations which SMART Sweep may utilize to find the desired compression level, to within the specified tolerance.

**Parameters**

- `<ch>` Any existing GCA channel. If unspecified, value is set to 1.
- `<num>` Maximum number of iterations. Choose a value between 1 and 10e8

**Examples**

```
SENS : GCS : SMAR : MIT 5
sense : gcsetup : smart : miterations 3
```

**Query Syntax** `SENSe<ch>:GCSetup:SMARt:MIterations?`

**Return Type** Numeric

**Default** 20

**SENSe<ch>:GCSetup:SMARt:SIterations <bool>**
(Read-Write) Set and read enable for showing intermediate results for each iteration of SMART Sweep

Parameters

[ch] Any existing GCA channel. If unspecified, value is set to 1.
<bool> Choose from:

ON or (1) Compression traces are updated after each iteration.

OFF or (0) Compression traces are updated after ALL iterations are complete.

Examples

SENS:GCS:SMAR:SIT 1
sense:gcsetup:smart:siterations off

Query Syntax

SENSe[ch]:GCSetup:SMARt:SITerations?

Return Type

Boolean

Default

OFF

SENS[ch]:GCSetup:SMARt:STIMe <num>

(Read-Write) Set and read the amount of time SMART Sweep will dwell at the first point where the input power changes by the Backoff or X level. Applies only to SMART Sweep when Backoff or XY compression methods are selected. Learn more.

Parameters

[ch] Any existing GCA channel. If unspecified, value is set to 1.
<bool> Settling time in seconds. Choose any positive value.

Examples

SENS:GCS:SMAR:STIM 1
sense:gcsetup:smart:stime .1

Query Syntax

SENSe[ch]:GCSetup:SMARt:STIMe?

Return Type

Numeric

Default

0

SENS[ch]:GCSetup:SMARt:TOlerance <num>
(Read-Write) Set and read the acceptable range SMART Sweep will allow for the measured compression level.

**Parameters**

- <ch> Any existing GCA channel. If unspecified, value is set to 1.
- <num> Tolerance level in dBm. Choose a value between .01 and 10

**Examples**

```plaintext
SENS:GCS:SMAR:TOL .1
sense:gcsetup:smart:tolerance .05
```

**Query Syntax**

SENSe<ch>:GCSetup:SMAR:t:Lerance?

**Return Type** Numeric

**Default** .05

---

**SENSe<ch>:GCSetup:SWEep:FREQuency:POINts <num>**

(Read-Write) Set and read the number of data points in each frequency sweep. Learn more

**Parameters**

- <ch> Any existing GCA channel. If unspecified, value is set to 1.
- <num> Frequency points. Do not exceed the max number of data points.

**Examples**

```plaintext
SENS:GCS:SWE:FREQ:POIN 201
sense:gcsetup:sweep:frequency:points 101
```

**Query Syntax**

SENSe<ch>:GCSetup:SWEep:FREQuency:POINts?

**Return Type** Numeric

**Default** 201

---

**SENSe<ch>:GCSetup:SWEep:POWer:POINts <num>**
(Read-Write) Set and read the number of data points in each power sweep. Applies ONLY to 2D acquisition modes.

Parameters

<ch>   Any existing GCA channel. If unspecified, value is set to 1.
<num>  Power points. Do not exceed the max number of data points.

See Data Points Limit

Examples

SENSe:GCS:SWE:POW:POIN 50
sense:gcsetup:sweep:power:points 21

Query Syntax  SENSe<ch>:GCSetup:SWEep:POWer:POINts?
Return Type   Numeric

Default        26

Last Modified:

11-Jan-2011    Minor edits
9-Nov-2007     MX New topic
**Sense:IF Commands**

Controls the IF source and gain settings for use with the E836xC H11 Option.

**Note:** See the Sens:IF commands for the PNA-X.

![IF Command Diagram](image)

Click on a blue keyword to view the command details.

**See Also**

- Learn about the Narrowband (Pulse) Application
- See IF Access User Interface Settings
- Synchronizing the PNA and Controller
- SCPI Command Tree

---

**SENSe<cnum>:IF:FILT:FILTer:SAMPle:COUNt <num>**

(Read-Write) Sets and returns the number of taps in the IF filter. The IF filter sample count setting is only used when SENSe:IF:FILT:FILTer:SAMPle:COUNt:MODE is set to MANUAL.

For use with the E836xC Opt H11

**Parameters**

- `<cnum>` Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.
- `<num>` An integer value. (MIN and MAX return the minimum and maximum allowed values, respectively.)

**Examples**

```
SENS:IF:FILT:SAMP:COUN 40
sense2:if:filter:sample:count maximum
```

**Query Syntax** SENSe:IF:FILT:FILTer:SAMPle:COUNt?

**Return Type** Numeric

**Default** Instrument dependent.
SENSe<cnum>:IF:FILT: Samp: Count: MODE <char>

(Read-Write) Sets and returns the IF filter sample count mode to the specified value. When in MANUAL mode, the value specified for the IF Filter sample count is used as the number of taps in the IF filter.

For use with the E836xC Opt H11

Parameters

- <cnum> Existing channel number to manipulate. If unspecified, <cnum> is set to 1.
- <char> Choose either AUTO or MANUAL.

Examples

SENSe:IF:FILT:SAMP:COUN:MODE MANUAL
sense2:if:filter:sample:count:mode AUTO

Query Syntax
SENSe:IF:FILT: Samp: Count: MODE?

Return Type
Character

Default
AUTO

SENSe<cnum>:IF:FILT: Samp: PERiod <num>

(Read-Write) Sets and returns the IF filter sample period. The IF filter sample period setting is only used by the instrument when the SENS:IF:FILT: Samp: PER: MODE is set to MANUAL.

For use with the E836xC Opt H11

Parameters

- <cnum> Existing channel number to manipulate. If unspecified, <cnum> is set to 1.
- <num> Sample period. Choose from values returned from the SENS:IF:FILT:SAMP:PER: MODE?

Examples

SENSe:IF:FILT:SAMP:PER 6 us
sense2:if:filter:sample:period maximum

Query Syntax
SENSe:IF:FILT: Samp: PERiod?

Return Type
Numeric

Default
Instrument dependent.

SENSe<cnum>:IF:FILT: Samp: PERiod: CATalog?
(Read-Only) Returns the list of allowed IF filter sample periods for this instrument.

For use with the E836xC Opt H11

Parameters

- `<cnum>` Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.

Examples

```
SENS:IF:FILT:SAMP:PER:CAT?
sense2:if:filter:sample:period:catalog?
```

Query Syntax

SENS:IF:FILTER:SAMPLE:PERIOD:CATalog?

Return Type

String

Default

AUTO

---

SENSe<cnum>:IF:FILTer:SAMPle:PERiod:MODE <char>

(Read-Write) Sets and returns the IF filter sample period mode to the specified value.

For use with the E836xC Opt H11

Parameters

- `<cnum>` Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.
- `<char>` Sample period mode. Choose from AUTO or MANUAL

Examples

```
SENS:IF:FILT:SAMP:PER:MODE MANUAL
sense2:if:filter:sample:period:mode AUTO
```

Query Syntax

SENS:IF:FILTER:SAMPLE:PERIOD:MODE?

Return Type

Character

Default

AUTO

---

SENSe<cnum>:IF:GAIN:ALL[:STATe] <char>

(Write only) Sets the gain state for ALL receivers to Auto or Manual.

For use with the E836xC Opt H11

Parameters

- `<cnum>` Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.
- `<char>` Choose from AUTO or MANUAL

Examples

```
SENS:IF:GAIN:ALL AUTO
sense:if:gain:all:state manual
```

Query Syntax

Not Applicable

Return Type

Character
SENSe<cnum>:IF:GAIN:LEVEL <id>, <level>

(Read Write) Manually sets the gain level for the specified receiver. This setting can be made for E836x and selected N5230A models. Learn more.

Parameters

- `<cnum>`: Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.
- `<id>`: (String) Choose from: 'A', 'B', 'R1', 'R2'..
- `<level>`: Gain level. Choose from:
  0 - Lowest gain setting
  1
  2 - Highest gain setting

Examples

```
SENS:IF:GAIN:LEVEL 'A', 1
```

Query Syntax

```
SENSe<cnum>:IF:GAIN:LEVEL? <id>
```

Return Type

Numeric

Default

0

SENSe<cnum>:IF:GAIN[:STATe]? <id>

(Read only) Returns the gain state for the specified receiver. Use SENS:IF:GAIN:ALL to set the gain state for all receivers in the channel. This setting can be made for E836x and selected N5230A models.

Parameters

- `<cnum>`: Existing channel number to manipulate. If unspecified, `<cnum>` is set to 1.
- `<id>`: String. Choose from: 'A', 'B', 'R1', 'R2'.

Note: The A and R1 receivers are always switched together. B and R2 are also always switched together. For example, if you specify 'A', R1 will also be switched.

Examples

```
SENS:IF:GAIN? 'A'
```

Return Type

Character - Returns AUTO or MAN. See SENS:IF:GAIN:ALL.

Default

AUTO
SENSe<cnum>:IF:GATE[:STATe] <boolean>

(Read-Write) Sets or returns the IF gate state.
For use with the E836xC Opt H11

Parameters

- <cnum> Existing channel number to manipulate. If unspecified, <cnum> is set to 1.
- <boolean> Choose from ON or OFF

Examples

```
SENS:IF:GATE ON
sense2:if:gate:state 0
```

Query Syntax

SENSe:IF:GATE[:STATe]?

Return Type

Boolean

Default

---

SENSe<cnum>:IF:SOURce:PATH <id>, <char>

(Read Write) Sets the source path for the specified receiver. An error is returned if <id> is not valid, or if <char> is not valid for the specified <id>.
For use with the E836xC Opt H11

Parameters

- <cnum> Existing channel number to manipulate. If unspecified, <cnum> is set to 1.
- <id> Choose from: 'A', 'B', 'R1', 'R2'

  Note: The A and R1 receivers are always switched together. B and R2 are also always switched together. For example, if you specify "A", R1 will also be switched.

- <char> Choose from:
  - NORMal - the PNA decides the appropriate IF input paths.
  - EXTernal - always use the rear panel IF inputs.

Examples

```
SENS:IF:SOUR:PATH 'A', Ext
```

Query Syntax

SENSe<cnum>:IF:SOURce:PATH? <id>

Return Type

Character

Default

Normal

---

Last Modified:

2637
**Sense:IF Commands**

Controls the DSP filters for use with the PNA X.

See the [Sense:IF commands](#) for the E836xC models.

### SCPI Command Tree

- **SENSe:**
  - **IF:**
    - **FILTER:**
      - **AUTO**
      - **CMODE**
      - **ERRO**
      - **STAGe**
        - **COEFFicients**
        - **COUNT?**
        - **FREQUENCY**
        - **CATalog?**
        - **PARameter**
        - **TYPE**
        - **PCATalog?**
    - **FREQuency**
      - **AUTO**
      - **VALue**

- Click on a [blue](#) keyword to view the command details.
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

### PNA-X DSP Block diagram

All of the Sense:IF commands, except the frequency commands, make settings related to the DSP section of the IF Path.

See the entire [IF Path Configuration](#).
For any of the Filter "Stage" parameters to take effect, `SENS:IF:FILT:AUTO` must be set to OFF (MANUAL) and mode, and `SENS:IF:FILT:CMOD` must be set to OFF.

Stage2 settings are ignored when using DSP 5 versions.

Programs that control these settings, or state files that are saved, will yield different results when run or recalled on PNAs with DSP 4 versions versus DSP 5 versions. Learn more about DSP versions.

**Critical Note:** These commands act on the selected measurement. You can select one measurement for each channel using `Calc:Par:Select`.

### SENSE\<cnum\>:IF:FILTER:AUTO <bool>

(Read-Write) Sets and returns whether the PNA configures the 3-stage digital filter settings or they will be configured manually. When making manual settings, also send `SENS:IF:FILT:CMOD OFF` which routes the IF through the 3-stage filter.

**Critical Note**

**Parameters**
- `<cnum>`: Existing channel number. If unspecified, `<cnum>` is set to 1.
- `<bool>`: (Boolean)
  - **ON** (or 1) - Automatic: PNA controls digital filter settings.
  - **OFF** (or 0) - Manual: You control digital filter settings using other Sens:IF commands.

**Examples**
- `SENS:IF:FILT:AUTO 1`
- `sense2:if:filter:auto 0`

**Query Syntax**
- `SENSe\<cnum\>:IF:FILTER:AUTO?`

**Return Type**
- Boolean
- **Default**
  - ON

### SENSE:IF:FILTER:CMODE <bool>
(Read-Write) Sets and returns the ADC capture mode modeled as a 2-pole switch in the above diagram. The switch either bypasses or routes the IF through the 3-stage digital filter.

**Critical Note**

**Parameters**

<bool> (Boolean)

**ON** (or 1) - The digital filters are bypassed and the raw ADC readings are taken directly. With DSP 4 versions, a maximum of 4096 data points per sweep can be acquired.

With DSP 5 versions, the PNA maximum data points per sweep can be acquired. Learn more about DSP Versions.

**OFF** (or 0) - The digital filters are used to process IF information. The filters can be configured automatically or manually using SENS:IF:FILT:AUTO.

**Examples**

SENS:IF:FILT:CMOD 1
sense2:if:filter:cmode 0

**Query Syntax**

SENSe<cnum>:IF:FILT:ERRors?

**Return Type**

Boolean

**Default**

OFF

---

**SENS<cnum>:IF:FILT:ERRors?**

(Read-only) Returns the error string associated with the digital filters. The return string has three fields separated by commas: "stage1 status, stage2 status, stage3 status"

Each of these fields can contain one or more of the following error codes:

- **NO ERROR**
- **NUMBER-OF-COEFFICIENTS** - the number of coefficients is excessive for that filter-stage
- **COEFFICIENT VALUE** - one or more coefficients are out of range for that filter-stage
- **SUM-OF-COEFFICIENTS** - the sum of all coefficients is excessive for that filter-stage,
- **FREQUENCY** - the frequency for Stage 1 is out of range (only applies stage1 field),
- **PARAMETER** - one or more parameters are out of range (only applies to stage 3 field)

---

2641
SENSe<cn>u>m>:IF:FiLT:STAGE<n>:COEFFicients <coef>

(Read-Write) Sets and returns the digital filter coefficients of the specified stage.

**Note:** Stage2 settings are ignored when using DSP Version 5. [Learn more.]

### Critical Note

#### Parameters

- `<cn>` Existing channel number. If unspecified, `<cn>` is set to 1.
- `<n>` Stage number. Choose 1 or 2
- `<coef>` An array of real numbers. Filter coefficients

#### Examples

```
SENS:IF:FILT:STAG2:COEF 0,0.1,0.7,0.7,0.1
```

#### Query Syntax

```
SENSe<cn>:IF:FILTer:STAGE<n>:COEFFicients?
```

#### Example

```
SENS:IF:FILT:STAG2:COEF?
```

#### Return Type

**Default** Stage dependent

---

**SENSe<cn>:IF:FILTer:STAGE<n>:COUNt? [char]**
(Read-only) Returns the number of taps in the digital filter of the specified stage. The filter sample count setting is only used when SENSE:IF:FILTer:AUTO is set to False (MANUAL).

Note: Stage2 settings are ignored when using DSP Version 5. Learn more.

Critical Note

Parameters

- `<cnum>`: Existing channel number. If unspecified, `<cnum>` is set to 1.
- `<n>`: Stage number. Choose 1 or 2
- `[char]`:
  - Optional parameter. Choose from:
    - `not specified`: returns the current number of coefficients for the specified stage.
    - `MIN`: returns the minimum number of coefficients for the specified stage. Stage1: 10, Stage2: 1
    - `MAX`: returns the maximum number of coefficients for the specified stage. Stage1 & 2: 1024

Examples

```
SENS:IF:FILT:STAG2:COUN?
sense2:if:filter:stage1:count? max
```

Return Type

- Numeric
- Default: Stage dependent

SENSe<cnum>:IF:FILTer:STAGE1:FREQuency <value>

(Read-Write) Sets and returns the Numerically Controlled Oscillator (NCO) frequency. This command is only used when SENSE:IF:FILTer:AUTO is set to False (Manual).

Critical Note

Parameters

- `<cnum>`: Existing channel number. If unspecified, `<cnum>` is set to 1.
- `<value>`: Stage 1 Frequency. Min value= 0 Hz
  - With DSP 4 versions, Max value= 15 MHz.
  - With DSP 5 versions, Max value = 38 MHz.
  - Or programmatically use the Max and Min queries to determine the range of settable values.

Learn more about DSP versions.
Example: SENS:IF:FILT:STAGe1:FREQ 9e6
sense2:if:filter:stage1:frequency 9.2e6

Query Syntax
SENS<cnm>:IF:FILTer:STAGE1:FREQuency?
'returns the current parameter value
SENS<cnm>:IF:FILTer:STAGE1:FREQuency? Min
'returns the minimum frequency value.
SENS<cnm>:IF:FILTer:STAGE1:FREQuency? Max
'returns the maximum frequency value.

Return Type
Numeric
Default: Nominal IF Frequency. Learn more

SENS<cnm>:IF:FILTer:STAGE3:CATalog?
(Read-only) Returns a list of strings for the currently supported filter types that can be used for the stage 3 filter. This command is only used when SENS:IF:FILTer:AUTO is set to False (Manual). See SENS:IF:FILT:STAGe3:TYPE for a list of currently supported filter types.

Parameters
<cnm> Existing channel number. If unspecified, <cnm> is set to 1.

Examples
SENS:IF:FILT:STAGE3:CAT?
sense2:if:filter:stage3:catalog?

Return Type
String array
Default: Not Applicable

SENS<cnm>:IF:FILTer:STAGE3:TYPE <value>
(Read-Write) Sets and returns the Stage 3 filter type. This command is only used when `SENSe:IF:FILTER:AUTO` is set to False (Manual).

**Critical Note**

**Parameters**

```
<cnum>
```
Existing channel number. If unspecified, `<cnum>` is set to 1.

```
<value>
```
(String) Filter type. Choose from:

- "RECT" Rectangular Window Filter
- "TUKEY" Tukey Filter
- "PWIN" Pulse Window Filter

**Examples**

```
SENS:IF:FILT:STAGE3:TYPE RECT
```

```
sense2:if:filter:stage3:type pwin
```

**Query Syntax**

```
SENSe<cnum>:IF:FILTer:STAGE3:TYPE?
```

**Return Type**

String

**Default**

"TUKEY"

---

`SENSe<cnum>:IF:FILTER:STAGE3:PARameter <p>, <value>`

(Read-Write) Sets and returns the Stage 3 filter parameters.

Must first select the filter type (`SENS:IF:FILT:STAGE3:TYPE`) before setting these parameters.

Use `SENSe:IF:FILT:STAGE3:PCAT?` to return a list of the available parameters for the currently selected filter type.

**Critical Note**

**Parameters**

```
<cnum>
```
Existing channel number. If unspecified, `<cnum>` is set to 1.

```
<p>
```
(String) Filter parameter. Choose from:

- "C" - Tap count (Tukey, RECT, PWIN)
- "P" - Period (PWIN ONLY)
- "D" - Delay (PWIN ONLY)
- "W" - Width (PWIN ONLY)
- "R" - Ramp Count (PWIN ONLY)
<value> (String) Parameter Value for the specified stage 3 parameter. Use the query form to return the minimum and maximum values for the specified parameter.

**Examples**

```
SENS:IF:FILT:STAGE3:PAR "C", 64
sense2:if:filter:stage3:parameter "d", 0.5E-6
```

**Query Syntax**


- returns the current parameter value
- `SENS<cnm>:IF:FILTer:STAGE3:PARameter? <p>,Min`
  - returns the minimum parameter value.
- `SENS<cnm>:IF:FILTer:STAGE3:PARameter? <p>,Max`
  - returns the maximum parameter value.

**Examples**

```
SENS:IF:FILT:STAGE3:PAR? "C"
sense2:if:filter:stage3:parameter? "d", min
```

**Return Type**

- Numeric

**Default**

- `RECT: C = 1`
- `PWIN: C=1E6, P=10ms, D=50us, W=50us, R=7`
- `TUKEY: C=1`

---

**SENS<cnm>:IF:FILTer:STAGE3:PCATalog?**

*(Read-only) Returns a list of the available parameters for the currently selected filter type.*

**Critical Note**

- **Parameters**
  - `<cnm>` Existing channel number. If unspecified, `<cnm>` is set to 1.

**Examples**

```
SENS:IF:FILT:STAGE3:PCAT?
sense2:if:filter:stage3:pcatalog?
```

**Return Type**

- String

**Default**

- Not Applicable

---

**SENS<cnm>:IF:FREQuency:AUTO <bool>**
(Read-Write) Sets and returns the method for specifying the way the IF Frequency is determined.

**Critical Note**

**Parameters**

- <cnum>: Existing channel number. If unspecified, <cnum> is set to 1.
- <bool>: (Boolean)
  - **ON** (or 1) - Automatic. PNA determines the setting for the IF frequency. The IF frequency is based on many PNA settings, including measurement frequency. Therefore, it is NOT possible to read the IF frequency that is being used.
  - **OFF** (or 0) - Manual. Use SENS:IF:FREQ to set the frequency.

**Examples**

```
SENS:IF:FREQ:AUTO 1
sense2:if:frequency:auto 0
```

**Query Syntax**

SENSe<cnum>:IF:FREQuency:AUTO?

**Return Type**

Boolean

**Default**

ON

---

**SENSe<cnum>:IF:FREQuency[:VALue] <value>**

(Read-Write) Sets and returns the IF frequency for ALL receiver paths being used for the specified channel. To set this frequency, SENS:IF:FREQ:AUTO must be set to OFF (Manual). Also returns the maximum and minimum allowable frequency settings. Learn more.

**Critical Note**

**Parameters**

- <cnum>: Existing channel number. If unspecified, <cnum> is set to 1.
- <value>: (Numeric) Frequency value. Use the Max and Min Queries to determine the range of this setting. (SENS:IF:FREQ? Max)

**Examples**

```
SENS:IF:FREQ 9.1e6
sense2:if:frequency:value 8.9e6
```

**Query Syntax**

SENSe<cnum>:IF:FREQuency?

- 'returns the current frequency setting
- SENS:IF:FREQ? Max

- 'returns the maximum allowable frequency setting
- SENS:IF:FREQuency? Min

- 'returns the minimum allowable frequency setting
Return Type  Numeric
Default  9 MHz

Last Modified:

26-May-2011  Changed IF freq default
26-Aug-2010   Updated for DSP 5
18-Jan-2007   MX New topic
# Sense:IMD Commands

Controls an IMD or IMDx measurement configuration.

<table>
<thead>
<tr>
<th>SENSE:IMD</th>
</tr>
</thead>
<tbody>
<tr>
<td>SWEep:TYPE</td>
</tr>
<tr>
<td>CSO:</td>
</tr>
<tr>
<td>NDPRoducts</td>
</tr>
<tr>
<td>NORMalized:POWer</td>
</tr>
<tr>
<td>OFFSet</td>
</tr>
<tr>
<td>CTB</td>
</tr>
<tr>
<td>NCARriers</td>
</tr>
<tr>
<td>NORMalized:POWer</td>
</tr>
<tr>
<td>OFFSet</td>
</tr>
<tr>
<td>FREQuency</td>
</tr>
<tr>
<td>DFRequency</td>
</tr>
<tr>
<td>[:CW]</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
<tr>
<td>F1[:CW]</td>
</tr>
<tr>
<td>F2[:CW]</td>
</tr>
<tr>
<td>FCENter</td>
</tr>
<tr>
<td>[:CW]</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
<tr>
<td>CENTER</td>
</tr>
<tr>
<td>SPAN</td>
</tr>
<tr>
<td>HOPRoduct?</td>
</tr>
<tr>
<td>IFBWidth</td>
</tr>
</tbody>
</table>
Click on a blue keyword to view the command details.

**Other Swept IMD commands**

- **CALC:CUSTom:DEFine** - creates a gain compression measurement.
- **Swept IMD Calibration** - these are supplemental to the Guided Cal commands

See Also

- **Example** Create and Cal an IMD Measurement
- **IM Spectrum commands**
- **Learn about Swept IMD**
- **Learn about Measurement Class**
- **Synchronizing the PNA and Controller**
SENSe<chn>:IMD:SWEep:TYPE <char>

(Read-Write) Sets and returns the sweep type for the IMD measurement.

**Parameters**

- `<chn>`  Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<char>`  Sweep type. Choose from:

  - **FCENTer** - (Center Frequency) Maintaining a constant tone spacing (DeltaF) and tone powers (P1 and P2), the center frequency (FC) is swept from Start to Stop.
  - **DFRequency** - (Delta Frequency) The center frequency (FC) is held constant. The tone spacing is increased from StartDeltaF to StopDeltaF.
  - **POWer** - The main tone frequencies are specified as either F1 and F2, or as FC and DeltaF. These frequencies are held constant while the power of each tone is stepped from the Start Power to the Stop Power.
  - **CW** - The main tone frequencies (F1 and F2) and power levels (P1 and P2) are held constant. Measurements are taken for the specified Number of Points.
  - **SEGment** - Not available for IMDx. Same as FCenter sweep, except that the center frequencies for the sweep are constructed using the standard **segment sweep commands**.
  - **LOPower** - All frequencies are fixed while the LO power is swept from Start to Stop power.

For each sweep type, use the commands that follow:

**FCENTer:**

- **SENS:IMD:FREQ:FCEN:STAR**
- **SENS:IMD:FREQ:FCEN:STOP**
- **SENS:IMD:FREQ:FCEN:CEN**
- **SENS:IMD:FREQ:FCEN:SPAN**
- **SENS:IMD:FREQ:DFR:[CW]**
- **SENS:IMD:TPOW:F1**
- **SENS:IMD:TPOW:F2**
DFRequency

- SENS:IMD:FREQ:DFR:STAR
- SENS:IMD:FREQ:DFR:STOP
- SENS:IMD:FREQ:FCEN:[CW]
- SENS:IMD:TPOW:F1
- SENS:IMD:TPOW:F2

POWer

- SENS:IMD:FREQ:F1:[CW]
- SENS:IMD:FREQ:F2:[CW]
- SENS:IMD:FREQ:FCEN:[CW]
- SENS:IMD:FREQ:DFR:[CW]
- SENS:IMD:TPOW:F1:STAR
- SENS:IMD:TPOW:F1:STOP
- SENS:IMD:TPOW:F2:STAR
- SENS:IMD:TPOW:F2:STOP

CW

- SENS:IMD:FREQ:F1:[CW]
- SENS:IMD:FREQ:F2:[CW]
- SENS:IMD:FREQ:FCEN:[CW]
- SENS:IMD:FREQ:DFR:[CW]
- SENS:IMD:TPOW:F1
- SENS:IMD:TPOW:F2

SEGMen - Not available for IMDx.

- SENS:IMD:FREQ:DFR:[CW]
- SENS:IMD:TPOW:F1
- **SENS:IMD:TPOW:F2**

  Use the standard *segment sweep commands* to set freq start and stop

**LOPower - IMDx ONLY**

- **SENS:MIX:LO:POW:STAR**
- **SENS:MIX:LO:POW:STOP**
- **SENS:IMD:FREQ:F1:[CW]**
- **SENS:IMD:FREQ:F2:[CW]**
- **SENS:IMD:FREQ:FCEN:[CW]**
- **SENS:IMD:FREQ:DFR:[CW]**
- **SENS:IMD:TPOW:F1**
- **SENS:IMD:TPOW:F2**

**Examples**

<table>
<thead>
<tr>
<th>SENSE IMD SWEep TYPE CW</th>
<th>sense2 imd sweep type power</th>
</tr>
</thead>
</table>

**Query Syntax**

SENSe<cnum>:IMD:SWEep:TYPE?

**Return Type** Character

**Default** FCENTer

---

**SENSe<cnum>:IMD:CSO:NDPRoducts <num>**

*(Read-Write)* Sets and returns the “N = number of distortion products” value for the calculation of the CSO parameter.

**Learn more.**

**Parameters**

- **<cnum>** Channel number of the IMD measurement. If unspecified, value is set to 1.
- **<num>** Number of distortion products.

**Examples**

| SENS IMD CSO NDPR 30 | sense2 imd cso ndproducts 7 |

**Query Syntax**

SENSe<cnum>:IMD:CSO:NDPRoducts?

**Return Type** Numeric

**Default** 40
SENSe<nump>:IMD:CSO:NORMalized:POWer <num>

(Read-Write) Sets and returns the CSO Power for POWER normalization mode. Valid only with measurement parameters: CSO2Lo and CSO2Hi and for Normalization Modes dBm and dBmV.  Learn more.

**Parameters**

- `<nump>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Power level. The units are determined by Sens:IMD:Norm:Mode, which must be set first.

**Examples**

SENSe<ch>:IMD:CSO:NORM:POW 0

sense2:imd:cso:normalized:power -5

**Query Syntax**

SENSe<nump>:IMD:CSO:NORMalized:POWer?

**Return Type**

Numeric

**Default**

0

---

SENSe<nump>:IMD:CSO:OFFSet <num>

(Read-Write) Sets and returns the offset that is applied to CSO measurements. Valid only with measurement parameters: CSO2Lo and CSO2Hi.  Learn more.

**Parameters**

- `<nump>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Offset value in dBm

**Examples**

SENSe<ch>:IMD:CSO:OFFS 3

sense2:imd:cso:offset 7

**Query Syntax**

SENSe<nump>:IMD:CSO:OFFSet?

**Return Type**

Numeric

**Default**

0

---

SENSe<nump>:IMD:CTB:NCARriers <num>
(Read-Write) Sets and returns the “N = Total number of carriers” value used in the calculation of the XMOD parameter. Learn more.

Parameters
<cnm> Channel number of the IMD measurement. If unspecified, value is set to 1.
<num> Number of carriers.

Examples
SENS:IMD:CTB:NCAR 10
sense2:imd:ctb:ncarriers 50

Query Syntax
SENS<cnm>:IMD:CTB:NCARriers?

Return Type
Numeric

Default
40

SENSe<cnm>:IMD:CTB:NORMalized:POWer <num>
(Read-Write) Sets and returns the CTB Power. Valid only with measurement parameters: CTBLo and CTBHi and for Normalization Modes dBm and dBmV. Learn more.

Parameters
<cnm> Channel number of the IMD measurement. If unspecified, value is set to 1.
<num> Power level. The units are determined by Sens:IMD:Norm:Mode, which must be set first.

Examples
SENS:IMD:CTB:NORM:POW 0
sense2:imd:ctb:norm:power -5

Query Syntax
SENS<cnm>:IMD:CTB:NORMalized:POWer?

Return Type
Numeric

Default
0

SENSe<cnm>:IMD:CTB:OFFSet <num>
(Read-Write) Sets and returns the offset that is applied to CTB measurements. Valid only with measurement parameters: CTB, CTBLo, CTBH, CTBE, CTBEL, and CTBEH. Learn more.

Parameters
- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Offset value in dBm

Examples
```
SENS:IMD:CTB:OFFS 3
sense2:imd:ctb:offset 7
```

Query Syntax
SENSe<cnum>:IMD:CTB:OFFSet?

Return Type
Numeric

Default
0

---

SENSe<cnum>:IMD:FREQuency:DFRequency[:CW] <num>
(Read-Write) Sets and returns fixed tone spacing.

Parameters
- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Tone spacing frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

Examples
```
SENS:IMD:FREQ:DFR 1e6
sense2:imd:frequency:dfrequency:cw 2e7
```

Query Syntax
SENSe<cnum>:IMD:FREQuency:DFRequency[:CW]?

Return Type
Numeric

Default
1 MHz

---

SENSe<cnum>:IMD:FREQuency:DFRequency:STARt <num>
(Read-Write) Sets and returns the starting main tone separation for sweep type = DFRequency (delta frequency).

**Parameters**

- **<cnum>** Channel number of the IMD measurement. If unspecified, value is set to 1.
- **<num>** Starting tone separation between F1 and F2 in Hz. Both F1 and F2 tones MUST be within the frequency range of the PNA where:

  \[
  \begin{align*}
  F1 \text{ (start)} &= \text{FREQ:FCEN} - \text{DFR:Start} / 2 \\
  F2 \text{ (start)} &= \text{FREQ:FCEN} + \text{DFR:Start} / 2
  \end{align*}
  \]

**Examples**

```plaintext
SENS:IMD:FREQuency:DFRequency:START 1e6
sense2:imd:frequency:dfrequency:start 2e7
```

**Query Syntax**  
SENS<cnun>:IMD:FREQuency:DFRequency:START?

**Return Type**  
Numeric

**Default**  
1 MHz

---

**SENS<cnun>:IMD:FREQuency:DFRequency:STOP <num>**

(Read-Write) Sets and returns the stopping main tone separation for sweep type = DFRequency (delta frequency).

**Parameters**

- **<cnum>** Channel number of the IMD measurement. If unspecified, value is set to 1.
- **<num>** Stopping tone separation between F1 and F2 in Hz. Both F1 and F2 tones MUST be within the frequency range of the PNA where:

  \[
  \begin{align*}
  F1 \text{ (stop)} &= \text{FREQ:FCEN} - \text{DFR:Stop} / 2 \\
  F2 \text{ (stop)} &= \text{FREQ:FCEN} + \text{DFR:Stop} / 2
  \end{align*}
  \]

**Examples**

```plaintext
SENS:IMD:FREQuency:DFRequency:STOP 1e6
sense2:imd:frequency:dfrequency:stop 2e7
```

**Query Syntax**  
SENS<cnun>:IMD:FREQuency:DFRequency:STOP?

**Return Type**  
Numeric

**Default**  
10 MHz

---

**SENS<cnun>:IMD:FREQuency:F1[:CW] <num>**
(Read-Write) Sets and returns the frequency of the F1 tone.

Parameters

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` F1 tone frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

Examples

```
SENS:IMD:FREQ:F1 1e9
sense2:imd:frequency:F1:cw 2e7
```

Query Syntax

```
SENSe<cnum>:IMD:FREQuency:F1[:CW]?
```

Return Type

Numeric

Default

.9995 GHz

(Read-Write) Sets and returns the frequency of the F2 tone.

Parameters

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` F2 tone frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

Examples

```
SENS:IMD:FREQ:F2 1e9
sense2:imd:frequency:F2:cw 2e7
```

Query Syntax

```
SENSe<cnum>:IMD:FREQuency:F2[:CW]?
```

Return Type

Numeric

Default

1.0005 GHz

SENSe<cnum>:IMD:FREQuency:FCENter[:CW] <num>
(Read-Write) Sets and returns the center frequency of the main tones.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Tone center frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMD:FREQ:FCEN 1e9
sense2:imd:frequency:fcenter:cw 2e7
```

**Query Syntax**

```
SENSe<cnum>:IMD:FREQuency:FCENter[:CW]?
```

**Return Type** Numeric

**Default** 1.0 GHz

---

**SENSe<cnum>:IMD:FREQuency:FCENter:CENTer <num>**

(Read-Write) Sets and returns the sweep center frequency when sweeping the main tones.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Center frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMD:FREQ:FCEN:CENT 1e9
sense2:imd:frequency:fcenter:cent 2e7
```

**Query Syntax**

```
SENSe<cnum>:IMD:FREQuency:FCENter:CENTer?
```

**Return Type** Numeric

**Default** 13.255 GHz

---

**SENSe<cnum>:IMD:FREQuency:FCENter:SPAN <num>**
(Read-Write) Sets and returns the frequency span when sweeping the main tones.

**Parameters**

- `<cnum>`: Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>`: Frequency span in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMD:FREQ:FCEN:SPAN 1e9
sense2:imd:frequency:fcenter:span 2e7
```

**Query Syntax**

SENSe<cnum>:IMD:FREQuency:FCENter:SPAN?

**Return Type**

Numeric

**Default**

26.489 GHz

---

SENSe<cnum>:IMD:FREQuency:FCENter:STARt <num>

(Read-Write) Sets and returns the start frequency when sweeping the main tones.

**Parameters**

- `<cnum>`: Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>`: Start frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMD:FREQ:FCEN:STAR 1e9
sense2:imd:frequency:fcenter:start 2e7
```

**Query Syntax**

SENSe<cnum>:IMD:FREQuency:FCENter:STARt?

**Return Type**

Numeric

**Default**

10.5 MHz

---

SENSe<cnum>:IMD:FREQuency:FCENter:STOP <num>

2660
(Read-Write) Sets and returns the stop frequency when sweeping the main tones.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Stop frequency in Hz. The F1 and F2 tones MUST be within the frequency range of the PNA.

**Examples**

```plaintext
SENS:IMD:FREQ:FCEN:STOP 1e9
sense2:imd:frequency:fcenter:stop 2e9
```

**Query Syntax**

SENSe<cnum>:IMD:FREQuency:FCENter:STOP?

**Return Type**

Numeric

**Default**

26.4995 GHz

---

**SENSe:IMD:HOPRooduct?**

(Read-only) Returns the highest order product that can be measured by SweptIMD.

**Parameters**

None

**Examples**

```plaintext
SENS:IMD:HOPR?
```

'always returns 9

**Return Type**

Numeric

**Default**

9

---

**SENSe<cnum>:IMD:IFBWidth:MAIN <num>**

(Read-Write) Sets and returns the IF Bandwidth for measurement of the main F1 and F2 tones. Learn more about setting IFBW for IMD.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Choose from: 1 | 2 | 3 | 5 | 7 | 10 | 15 | 20 | 30 | 50 | 70 | 100 | 150 | 200 | 300 | 500 | 700 | 1k | 1.5k | 2k | 3k | 5k | 7k | 10k | 15k | 20k | 30k | 50k | 70k | 100k | 150k | 200k | 280k | 360k | 600k

If an invalid number is specified, the analyzer will round up to the closest valid number.
**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:IMD:IFBWidth:MAIN?</td>
<td>Numeric</td>
<td>1 kHz</td>
</tr>
</tbody>
</table>

**SENSe<cnum>:IMD:IFBWidth:IMTone <num>**

*(Read-Write)* Sets and returns the IF Bandwidth for measurement of the intermodulation products. Learn more about setting IFBW for IMD.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Choose from: 1 | 2 | 3 | 5 | 7 | 10 | 15 | 20 | 30 | 50 | 70 | 100 | 150 | 200 | 300 | 500 | 700 | 1k | 1.5k | 2k | 3k | 5k | 7k | 10k | 15k | 20k | 30k | 50k | 70k | 100k | 150k | 200k | 280k | 360k | 600k

If an invalid number is specified, the analyzer will round up to the closest valid number.

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
<th>Return Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:IMD:IFBWidth:IMT 50</td>
<td>Numeric</td>
<td>1 kHz</td>
</tr>
<tr>
<td>sense2:imd:ifbwidth:imtone 200</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**SENSe<cnum>:IMD:NORMalized:MODE <char>**
(Read-Write) Sets and returns the method by which CTB and CSO calculations are performed.

**Parameters**

<cnun> Channel number of the IMD measurement. If unspecified, value is set to 1.
<char> Normalization mode. Choose from:

**NONE** - the normalized power is not used in calculation

**NCARrier** - CTB and CSO is corrected by subtracting 10*log(N/2), where

- N = # of carriers for CTB
- N = # of distortion products for CSO

**DBM** - the composited normalized power for CTB or CSO is treated as a dBm value

**DBMV** - the composited normalized power for CTB or CSO is treated as a dBmV value.

**Note:** Power values are stored using the currently-set units. Therefore, first set units with this command, then set power values using:


**Examples**

```
SENS:IMD:NORM:MODE NCAR
sense2:imd:normalized:mode none
```

**Query Syntax**

`SENSe<cnun>:IMD:NORMALized:MODE?`

**Return Type**

Character

**Default**

NCARrier

**SENSe<cnun>:IMD:PMAP <input>,<output>**
(Write-only) Sets the input port and output port of an IMD or IMDx channel. This setting is necessary only when using the limited port mapping feature. Learn more.

Parameters

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<input>` PNA port connected to the DUT Input. Choose from 1 or 3. When input is 3, an external combiner must be used.
- `<output>` PNA port connected to the DUT Output.
  
  When input is 1, output must be 2.

  When input is 3, output must be 4.

Examples

```
SENS:IMD:PMAP 3,4
sense2:imd:pmap 3,4
```

Query Syntax

Not Applicable

Default

1,2

SENS<cnm>:IMD:PMAP:INPut?

(Read-only) Returns the PNA test port to be connected to the DUT input for an IMD or IMDx channel. Set the PNA port to DUT mapping using SENS:IMD:PMAP

Parameters

- `<cnum>` IMD channel number. If unspecified, value is set to 1.

Examples

```
SENS:IMD:PMAP:INP?
sense2:imd:pmap:input?
```

Default

1

SENS<cnm>:IMD:PMAP:OUTPut?
(Read-only) Returns the PNA test port to be connected to the DUT output for an IMD or IMDx channel. Set the PNA port to DUT mapping using SENS:IMD:PMAP

**Parameters**

- <cnum> IMD channel number. If unspecified, value is set to 1.

**Examples**

```plaintext
SENS:IMD:PMAP:OUTP?
sense2:imd:pmap:output?
```

**Default** 2

**SENSe<cnum>:IMD:TPOWer:COUPle[:STATe] <bool>**

(Read-Write) Sets and returns the state of power coupling for F1 and F2. Learn more about tone power.

**Parameters**

- <cnum> Channel number of the IMD measurement. If unspecified, value is set to 1.
- <bool> Tone power level coupling state. Choose from:
  - **ON (or 1)** - F1 and F2 power is coupled.
  - **OFF (or 0)** - F1 and F2 power is NOT coupled. Set power levels individually.

**Examples**

```plaintext
SENS:IMD:TPOW:COUP 0
sense2:imd:tpower:couple:state ON
```

**Query Syntax**

SENSe<cnum>:IMD:TPOWer:COUPle[:STATe]?

**Return Type** Boolean

**Default** ON

**SENSe<cnum>:IMD:TPOWer:EQUalize:STATe <bool>**
(Read-Write) Sets and returns the state of Equal Tone Power setting. Learn more about tone power.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<bool>` Equal Tone Power state. Choose from:
  - **ON (or 1)** - Equalize f1 and f2 power.
  - **OFF (or 0)** - Do NOT equalize f1 and f2 power. Use source power cal.

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:IMD:TPOWer:EQUalize:STATe?</td>
</tr>
</tbody>
</table>

**Return Type** Boolean

**Default** OFF

**SENSe<cnum>:IMD:TPOWer:F1 <num>**

(Read-Write) Sets and returns the power level of the F1 tone. When SENSe:IMD:TPOW:COUP ON (tone power is coupled), setting either F1 or F2 power sets both. Learn more about tone power.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Tone power level in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**

<table>
<thead>
<tr>
<th>Query Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe&lt;cnum&gt;:IMD:TPOWer:F1 0</td>
</tr>
<tr>
<td>sense2:imd:tpower:F1 -10</td>
</tr>
</tbody>
</table>

**Query Syntax** SENSe<cnum>:IMD:TPOWer:F1?

**Return Type** Numeric

**Default** -20 dBm

**SENSe<cnum>:IMD:TPOWer:F2 <num>**
(Read-Write) Sets and returns the power level of the F2 tone. When SENS:IMD:TPOW:COUP ON (tone power is coupled), setting either F1 or F2 power sets both. Learn more about tone power.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Tone power level in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**

<table>
<thead>
<tr>
<th>Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:IMD:TPOW:F2 0</td>
</tr>
<tr>
<td>sense2:imd:tpower:F2 -10</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENS<cnum>:IMD:TPOWER:F2?

**Return Type**

Numeric

**Default**

-20 dBm

SENSe<cnum>:IMD:TPOWer:F1:STARt `<num>`

(Read-Write) Sets and returns the start power level of the F1 tone. Learn more about tone power.

**Parameters**

- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Start power in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**

<table>
<thead>
<tr>
<th>Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:IMD:TPOW:F1:STAR 0</td>
</tr>
<tr>
<td>sense2:imd:tpower:F1:start -10</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENS<cnum>:IMD:TPOWER:F1:STARt?

**Return Type**

Numeric

**Default**

-20 dBm

SENSe<cnum>:IMD:TPOWer:F1:STOP `<num>`
(Read-Write) Sets and returns the stop power level of the F1 tone. Learn more about tone power.

**Parameters**
- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Stop power in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**
```
SENS:IMD:TPOW:F1:STOP 0
sense2:imd:tpower:F1:stop 10
```

**Query Syntax**
SENSe<cnum>:IMD:TPOWer:F1:STOP?

**Return Type**
Numeric

**Default**
-10 dBm

---

**SENSe<cnum>:IMD:TPOWer:F2:STARt <num>**
(Read-Write) Sets and returns the start power level of the F2 tone. Learn more about tone power.

**Parameters**
- `<cnum>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Start power in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**
```
SENS:IMD:TPOW:F2:STAR 0
sense2:imd:tpower:F2:start -10
```

**Query Syntax**
SENSe<cnum>:IMD:TPOWer:F2:STARt?

**Return Type**
Numeric

**Default**
-20 dBm

---

**SENSe<cnum>:IMD:TPOWer:F2:STOP <num>**
**SENS<cnm>:IMD:TPOWer:F2:STOP**

(Read-Write) Sets and returns the stop power level of the F2 tone.  Learn more about tone power.

### Parameters

- `<cnm>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<num>` Stop power in dBm. Choose a value between +30 dBm and -30 dBm.

### Examples

```
SENS:IMD:TPOW:F2:STOP 0
sense2:imd:tpower:F2:stop 10
```

### Query Syntax

`SENS<cnm>:IMD:TPOWer:F2:STOP?`

### Return Type

Numeric

### Default

-10 dBm

---

**SENS<cnm>:IMD:TPOWer:SET <char>**

(Read-Write) Sets and returns whether tone power is specified at the DUT input or output.

### Parameters

- `<cnm>` Channel number of the IMD measurement. If unspecified, value is set to 1.
- `<char>` Choose from:

  - **INPUT** - Specified power level is set at the DUT input.
  - **OUTPUT** - Specified power level is set at the DUT output.

### Examples

```
SENS:IMD:TPOW:SET INPUT
sense2:imd:tpower:SET output
```

### Query Syntax

`SENS<cnm>:IMD:TPOWer:SET?`

### Return Type

Character

### Default

INPUT

---

Last Modified:

- 5-May-2011  Added tone power settings
- 11-Aug-2009 Added PMAP
- 13-Mar-2009 Edited for IMDx
- 15-Aug-2008 MX New topic
Sense:IMS Commands

Controls IM Spectrum measurement configuration.

SENSe:IMS

PMAP

| INPut
| OUTPut

RBW

RESPonse

| STARt
| STOP
| CENTER
| SPAN

STIMulus

| DFRequency
| FCENter
| F1FRequency
| F2FRequency
| TPOWer

| F1
| F2

SWEep:

| TYPE
| ORDer

TPOWer

| COUPle[:STATe]
| EQUal
| F1
SENSe<cnun>:IMS:PMAP <input>,<output>

(Write-only) Sets the input port and output port of an IMS or IMSx channel. This setting is necessary only when using the **limited port mapping feature**. [Learn more.](#)

**Parameters**

- **<cnun>** Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- **<input>** PNA port connected to the DUT Input. Choose from 1 or 3. When input is 3, an external combiner must be used.
- **<output>** PNA port connected to the DUT Output.
  
  When input is 1, output must be 2.
  
  When input is 3, output must be 4.

**Examples**

SENSE:IMS:PMAP 3,4

sense2:ims:pmap 3,4
**SENSe<cnum>:IMS:PMAP:INPut?**

*(Read-only)* Returns the PNA test port to be connected to the DUT input for an IMS or IMSx channel. Set the PNA port to DUT mapping using **SENSe:IMS:PMAP**

**Parameters**

- `<cnum>`: IMS channel number. If unspecified, value is set to 1.

**Examples**

```
SENSe:IMS:PMAP:INP?
sense2:ims:pmap:input?
```

**Default** 1

**SENSe<cnum>:IMS:PMAP:OUTPut?**

*(Read-only)* Returns the PNA test port to be connected to the DUT output for an IMS or IMSx channel. Set the PNA port to DUT mapping using **SENSe:IMS:PMAP**

**Parameters**

- `<cnum>`: IMS channel number. If unspecified, value is set to 1.

**Examples**

```
SENSe:IMS:PMAP:OUTP?
sense2:ims:pmap:output?
```

**Default** 2

**SENSe<cnum>:IMS:RBW <num>**
**(Read-Write)** Sets and returns the Resolution Bandwidth for the IM Spectrum measurement.

**Parameters**

- `<cnum>`: Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>`: Resolution BW in Hz. Choose from: 60k | 100k | 150k | 300k, 600k | 1.0M | 3.0M

  If an invalid number is specified, the PNA will round up to the closest valid number.

**Examples**

```
SENS:IMS:RBW 600e3
sense2:ims:rbw 1MHz
```

**Query Syntax**

```
SENSe<cnum>:IMS:RBW?
```

**Return Type**

Numeric

**Default**

600 kHz

---

**SENSe<cnum>:IMS:RESPonse:STARt <num>**

**(Read-Write)** Sets and returns the receiver Start frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

**Parameters**

- `<cnum>`: Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>`: Start frequency in Hz. Choose a frequency within the range of the PNA.

**Examples**

```
SENS:IMS:RESP:STAR 1e9
sense2:ims:response:start 100e6
```

**Query Syntax**

```
SENSe<cnum>:IMS:RESPonse:STARt?
```

**Return Type**

Numeric

**Default**

950 MHz

---

**SENSe<cnum>:IMS:RESPonse:STOP <num>**
(Read-Write) Sets and returns the receiver Stop frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

**Parameters**
- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>` Stop frequency in Hz. Choose a frequency within the range of the PNA.

**Examples**
```
SENS:IMS:RESP:STOP 26e9
sense2:ims:response:stop 100e6
```

**Query Syntax**
```
SENSe<cnum>:IMS:RESPonse:STOP?
```

**Return Type** Numeric

**Default** 1.05 MHz

---

**SENSe<cnum>:IMS:RESPonse:CENTer <num>**
(Read-Write) Sets and returns the receiver Center frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

**Parameters**
- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>` Center frequency in Hz. Choose a frequency within the range of the PNA.

**Examples**
```
SENS:IMS:RESP:CENT 26e9
sense2:ims:response:center 100e6
```

**Query Syntax**
```
SENSe<cnum>:IMS:RESPonse:CENTer?
```

**Return Type** Numeric

**Default** 1.0 GHz

---

**SENSe<cnum>:IMS:RESPonse:SPAN <num>**
(Read-Write) Sets and returns the Span of receiver frequencies for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled and when Sweep Type = Linear. Otherwise, this setting is ignored.

**Parameters**

- **<cnum>**  Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- **<num>**  Frequency span in Hz. All receiver frequencies should be within the range of the PNA.

**Examples**

```
SENS:IMS:RESP:SPAN 10e9
sense2:ims:response:span 100e6
```

**Query Syntax**  SENSe<cnum>:IMS:RESPonse:SPAN?

**Return Type**  Numeric

**Default**  100 MHz

---

**SENSe<cnum>:IMS:STIMulus:DFRequency <num>**

(Read-Write) Sets and returns the DeltaF (tone spacing) for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled. Otherwise, this setting is ignored.

**Parameters**

- **<cnum>**  Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- **<num>**  Delta frequency in Hz. All stimulus settings MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMS:STIM:DFRequency 1e6
sense2:ims:stimulus:dfrequency 100e6
```

**Query Syntax**  SENSe<cnum>:IMS:STIMulus:DFRequency?

**Return Type**  Numeric

**Default**  10 MHz

---

**SENSe<cnum>:IMS:STIMulus:FCENter <num>**
(Read-Write) Sets and returns the Center frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled. Otherwise, this setting is ignored.

**Parameters**

- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>` Center frequency in Hz. All stimulus settings MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMS:STIM:FCEN 1e6
sense2:ims:stimulus:fcenter 100e6
```

**Query Syntax**

`SENSe<cnum>:IMS:STIMulus:FCENter?`

**Return Type** Numeric

**Default** 1.0 GHz

---

**SENSe<cnum>:IMS:STIMulus:F1FRequency <num>**

(Read-Write) Sets and returns the F1 frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled. Otherwise, this setting is ignored.

**Parameters**

- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>` F1 frequency in Hz. All stimulus settings MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMS:STIM:F1FR 1e6
sense2:ims:stimulus:f1frequency 100e6
```

**Query Syntax**

`SENSe<cnum>:IMS:STIMulus:F1FRequency?`

**Return Type** Numeric

**Default** 995 MHz

---

**SENSe<cnum>:IMS:STIMulus:F2FRequency <num>**
(Read-Write) Sets and returns the F2 frequency for the IM Spectrum measurement. Valid ONLY when Tracking is NOT enabled. Otherwise, this setting is ignored.

**Parameters**

- `<cnum>`  Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<num>`  F2 frequency in Hz. All stimulus settings MUST be within the frequency range of the PNA.

**Examples**

```
SENS:IMS:STIM:F2FR 1e6
sense2:ims:stimulus:f2frequency 100e6
```

**Query Syntax**

`SENSe<cnum>:IMS:STIMulus:F2FRequency?`

**Return Type**  Numeric

**Default**  1.005 MHz

---

`SENSe<cnum>:IMS:SWEep:TYPE <char>`

(Read-Write) Sets and returns the method in which the spectrum of signals to view are specified. When Tracking is enabled the frequency of the main tones (f1 and f2) are always determined by the Swept IMD channel.

**Parameters**

- `<cnum>`  Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<char>`  IM Spectrum sweep type. Choose from:
  
  - **LiNear**  When Tracking is enabled, allows tuning the Response Settings (receiver) to any values within the frequency range of the PNA. When Tracking is NOT enabled also allows setting the Stimulus (sources) to any values within the frequency range or the PNA.
  
  - **SECond**  The receiver is tuned to view the 2nd order products (f2-f1 and f1+f2) of the main tones that are currently specified in Stimulus Settings. When Tracking is enabled, the main tones are specified in the Swept IMD channel.
  
  - **THIRD**  The receiver is tuned to view the 3rd order products (2f1-f2 and 2f2-f1) of the main tones that are currently specified in Stimulus Settings. When Tracking is enabled, the main tones are specified in the Swept IMD channel.
  
  - **NTH**  The frequency range is set to N * DeltaF. This algorithm will NOT tune the receivers to view the EVEN order products.
Examples

SENS:IMS:SWEep:TYPE LIN
sense2:ims:sweep:type nth

Query Syntax

SENSe<cnum>:IMS:SWEep:TYPE?

Return Type

Character

Default

NTH

SENSe<cnum>:IMS:SWEep:ORDer <num>

(Read-Write) Sets and returns the order number of signals to view when SENS:IMS:SWE:TYPE NTH is specified.

Parameters

- <cnum>: Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- <num>: Order number of IM products to view. The frequency range is set to N (this number) x DeltaF (set with SENS:IMS:STIM:DFR).

Examples

SENS:IMS:SWEep:ORD 5
sense2:ims:sweep:order 12

Query Syntax

SENSe<cnum>:IMS:SWEep:ORDer?

Return Type

Numeric

Default

9

SENSe<cnum>:IMS:TPOWer:COUPle[:STATe] <bool>
(Read-Write) Sets and returns the state of power coupling for F1 and F2.

**Parameters**
- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<bool>` Tone power level coupling. Choose from:
  - **ON (or 1)** - F1 and F2 power is coupled.
  - **OFF (or 0)** - F1 and F2 power is NOT coupled. Set power levels individually.

**Examples**
- `SENS:IMS:TPOW:COUP 0`
- `sense2:ims:tpower:couple:state ON`

**Query Syntax**
- `SENSe<cnum>:IMS:TPOWer:COUPle[:STATe]?

**Return Type**
- Boolean

**Default**
- ON

---

**SENSe<cnum>:IMS:TPOWer:EQUalize:STATe <bool>**

(Read-Write) Sets and returns the state of Equal Tone Power setting. Learn more about tone power.

**Parameters**
- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<bool>` Equal Tone Power state. Choose from:
  - **ON (or 1)** - Equalize f1 and f2 power.
  - **OFF (or 0)** - Do NOT equalize f1 and f2 power. Use source power cal.

**Examples**
- `SENS:IMS:TPOW:EQU 0`
- `sense2:ims:tpower:equalize:state ON`

**Query Syntax**
- `SENSe<cnum>:IMS:TPOWer:EQUalize:STATe?`

**Return Type**
- Boolean

**Default**
- OFF

---

**SENSe<cnum>:IMS:STIMulus:TPOWer:F1 <value>**
(Read-Write) Sets and returns the power level of the F1 tone. When SENS:IMS:TPOW:COUP ON (tone power is coupled), setting either F1 or F2 power sets both. This setting is ignored if SENS:IMS:TRAC:STAT is enabled.

**Parameters**

- `<cnum>` Channel number of the IM Spectrum measurement. If unspecified, value is set to 1.
- `<value>` F1 tone power level in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**

```plaintext
SENS:IMS:STIM:TPower:F1 -10
sense2:ims:stimulus:tpower:f1 0
```

**Query Syntax**

SENSe<cnum>:IMS:STIMulus:TPOWer:F1?

**Return Type**

Numeric

**Default**

-20

---

**SENSe<cnum>:IMS:STIMulus:TPOWer:F2 <value>**

(Read-Write) Sets and returns the power level of the F2 tone. When SENS:IMS:TPOW:COUP ON (tone power is coupled), setting either F1 or F2 power sets both. This setting is ignored if SENS:IMS:TRAC:STAT is enabled.

**Parameters**

- `<cnum>` Channel number of the IM Spectrum measurement. If unspecified, value is set to 1.
- `<value>` F2 tone power level in dBm. Choose a value between +30 dBm and -30 dBm.

**Examples**

```plaintext
SENS:IMS:STIM:TPower:F2 -10
sense2:ims:stimulus:tpower:f2 0
```

**Query Syntax**

SENSe<cnum>:IMS:STIMulus:TPOWer:F2?

**Return Type**

Numeric

**Default**

-20

---

**SENSe<cnum>:IMS:TPOWer:SET <char>**
(Read-Write) Sets and returns whether tone power is specified at the DUT input or output.

**Parameters**

- `<cnum>` Channel number of the IMSpectrum measurement. If unspecified, value is set to 1.
- `<char>` Choose from:

  - **INPUT** - Specified power level is set at the DUT input.
  - **OUTPUT** - Specified power level is set at the DUT output.

**Examples**

```
SENS:IMS:TPOW:SET INPUT
sense2:ims:tpower:set output
```

**Query Syntax**

```
SENSe<cnum>:IMS:TPOWer:SET?
```

**Return Type** Character

**Default** INPUT

---

**SENSe<cnum>:IMS:TRACking:CHANnel <num>**

(Read-Write) Sets and returns the IMD channel number to which the IM Spectrum channel is coupled.

**Parameters**

- `<cnum>` IMS channel. If unspecified, value is set to 1.
- `<num>` Existing IMD channel to which frequency and power settings are coupled.

**Examples**

```
SENS:IMS:TRAC:CHAN 2
sense2:ims:tracking:channel 1
```

**Query Syntax**

```
SENSe<cnum>:IMS:TRACking:CHANnel?
```

**Return Type** Numeric

**Default** First IMD channel

---

**SENSe<cnum>:IMS:TRACking:MSENable <bool>**
(Read-Write) Sets and returns the step sweep mode for the IM Spectrum channel.

**Parameters**

- `<cnum>` IMS channel. If unspecified, value is set to 1.
- `<bool>` Choose from:
  - **OFF (or 0)** - Automatic Step
  - **ON (or 1)** - Manual Step

**Examples**

```
SENS:IMS:TRAC:MSEN 1
sense2:ims:tracking:msenable 0
```

**Query Syntax**

`SENSe<cnum>:IMS:TRACking:MSENable?`

**Return Type** Boolean

**Default** 0 - Automatic

---

**SENSe<cnum>:IMS:TRACking:SINDex <num>**

(Read-Write) When `SENS:IMS:TRAC:MSEN = Manual`, sets and returns the data point number at which a sweep is performed.

**Parameters**

- `<cnum>` IMS channel. If unspecified, value is set to 1.
- `<num>` Step index. Choose from 1 to the specified number of points.

**Examples**

```
SENS:IMS:TRAC:SINDex 201
sense2:ims:tracking:sindex 1
```

**Query Syntax**

`SENSe<cnum>:IMS:TRACking:SINDex?`

**Return Type** Numeric

**Default** 1

---

**SENSe<cnum>:IMS:TRACking:STATe <bool>**
(Read-Write) When an IMD channel exists, allows the IM Spectrum frequency and power setting to track (couple with) the IMD channel settings.

Parameters

<cn> IMS channel. If unspecified, value is set to 1.

<bool> Tracking state. Choose from:

ON (or 1) - IM Spectrum frequency and power settings track the IMD channel settings.

OFF (or 0) - IM Spectrum frequency and power settings are specified in the IMS channel.

Examples

SENS:IMS:TRAC:STAT 0
sense2:ims:tracking:state ON

Query Syntax

SENSe<cn>:IMS:TRACking:STATe?

Return Type

Boolean

Default

OFF

Last Modified:

5-May-2011 Added tone power settings
11-Aug-2009 Added PMAP (9.0)
11-Aug-2009 MX New topic
Sense:Mixer Commands

Perform Mixer setup and configuration.

<table>
<thead>
<tr>
<th>SENSE:MIXer:</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPLY</td>
</tr>
<tr>
<td>AVOIDspurs</td>
</tr>
<tr>
<td>CALC</td>
</tr>
<tr>
<td>DISCard</td>
</tr>
</tbody>
</table>

ELO - More Commands

<table>
<thead>
<tr>
<th>IF:FREQ:</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIDeband</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>INPUT:FREQ:</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENominator</td>
</tr>
<tr>
<td>FIXed</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>NUMerator</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>INPUT:POWER</th>
</tr>
</thead>
<tbody>
<tr>
<td>USENominal</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LO:FREQ:</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENominator</td>
</tr>
<tr>
<td>FIXed</td>
</tr>
<tr>
<td>ILTI</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>NUMerator</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>

| LO:NAME      |

<table>
<thead>
<tr>
<th>LO:POWER</th>
</tr>
</thead>
<tbody>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>

LOAD

NORMALize

<table>
<thead>
<tr>
<th>OUTPUT:FREQ:</th>
</tr>
</thead>
<tbody>
<tr>
<td>FIXed</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>SIDeband</td>
</tr>
<tr>
<td>STARt</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>
Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about the Frequency Converter Application
- Synchronizing the PNA and Controller
- SCPI Command Tree

Note: If you are changing several mixer configuration settings, you can make all the changes first and then issue the Calculate and Apply commands as you would do from the user interface.

**SENSe<ch>:MIXer:APPLy**

(Write only) Applies the mixer setup settings and turns the channel ON. (Performs the same function as the Apply button on the mixer setup dialog box).

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1

**Examples**

- **SENS: MIX: APPL**

**Query Syntax**

- Not Applicable

**Default**

- Not Applicable

**SENSe<ch>:MIXer:AVOidspurs <bool>**
(Read Write) Sets and returns the state of the avoid spurs feature. Learn more about avoid spurs.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<bool> Avoid spurs state. Choose from

- 0 - Avoid spurs OFF
- 1 - Avoid spurs ON

**Examples**

```plaintext
SENS:MIX:AVO
sense2:mixer:avoidspurs 1
```

**Query Syntax**

SENS<ch>:MIXer:AVOidspurs?

**Return Type**

Boolean

**Default**

0 (OFF)

---

**SENS<ch>:MIXer:CALCulate <char>**

(Write only) Calculates the Input, IF, or Output frequencies of the mixer setup and updates the channel settings.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<char> Mixer port to be calculated. Choose from:

<table>
<thead>
<tr>
<th>&lt;char&gt;</th>
<th>1st or only stage requires:</th>
<th>In addition, 2nd stage requires:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>INPut</strong></td>
<td>• Output Start/Stop/Fixed frequencies</td>
<td>• IF Start/Stop/Fixed frequencies</td>
</tr>
<tr>
<td></td>
<td>• LO Start/Stop/Fixed frequencies</td>
<td>• 2nd Start/Stop/Fixed frequencies</td>
</tr>
<tr>
<td></td>
<td>• Output sideband (High or Low)</td>
<td>• IF sideband (High or Low)</td>
</tr>
<tr>
<td><strong>BOTH</strong></td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td><strong>OUTPut</strong></td>
<td>• Input Start/Stop/Fixed</td>
<td>• IF Start/Stop/Fixed</td>
</tr>
</tbody>
</table>
### SENSE<ch>:MIXer:DISCard

*(Write only)* Cancels changes that have been made to the Converter setup and reverts to the previously-saved setup. Same as the Cancel button on the **mixer setup dialog box**.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;ch&gt;</td>
<td>Any existing channel number. If unspecified, value is set to 1</td>
</tr>
</tbody>
</table>

#### Examples

**SENSe:MIX:DISC**

**Query Syntax** | Not Applicable
---|---
**Default** | Not Applicable
SENSe<ch>:MIXer:IF:FREQuency:SIDeband <char>

(Read-Write) When two LO stages are used, sets or returns whether to select the sum or difference for the IF1 product. (Input + or - LO1 = IF1)

- This setting corresponds to the buttons on LO1 on the Mixer setup dialog.
- This setting is ignored when ONE LO stage is selected.
- Also set SENS:MIX:OUTP:FREQ:SID to LOW or HIGH to determine the output frequency of the mixer.

See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<char> Sideband value. Choose from

LOW - Difference (-)

HIGH - Sum (+)

Examples

SENS:MIX:IF:FREQ:SID LOW
SENSe2:MIXer:IF:FREQ:SIDeband HIGH

Query Syntax

SENSe<ch>:MIXer:IF:FREQuency:SIDeband?

Return Type Character

Default LOW

SENSe<ch>:MIXer:IF:FREQuency:STARt <num>

(Read-Write) Sets or returns the IF start frequency value of the mixer. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<num> IF Start Frequency value

Examples

SENS:MIX:IF:FREQ:STAR 1e9
SENSe2:MIXer:IF:FREQ:STARt 1000000000

Query Syntax

SENSe<ch>:MIXer:IF:FREQuency:STARt?

Return Type Numeric

Default Not Applicable
**SENSe<ch>:MIXer:IF:FREQuency:STOP <num>**

(Read-Write) Sets or returns the stop frequency value of the mixer IF frequency. See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<num>` IF Stop Frequency value

**Examples**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:</td>
<td>MIX:IF:FREQ:STOP 2e9</td>
</tr>
<tr>
<td>SENSe2:</td>
<td>MIXer:IF:FREQ:STOP 2000000000</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENSe<ch>:MIXer:IF:FREQuency:STOP?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<ch>:MIXer:INPut:FREQuency:DENominator <value>**

(Read-Write) Sets or returns the denominator value of the Input Fractional Multiplier. See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<value>` Input denominator value.

**Examples**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:</td>
<td>MIX:INP:FREQ:DEN 5</td>
</tr>
<tr>
<td>SENSe2:</td>
<td>MIXer:INPut:FREQ:DENominator 4</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENSe<ch>:MIXer:INPut:FREQuency:DENominator?

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<ch>:MIXer:INPut:FREQuency:FIXed<value>**
**SENSe<ch>:MIXer:INPut:FREQuency:FIXed** <value>

(Read-Write) Sets or returns the fixed frequency of the input. See Note

**Parameters**

- `<ch>`: Any existing channel number. If unspecified, value is set to 1.
- `<value>`: Input frequency.

**Examples**

```
SENSe:MIXer:INPut:FREQ:FIXed 1e9
SENSe2:MIXer:INPut:FREQ:FIXed 1000000000
```

**Query Syntax**

```
SENSe<ch>:MIXer:INPut:FREQuency:FIXed?
```

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<ch>:MIXer:INPut:FREQuency:MODE** `<char>`

(Read-Write) Sets or returns the Input sweep mode.

**Parameters**

- `<ch>`: Any existing channel number. If unspecified, value is set to 1.
- `<char>`: Input sweep mode. Choose either **FIXED** or **SWEPT**

**Examples**

```
SENS:MIX:INP:FREQ:MODE FIXED
SENSe2:MIXer:INPut:FREQ:MODE swept
```

**Query Syntax**

```
SENSe<ch>:MIXer:INPut:FREQuency:MODE?
```

**Return Type**

Character

**Default**

Fixed

---

**SENSe<ch>:MIXer:INPut:FREQuency:NUMerator** `<value>`

(Read-Write) Sets or returns the numerator value of the Input Fractional Multiplier. See Note

**Parameters**

- `<ch>`: Any existing channel number. If unspecified, value is set to 1.
- `<value>`: Input numerator value.

**Examples**

```
SENS:MIX:INP:FREQ:NUM 3
SENSe2:MIXer:INPut:FREQ:NUMerator 1
```

**Query Syntax**

```
SENSe<ch>:MIXer:INPut:FREQuency:NUMerator?
```

**Return Type**

Numeric

**Default**

Not Applicable
SENSe<ch>:MIXer:INPut:FREQuency:STARt <value>
(Read-Write) Sets or returns the Input start frequency value of the mixer. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<value> Input Start frequency

Examples

SENSe:MIX:INP:FREQ:STAR 1e9
SENSe2:MIXer:INPut:FREQ:STARt 1000000000

Query Syntax
SENSe<ch>:MIXer:INPut:FREQuency:STARt?

Return Type
Numeric

Default
Not Applicable

SENSe<ch>:MIXer:INPut:FREQuency:STOP <value>
(Read-Write) Sets or returns the Input stop frequency value of the mixer. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<value> Input stop frequency

Examples

SENSe:MIX:INP:FREQ:STOP 2e9
SENSe2:MIXer:INPut:FREQ:STOP 2000000000

Query Syntax
SENSe<ch>:MIXer:INPut:FREQuency:STOP?

Return Type
Numeric

Default
Not Applicable

SENSe<ch>:MIXer:INPut:POWer <value>
(Read-Write) Sets or returns the value of the Input Power.

**Parameters**

<ch>  Any existing channel number. If unspecified, value is set to 1.

<value>  Input power in dBm.

**Examples**

```
SENS:MIX:INP:POW 9
SENSe2:MIXer:INPut:POWer 5
```

**Query Syntax**  
SENSe<ch>:MIXer:INPut:POWer?

**Return Type**  
Numeric

**Default**  
Not Applicable

---

**SENSe<ch>:MIXer:INPut:POWer:USENominal <bool>**

(Read-Write) Toggles the Use Nominal Incident Power setting ON and OFF. This setting is ONLY to be used with SMC measurements. [Learn more about Nominal Incident Power](#).

**Parameters**

<ch>  Any existing channel number. If unspecified, value is set to 1.

<value>  (boolean) - Nominal Incident Power State. Choose from:

ON (1) - Turn nominal incident power ON

OFF (0) - Turn nominal incident power OFF

**Examples**

```
SENS:MIX:INP:POW:USEN 1
SENSe2:MIXer:INPut:POWer:USENominal OFF
```

**Query Syntax**  
SENSe<ch>:MIXer:INPut:POWer:USENominal?

**Return Type**  
Boolean

**Default**  
OFF

---

**SENSe<ch>:MIXer:LO<n>:FREQuency:DENominator <value>**
(Read-Write) Sets or returns the denominator value of the LO Fractional Multiplier. See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage number. Choose 1 or 2.
- `<value>` LO denominator.

**Examples**

```
SENS:MIX:LO:FREQ:DEN 5
SENS2:MIXer:LO2:FREQ:DENominator 4
```

**Query Syntax**

```
SENSe<ch>:MIXer:LO<n>:FREQuency:DENominator?
```

**Return Type**

Numeric

**Default**

1

---

**SENSe<ch>:MIXer:LO<n>:FREQuency:FIXed <value>**

(Read-Write) Sets or returns the fixed frequency of the specified mixer LO. See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage number. Choose 1 or 2.
- `<value>` LO frequency.

**Examples**

```
SENS:MIX:LO:FREQ:FIX 1e9
SENS2:MIXer:LO2:FREQ:FIXed 1000000000
```

**Query Syntax**

```
SENSe<ch>:MIXer:LO<n>:FREQuency:FIXed?
```

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<ch>:MIXer:LO<n>:FREQuency:ILT1 <bool>**
(Read-Write) Specifies whether to use the Input frequency that is greater than the LO or less than the LO. To learn more, see the mixer setup dialog box help.

**Parameters**
- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage number. Choose 1 or 2
- `<bool>` **ON** (1) - Use the Input that is Greater than the specified LO.
  **OFF** (0) - Use the Input that is Less than the specified LO.

**Examples**
```
SENS:MIX:LO1:FREQ:ILT I 1
sense2:mixer:lo2:frequency:ilti ON
```

**Query Syntax**
```
SENSe<ch>:MIXer:LO<n>:FREQuency:ILT I?
```

**Return Type**
Boolean

**Default**
OFF

```
SENSe<ch>:MIXer:LO<n>:FREQuency:MODE <char>
```

(Read-Write) Sets or returns the LO sweep mode.

**Parameters**
- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage number. Choose 1 or 2
- `<char>` LO sweep mode. Choose either **FIXED** or **SWEPT**

**Examples**
```
SENS:MIX:LO:FREQ:MODE FIXED
SENSe2:MIXer:LO2:FREQ:MODE swept
```

**Query Syntax**
```
SENSe<ch>:MIXer:LO<n>:FREQuency:MODE?
```

**Return Type**
Character

**Default**
Fixed

```
SENSe<ch>:MIXer:LO<n>:FREQuency:NUMerator <value>
```
(Read-Write) Sets or returns the numerator value of the LO Fractional Multiplier. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.

<n> LO stage number. Choose 1 or 2

<value> LO Numerator.

Examples

SENS:MIX:LO:FREQ:NUM 5
SENS2:MIXer:LO2:FREQ:NUMerator 4

Query Syntax SENSe<ch>:MIXer:LO<n>:FREQuency:NUMerator?

Return Type Numeric

Default Not Applicable

SENSe<ch>:MIXer:LO<n>:FREQuency:STARt <value>

(Read-Write) Sets or returns the LO start frequency value. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.

<n> LO stage number. Choose 1 or 2

<value> LO Start Frequency in Hertz.

Examples

SENS:MIX:LO:FREQ:STAR 5E9

Query Syntax SENSe<ch>:MIXer:LO<n>:FREQuency:STARt?

Return Type Numeric

Default Not Applicable

SENSe<ch>:MIXer:LO<n>:FREQuency:STOP <value>
(Read-Write) Sets or returns the LO stop frequency value. See Note

Parameters

  <ch>  Any existing channel number. If unspecified, value is set to 1.
  <n>  LO stage number. Choose 1 or 2
  <value>  LO Stop Frequency in Hertz.

Examples  SENS:MIX:LO:FREQ:STOP 5E9

Query Syntax  SENSe<ch>:MIXer:LO<n>:FREQuency:STOP?

Return Type  Numeric

Default  Not Applicable

SENSe<ch>:MIXer:LO<n>:NAME <value>

(Read-Write) Sets or returns the name of the PNA internal source or external source to use as the LO in a converter measurement.

See Remotely Specifying a Source Port.

Parameters

  <ch>  Any existing channel number. If unspecified, value is set to 1.
  <n>  LO stage number. Choose 1 or 2.
  <value>  (string) - LO Source name. Use Source:CAT? to return a list of valid source ports. An external source must be configured and selected to be valid. Learn more about external source configuration.

Examples  SENS:MIX:LO:NAME "MySource"

Query Syntax  SENSe<ch>:MIXer:LO<n>:NAME?

Return Type  String

Default  "Not Controlled"

SENSe<ch>:MIXer:LO<n>:POWer <value>
(Read-Write) Sets or returns the LO Power fixed value.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage. Choose 1 or 2
- `<value>` LO Power in dBm

**Examples**

```
SENS: MIX: LO: POW 9
```

**Query Syntax**

`SENSe<ch>: MIXer: LO<n>: POWer?`

**Return Type** Numeric

**Default** Not Applicable

---

**SENSe<ch>: MIXer: LO<n>: POWer: STARt <value>**

(Read-Write) For an LO power sweep, sets or returns the LO power start value.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage. Choose 1
- `<value>` LO start power in dBm

**Examples**

```
SENS: MIX: LO1: POW: STAR -10
```

**Query Syntax**

`SENSe<ch>: MIXer: LO1: POWer: STARt?`

**Return Type** Numeric

**Default** - 20 dBm

---

**SENSe<ch>: MIXer: LO<n>: POWer: STOP <value>**
(Read-Write) For an LO power sweep, sets or returns the LO power stop value.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` LO stage. Choose 1
- `<value>` LO stop power in dBm

**Examples**

```
SENS:MIX:LO1:POW:STOP 10
```

**Query Syntax**

SENSe<ch>:MIXer:LO1:POWer:STOP?

**Return Type**

Numeric

**Default** -10 dBm

---

**SENSe<ch>:MIXer:LOAD <name>**

(Write-only) Loads a previously-configured mixer attributes file (.mxr)

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<name>` Path and file name (including .mxr extension) to load.

**Examples**

```
SENSe:MIXer:LOAD "C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr"
```

**Default** Not Applicable

---

**SENSe<ch>:MIXer:NORMalize <value>**

(Read-Write) Sets or returns the data point for normalizing the phase measurement. Learn more.

**Parameters**

- `<ch>` Channel number of the SMC measurement. If unspecified, value is set to 1.
- `<value>` Normalization data point. Choose a data point number between 1 and the max number of data points in the sweep that has the least amount of expected noise.

**Examples**

```
SENS:MIX:NORM 101
sense2:mixer:normalize 50
```

**Query Syntax**

SENSe<ch>:MIXer:NORMalize?

**Return Type**

Numeric

**Default** Middle point in the sweep
SENSe<ch>:MIXer:OUTPut:FREQuency:FIXed <value>

(Read-Write) Sets or returns the output fixed frequency of the mixer. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<value> Output fixed frequency in Hertz.

Examples

```
SENS:MIX:OUTP:FREQ:FIX 5e9
```

Query Syntax

SENSe<ch>:MIXer:OUTPut:FREQuency:FIXed?

Return Type Numeric

Default Not Applicable

SENSe<ch>:MIXer:OUTPut:FREQuency:MODE <char>

(Read-Write) Sets or returns the Output sweep mode.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<char> Output sweep mode. Choose either FIXED or SWEPT

Examples

```
SENS:MIX:OUTP:FREQ:MODE FIXED
SENSe2:MIXer:OUTput:FREQuency:MODE swept
```

Query Syntax

SENSe<ch>:MIXer:OUTPut:FREQuency:MODE?

Return Type Character

Default Fixed

SENSe<ch>:MIXer:OUTPut:FREQuency:SIDeband <value>
Specify whether to select the sum (High) or difference (Low) products.

- When one LO is used: Input + or - LO1 = Output frequency
- When two LOs are used: IF1 + or - LO2 = Output frequency

Use `SENS:MIX:IF:FREQ:SID` when two LOs are used to determine the IF1 frequency.

Use `Sens:Mixer:Stage` to set 1 or 2 LOs

See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<value>` Sideband value. Choose from

  - **LOW** - Low or Difference (-)
  - **HIGH** - High or Sum (+)

**Examples**

- `SENS:MIX:OUTP:FREQ:SID LOW`
- `SENSe2:MIXer:OUTPut:FREQ:SIDeband HIGH`

**Query Syntax**

`SENSe<ch>:MIXer:OUTPut:FREQuency:SIDeband?`

**Return Type**

Character

**Default**

LOW

---

`SENSe<ch>:MIXer:OUTPut:FREQuency:STARt <value>`

(Read-Write) Sets or returns the Output start frequency of the mixer. See Note

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<value>` Output start frequency

**Examples**

- `SENS:MIX:OUTP:FREQ:STAR 1e9`
- `SENSe2:MIXer:OUTPut:FREQ:STARt 1000000000`

**Query Syntax**

`SENSe<ch>:MIXer:OUTPut:FREQuency:STARt?`

**Return Type**

Numeric

**Default**

Not Applicable

---

`SENSe<ch>:MIXer:OUTPut:FREQuency:STOP <value>`
(Read-Write) Sets or returns the Output stop frequency of the mixer. See Note

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<value>` Output stop frequency

Examples

```
SENS:MIX:OUTP:FREQ:STOP 1e9
SENS2:MIXer:OUTPut:FREQ:STOP 1000000000
```

Query Syntax

```
SENSe<ch>:MIXer:OUTPut:FREQuency:STOP?
```

Return Type

Numeric

Default

Not Applicable

**SENSe<ch>:MIXer:PHASe <bool>**

(Read Write) Sets and returns the state of SMC Phase measurements and calibrations. Learn more.

In the User Interface, there are two "enable phase" checkboxes: in the Phase Settings dialog and in the Calibration Wizard. Checking one enables both. This single command also enables both.

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<bool>` Include Phase measurement state. Choose from

  - ON or 1 - Include phase in SMC measurements
  - OFF or 0 - Do NOT include phase in SMC measurements

Examples

```
SENS:MIX:PHAS 1
sense2:mixer:phase off
```

Query Syntax

```
SENSe<ch>:MIXer:PHASe?
```

Return Type

Boolean

Default

0 (OFF)

**SENSe<ch>:MIXer:PMAP <in>,<out>**
(Write-only) Sets the PNA to DUT port map for FCA measurements. Use SENS:MIX:PMAP:INP? and SENS:MIX:PMAP:OUTP? to read these values. Learn about selectable FCA DUT ports.

Changing the ports may limit your ability to use an internal second source. If a selected port is shared by one of the sources, then that source will not be available as an LO source. Learn more about Internal second sources.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<in>` PNA port to connect to the DUT input.
  - For SMC, choose any unused PNA port.
  - For VMC, set to 1.
- `<out>` PNA port to connect to the DUT output. Choose any unused port for SMC and VMC.

**Examples**

```
SENS:MIX:PMAP 2,1
sense2:mixer:pmap 4,2
```

**Query Syntax** Not Applicable

**Default** 1,2

---

**SENS[ch]:MIXer:PMAP:INPut?**

(Read-only) Returns the PNA port that is mapped to the DUT input. Use SENS:MIX:PMAP to set this value.

Learn about selectable FCA DUT ports.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.

**Examples**

```
SENS:MIX:PMAP:INP?
sense2:mixer:pmap:input?
```

**Default** 1

---

**SENS[ch]:MIXer:PMAP:OUTPut?**
(Read-only) Returns the PNA port that is mapped to the DUT output. Use **SENS:MIX:PMAP** to set this value.

Learn about selectable FCA DUT ports.

**Parameters**

<ch>  
Any existing channel number. If unspecified, value is set to 1.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:MIX:PMAP:OUTP?</td>
<td></td>
</tr>
<tr>
<td>sense2:mixer:pmap:output?</td>
<td></td>
</tr>
</tbody>
</table>

**Default**  
2

**SENS<ch>:MIXer:RECalculate**

(Write only) Repeats the last calculation that was performed, including all ON (state) segments in segment table.

**Parameters**

<ch>  
Any existing channel number. If unspecified, value is set to 1

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:MIX:REC</td>
<td></td>
</tr>
</tbody>
</table>

**Query Syntax**  
Not Applicable

**Default**  
Not Applicable

**SENS<ch>:MIXer:REV**re <bool>

(Read-Write) Sets whether to include SC12 sweeps during measurements.

**Parameters**

<ch>  
Any existing channel number. If unspecified, value is set to 1.

<bool>  
(Boolean) Choose from:

- **ON (1)** - Include the SC12 (reverse) sweep.
- **OFF (0)** - Do NOT Include the SC12 (reverse) sweep.

**Examples**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:MIX:REV 1</td>
<td></td>
</tr>
<tr>
<td>sense2:mixer:reverse ON</td>
<td></td>
</tr>
</tbody>
</table>

**Query Syntax**  
SENS<ch>:MIXer:REV?  

**Return Type**  
Boolean

**Default**  
ON (1)
**SENSe<ch>:MIXer:ROLE:CATalog? - Superseded**

*(Read-only)* This command is replaced with **SENSe:ROLE:CATalog** which can be used by all channels.

Returns a list of valid roles for the IMD Converter application.

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1.

**Examples**

```
SENS:MIX:ROLE:CAT?
sense2:mixer:role:catalog?
```

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ROLE:DEVice <role>,<source> - Superseded**

*(Read-Write)* This command is replaced with **SENSe:ROLE:DEVice** which can be used by all channels.

Assigns a configured external source to the specified role for the converter application.

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1.
- **<role>** (String) Role to which the external source is assigned. Choose from:
  - For IMDX and IMSX, choose from:
    - "RF2"
    - "LO1"
    - "LO2"
  - For all other converter applications, choose from:
    - "LO1"
    - "LO2"
- **<source>** String) Source name from *Source Configuration dialog.*

**Examples**

```
SENS:MIX:ROLE:DEV "LO1", "LO1Name"
sense2:mixer:role:device "LO1", "LO1Name"
```

**Query Syntax**

```
SENSe<ch>:MIXer:ROLE:DEVice? <source>
```

2704
SENSe<ch>:MIXer:SAVE <name>

(Write-only) Saves the settings for the mixer/converter test setup to a mixer attributes file.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<name> Path and file name (including .mxr extension) to save.

Examples

SENSe:MIXer:SAVE "C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr"

Default Not Applicable

SENSe<ch>:MIXer:STAGe <n>

(Read-Write) Number of IF stages (LOs) used in the mixer. See Note

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1.
<n> Number of stages. Choose either 1 or 2

Examples

SENSe:MIX:STAG 2
SENSe:MIXer:STAGE 1

Query Syntax

SENSe<ch>:MIXer:STAGe?

Return Type

Numeric

Default

1

SENSe<ch>:MIXer:XAXis <char>
(Read-Write) Sets or returns the swept frequency range to display on the X-axis for the IMDx channel.

For FCA and GCX measurements, use `CALC:MIXer:XAXis`

**Parameters**

- `<ch>` Channel number of the IMDx Converter measurement. If unspecified, value is set to 1.
- `<char>` Frequency range to display on the X-Axis. NOT case-sensitive. Choose from:
  - `INPUT` - Input frequency range
  - `LO_1` - LO frequency range
  - `OUTPUT` - Output frequency range

If the specified frequency range is not swept, the default swept range is used.

**Examples**

```
SENSe:MIXer:XAXis INPUT
sense2:mixer:xaxis LO_1
```

**Return Type** Character

**Default** Search is made in the following order until a swept range is found:

1. OUTPUT
2. INPUT (If the OUTPUT is fixed)
3. Number of Points (If ALL ranges are fixed)

---

_Last Modified:_

- 25-Mar-2010  Added Include Phase (9.2)
- 12-Feb-2010  Added Include Reverse
- 12-Oct-2009  Clarified Stages
- 13-Mar-2009  Added roles and X-axis for IMDx
- 3-Mar-2009   Fixed Mix:Calc for INP and OUTP
- 24-Feb-2009  Replaced True/False
- 17-Dec-2008  Fixed map & typo
- 2-Oct-2008   Clarification of sideband commands
- 5-May-2008   Added Port Map commands
- 6-Mar-2008   Added note to page top
- 23-Jul-2007  Clarified LO Name command
Sense:Mixer:ELO Commands

Allows measurements of a Mixer with an Embedded LO.

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Embedded LO Settings
- Synchronizing the PNA and Controller
**Note:** The Embedded LO DIAGnostic commands read data from the various broadband and precise tuning sweeps, similar to the textual and graphical data that are available in the user interface.

**SENSe<ch>:MIXer:ELO:LO:DELTa <num>**

(Read-Write) Sets and returns LO Frequency Delta. There is usually no need to set this value. Read this value to determine the difference between the LO Frequency that is stated in the Mixer dialog box and the last measured LO Frequency.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<num>` LO Frequency delta in Hertz.

**Examples**

```plaintext
SENS:MIX:ELO:LO:DELT 10.3
```

**Query Syntax**

SENSe<ch>:MIXer:ELO:LO:DELTa?

**Return Type**

Numeric

**Default** Not Applicable

**SENSe<ch>:MIXer:ELO:LO:RESet**

(Write-only) Resets the LO Delta Frequency to 0 (zero).

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1

**Examples**

```plaintext
SENS:MIX:ELO:LO:RES
sense2:mixer:elo:lo:reset
```

**Query Syntax** Not Applicable

**Default** Not Applicable

**SENSe<ch>:MIXer:ELO:NORMalize:POINt <num>**
(Read-Write) Sets and returns the sweep data point around which to perform broadband and precise tuning.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Mixer Sweep data point. Choose a data point number, between 1 and the max number of data points in the sweep, that has the least amount of expected noise.

**Examples**

```
SENS:MIX:ELO:LO:NORM:POIN 200
sense2:mixer:elo:normalize:point 101
```

**Query Syntax**

`SENS<ch>:MIXer:ELO:NORMalize:POINt?`

**Return Type**

Numeric

**Default** Center point in the sweep span

---

**SENSe<ch>:MIXer:ELO:STATe <bool>**

(Read-Write) Sets and returns the ON | OFF state of Embedded LO.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<bool>` ON | OFF state. Choose from
  - 0 - Embedded LO OFF
  - 1 - Embedded LO ON

**Examples**

```
SENS:MIX:ELO:STAT 1
sense2:mixer:elo:state 0
```

**Query Syntax**

`SENSe<ch>:MIXer:ELO:STATe?`

**Return Type**

Boolean

**Default** OFF

---

**SENSe<ch>:MIXer:ELO:TUNing:IFBW <num>**
**SENSe<ch>:MIXer:ELO:TUNing:IFBW <num>**

*(Read-Write)* Sets and returns the IF Bandwidth for Broadband and Precise tuning sweeps.

**Parameters**

- **<ch>**  Any existing channel number. If unspecified, value is set to 1
- **<num>**  IF Bandwidth

**Examples**

```
SENS:MIX:ELO:TUN:IFBW 10kHz
sense2:mixer:elo:tuning:ifbw 20e3
```

**Query Syntax**

```
SENSe<ch>:MIXer:ELO:TUNing:IFBW?
```

**Return Type**  Numeric

**Default**  30kHz

---

**SENSe<ch>:MIXer:ELO:TUNing:INTerval <num>**

*(Read-Write)* Sets and returns how often a tuning sweep is performed.

**Parameters**

- **<ch>**  Any existing channel number. If unspecified, value is set to 1
- **<num>**  Tuning sweep interval

**Examples**

```
SENS:MIX:ELO:TUN:INT 2
sense2:mixer:elo:tuning:interval 1
```

**Query Syntax**

```
SENSe<ch>:MIXer:ELO:TUNing:INTerval?
```

**Return Type**  Numeric

**Default**  1

---

**SENSe<ch>:MIXer:ELO:TUNing:ITERations <num>**
(Read-Write) Sets and returns the maximum number of tuning iterations to achieve the precise tolerance.

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Number of tuning iterations. Choose a number between 1 and 100.

Examples

```
SENS:MIX:ELO:TUN:ITER 5
sense2:mixer:elo:tuning:iterations 3
```

Query Syntax

`SENSe<ch>:MIXer:ELO:TUNing:ITERations?`

Return Type

Numeric

Default 5

---

`SENSe<ch>:MIXer:ELO:TUNing:MODE <char>`

(Read-Write) Sets and returns the method used to determine the embedded LO Frequency.

Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Tuning mode. Choose from:
  - **BROadband** Both broadband and precise tuning
  - **PRECise** Precise tuning only
  - **NONE** No tuning; just apply the LO Frequency Delta value.

Examples

```
SENS:MIX:ELO:TUN:MODE BRO
sense2:mixer:elo:tuning:mode precise
```

Query Syntax

`SENSe<ch>:MIXer:ELO:TUNing:MODE?`

Return Type

Character

Default BROadband

---

`SENSe<ch>:MIXer:ELO:TUNing:RESet`
(Write-only) Resets the tuning parameters to their default values.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SENS:MIX: ELO:TUN:RES
sense2:mixer: elo: tuning: reset
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SENSe<ch>:MIXer:ELO:TUNing:SPAN <num>**

(Read-Write) Sets and returns the frequency span for the broadband tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Broadband frequency span in Hz.

**Examples**

```
SENS:MIX: ELO:TUN:SPAN 1e6
sense2:mixer: elo: tuning: span 1mhz
```

**Query Syntax**

SENSe<ch>:MIXer:ELO:TUNing:SPAN?

**Return Type**

Numeric

**Default**

3 MHz

---

**SENSe<ch>:MIXer:ELO:TUNing:TOLerance <num>**
(Read-Write) Sets and returns the tuning tolerance for precise tuning.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1
<num> Tuning tolerance in Hz. Choose a number between .001 and 1e3.

Examples

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:MIX:ELo:TUN:TOL .5</td>
</tr>
<tr>
<td>sense2:mixer:elo:tuning:tolerance 1</td>
</tr>
</tbody>
</table>

Query Syntax

SENSe<ch>:MiXer:ELo:TUNing:TOLerance?

Return Type

Numeric

Default

1 Hz

SENSe<ch>:MiXer:ELo:DIAGnostic:CLEar

(Write-only) Clears current diagnostic information.

Parameters

<ch> Any existing channel number. If unspecified, value is set to 1

Examples

<table>
<thead>
<tr>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:MIX:ELo:DIAG:CLEar</td>
</tr>
<tr>
<td>sense2:mixer:elo:diagnostic:clear</td>
</tr>
</tbody>
</table>

Query Syntax

Not Applicable

Default

Not Applicable

SENSe<ch>:MiXer:ELo:DIAGnostic:STATus?
(Read-only) Returns a string that describes the result of the last tuning sweeps.

### Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1

### Examples

```
SENS:MIX:ELO:DIAG:STAT?
sense2:mixer:elo:diagnostic:status
```

### Return Type

String

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep:COUNT?**

(Read-only) Returns the number of tuning sweeps used for the latest embedded LO measurement.

### Parameters

- `<ch>` Any existing channel number. If unspecified, value is set to 1

### Examples

```
SENS:MIX:ELO:DIAG:SWEep:COUNT
sense2:mixer:elo:diagnostic:sweep:count?
```

### Return Type

Numeric

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep<n>:DATA?**
(Read-only) Returns an array of data that describes the data retrieved from the specified tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNT?` to find the number of sweeps taken.

**Examples**

```
SENS:MIX:ELO:DIAG:SWE2:DATA?
sense2:mixer:elo:diagnostic:sweep1:data?
```

**Return Type** Array

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep<n>:LO:DELTa?**

(Read-only) Returns the LO frequency delta from the specified tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNT?` to find the number of sweeps taken.

**Examples**

```
sense2:mixer:elo:diagnostic:sweep1:lo:delta?
```

**Return Type** Numeric

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep<n>:MARKer:ANNotation?**
(Read-only) Returns the Y-axis marker value from the specified tuning sweep. This command assumes that a marker was used. Use SENS:MIX:ELO:DIAG:SWE:MARK:STATe? to confirm if a marker was used for the tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use SENS:MIX:ELO:DIAG:SWE:COUNt? to find the number of sweeps taken.

**Examples**

SENS:MIX:ELO:DIAG:SWE2:MARKer:ANN?


**Return Type** Numeric

**Default** Not Applicable

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep<n>:MARKer:POSition?**

(Read-only) Returns the X-axis marker position from the specified tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use SENS:MIX:ELO:DIAG:SWE:COUNt? to find the number of sweeps taken.

**Examples**

SENS:MIX:ELO:DIAG:SWE2:MARKer:POS?

sense2:mixer:elo:diagnostic:sweep1:marker:position?

**Return Type** Numeric

**Default** Not Applicable

**SENSe<ch>:MIXer:ELO:DIAGnostic:SWEep<n>:MARKer:STATe?**
(Read-only) Returns whether or not a marker was used for the specified tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNT?` to find the number of sweeps taken.

**Examples**

```
sense2:mixer:elo:diagnostic:sweep1:marker:position?
```

**Return Type** Numeric

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELo:DIAGnostic:SWEep<n>:PARameter?**

(Read-only) Returns the name of the parameter of the specified tuning sweep.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1
- `<n>` Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNT?` to find the number of sweeps taken.

**Examples**

```
SENS:MIX:ELO:DIAG:SWE2:PAR?
sense2:mixer:elo:diagnostic:sweep1:parameter?
```

**Return Type** String - either "VC21" or "B,1"

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELo:DIAGnostic:SWEep<n>:TIITLE?**
(Read-only) Returns the title of the specified tuning sweep.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Tuning sweep number. Use SENS:MIX:ELO:DIAG:SWE:COUNt? to find the number of sweeps taken.

**Examples**

SENS:MIX:ELO:DIAG:SWE2:TITL?
sense2:mixer:elo:diagnostic:sweep1:title?

**Return Type** String

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELo:DIAGnostic:SWEep<n>:X:ANNotation?**

(Read-only) Returns the X-Axis annotation of the specified tuning sweep.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1

<n> Tuning sweep number. Use SENS:MIX:ELO:DIAG:SWE:COUNt? to find the number of sweeps taken.

**Examples**

sense2:mixer:elo:diagnostic:sweep1:x:annotation?

**Return Type** String - either "Hz" or "s"

**Default** Not Applicable

---

**SENSe<ch>:MIXer:ELo:DIAGnostic:SWEep<n>:X:STARt?**
Returns the start value of the specified tuning sweep.

**Parameters**

- `<ch>`: Any existing channel number. If unspecified, value is set to 1
- `<n>`: Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNt?` to find the number of sweeps taken.

**Examples**

```
sense2:mixer:elo:diagnostic:sweep1:x:start?
```

**Return Type**: Numeric

**Default**: Not Applicable

---

Returns the stop value of the specified tuning sweep.

**Parameters**

- `<ch>`: Any existing channel number. If unspecified, value is set to 1
- `<n>`: Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNt?` to find the number of sweeps taken.

**Examples**

```
SENS:MIX:ELO:DIAG:SWE2:X:STOP?
sense2:mixer:elo:diagnostic:sweep1:x:stop?
```

**Return Type**: Numeric

**Default**: Not Applicable

---

(Read-only) Returns the annotation of the specified tuning sweep.
(Read-only) Returns Y-axis annotation value of the specified tuning sweep.

**Parameters**

- **<ch>** Any existing channel number. If unspecified, value is set to 1
- **<n>** Tuning sweep number. Use `SENS:MIX:ELO:DIAG:SWE:COUNT?` to find the number of sweeps taken.

**Examples**

```
sense2:mixer:elo:diagnostic:sweep1:y:annotation?
```

**Return Type** String - either "U" or "Phase"

**Default** Not Applicable
Sense:Multiplexer Commands

Controls External Test Sets (N44xx, E5091A, "Z", and "H" series).

Click on a blue keyword to view the command details.

Red commands are superseded.

See Also

- See an example program using these commands.
- Learn about External Test Set Control
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe:MULTiplexer<id>:ADDRess <address>

(Read-Write) Sets and returns the address for the external test set at the specified ID. This command should be immediately preceded by the SENSe:MULT:TYPE command.

Parameters

- <id>  Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSe:MULT:TYPE command.

- <address> Integer  The test set address.

  - For a GPIB test set (N44xx and some specials), this is the GPIB address.
  - For a test set I/O test set (some specials), it is the position of the test set in the chain (starting at 0).
  - For USB test sets (E5091A), the address is set by DIP switches on the rear panel.
Examples

SENS:MULT1:TYPE "Z5623A_K66" ' use K66 test set, and reference it through ID 1
SENS:MULT1:ADDR 0 ' first test set in sequence
' All subsequent commands using SENS:MULT1 will refer to this test set

Query Syntax
SENSe:MULTiplexer<id>:ADDRess?

Return Type
Numeric

Default
Not Applicable

SENSe<cnum>:MULTiplexer<id>:ALLPorts <string>
(Read-Write) Sets or gets the port selections for all available ports on the specified channel.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1.

<id> Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSe:MULT:TYPE command.

<string> Comma-separated list of port selections, one for each port. Each port selection must correspond to one of the values returned by SENS:MULT:PORT:CAT?.

Do NOT include + and -.

Examples

' for channel 5 and test set 1, set port 1 to T1,
' port 2 to A, port 3 to R2+, port 4 to R3-
SENS5:MULT1:ALLP "T1,A,R2,R3 "

Query Syntax
SENSe<cnum>:MULTiplexer<id>:ALLPorts?

Return Type
STRING

Default
Not Applicable

SENSe:MULTiplexer:CATalog?
(Read-Only) Returns a comma-separated list of the external test sets models that are currently supported. Choose one of these items to send SENS:MULT1:TYPE.

Examples

SEN :MULT:CAT?

Return Type
String

Default
Not Applicable
SENSe:MULTiplexer<id>:COUNT?

(Read-Only) Returns the total number of ports of the specified test set.

Returns 0 if no test set is connected (GPIB test sets only).

Parameters

   <id>  Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSe:MULT:TYPE command.

Examples

    SENS:MULT1:COUN?
    sense:multiplexer2:count?

Return Type Numeric

Default Not Applicable

SENSe:MULTiplexer<id>:DISPLAY[:STATE] <bool>

(Read-Write) Turns ON and OFF the display of the test set control status bar. This status bar indicates the test set that is being controlled and the current port mappings. This setting is turned ON automatically when the test set is enabled.

Parameters

   <id>  Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSe:MULT:TYPE command.
   <bool> ON(1) Turns ON the display.
          OFF (0) Turns OFF the display.

Examples

    SENS:MULT1:DISP 1
    sense:multiplexer2:display:state on

Query Syntax SENSe:MULTiplexer<id>:DISPLAY[:STATE]?

Return Type Boolean

Default OFF (0)
(Read-Only) Returns the number of input ports for the specified test set.

- For test sets such as the E5091A that do NOT use jumper cables to route the stimulus and response signals, this command returns the number of test set ports that can be connected to the PNA.
- For test sets that DO use jumper cables to route the stimulus and response signals, such as the N44xx, the return value is not valid.

**Parameters**

**<id>**  Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSE:MULT:TYPE command.

**Examples**  
SENSe3:MULT:INC?  'returns the number of input ports for test set 1 on channel 3

**Return Type**  Numeric

**Default**  Not Applicable

---

**SENSe<cnum>:MULTiplexer:LAbel <string>**

(Read-Write) Sets and returns the display label for the testset on the specified channel. The label appears in a status bar at the bottom of the PNA display when SENS:MULT:DISP is set to ON.

**Parameters**

**<cnum>**  Any existing channel number. If unspecified, value is set to 1.

**<string>**  Display label text.

**Examples**  SENSe3:MULT:LAB 'High-power output'

**Query Syntax**  SENSe<cnum>:MULTiplexer:LAbel?

**Return Type**  String

**Default**  Not Applicable

---

**SENSe<cnum>:MULTiplexer<id>:OUTPut[:DATa] <num>**
(Read-Write) Sets or returns the control line value for the specified channel.

**Parameters**

- `<cnum>`  Any existing channel number. If unspecified, value is set to 1.
- `<id>`    Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the `SENSe:MULT:TYPE` command.
- `<numr>`  An integer specifying the decimal value of the control line. Values are obtained by adding weights from the following table that correspond to individual lines.

<table>
<thead>
<tr>
<th>Line</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
</tr>
<tr>
<td>5</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>32</td>
</tr>
<tr>
<td>7</td>
<td>64</td>
</tr>
<tr>
<td>8</td>
<td>128</td>
</tr>
</tbody>
</table>

**Note:**

- The E5091A interprets SENS:MULT1:OUTP 0 as all lines OFF.
- All "Z"and "H" series test sets interpret SENS:MULT1:OUTP 0 as all lines ON.

Refer to your test set documentation for setting control line values.

**Examples**

`SENSe3:MULT1:OUTP 48`  'For Z5623A K64, lines 5 and 6 are OFF; all other lines are set to ON state.'

**Query Syntax**

`SENSe<cnum>:MULTiplexer<id>:OUTPut[:DATa]`?

**Return Type**

Numeric

**Default**

Not Applicable
(Read-Only) Returns a comma-separated list of valid port selections for the specified port.

**Parameters**

- `<id>`: Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the `SENSe:MULT:TYPE` command.
- `<pnum>`: Integer - Input port number for which to return valid Output port selections. Read the number of input ports for the test set using `SENSe:MULT:INCount?`.

**Examples**

```
SENSe:MULT1:PORT3:CAT?  'returns the valid port selections for port 3
```

**Return Type**

String

**Default**

Not Applicable

---

**SENSe<cnum>:MULTiplexer<id>:PORT<pnum>:SELect <string>**

(Write-Only) Sets and returns a port mapping for a single port. If this command creates a conflict with an existing port, the PNA will resolve the conflict.

**Note:** This command is not supported for the Z5623AK44.

**Parameters**

- `<cnum>`: Channel number of the measurement. If unspecified, value is set to 1.
- `<id>`: Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the `SENSe:MULT:TYPE` command.
- `<pnum>`: Integer - Logical port number.
- `<string>`: Physical port number.

**Examples**

```
SENSe:MULT1:PORT3:SEL "4"  'sets logical port 3 to physical port 4.
```

**Return Type**

String

**Default**

Not Applicable

---

**SENSe:MULTiplexer<id>:STATe <bool>**
(Read-Write) Enables and disables (ON/OFF) the port mapping and control line output of the specified test set.

If the specified test set is not connected or not ON, then setting State ON will report an error. All other properties can be set when the test set is not connected.

When this command is set to ON, then the display of the test set status bar (SENS:MULT:DISP) is also set to ON.

**Parameters**

- `<id>`: Id of the external test set. If unspecified, Id is assumed to be 1. Must be previously set by the SENSE:_MULT:TYPE command.
- `<bool>`: ON(1) Enables test set control.
  - OFF (0) Disables test set control.

**Examples**

```plaintext
SENS:MULT1:STAT 1
sense2:multiplexer2:state on
```

**Query Syntax**

SENSe<cnum>:MULTiplexer<id>:STATe?

**Return Type**

Boolean

**Default**

OFF (0)

---

SENSe<cnum>:MULTiplexer<id>:TSET9:OUTPut[:DATA] <data> Superseded

**Note:** This command is replaced with SENS:MULT:OUTP

(Read-Write) Sets the control lines of the specified E5091A. Control lines, provided through a E5091A front panel connector, are used to control external equipment such as a part handler. See your E5091A documentation to learn more about control lines.

**Parameters**

- `<cnum>`: Channel number of the measurement. If unspecified, value is set to 1.
- `<id>`: Id of the E5091A test set. Choose from 1 or 2. Learn how to set ID value.
- `<data>`: Data value used to set control lines. Values are obtained by adding weights from the following table that correspond to individual lines. HIGH =1; LOW=0.
<table>
<thead>
<tr>
<th>Line</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
</tr>
<tr>
<td>5</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>32</td>
</tr>
<tr>
<td>7</td>
<td>64</td>
</tr>
<tr>
<td>8</td>
<td>128</td>
</tr>
</tbody>
</table>

0 - Sets all lines low

255 - Sets all lines high

**Examples**

'The following sets line 3 and 4 high. All other lines low.

SENS:MULT1:TSET9:OUTP 12

**Query Syntax**

SENSe<cnum>:MULTiplexer<id>:TSET9:OUTPut[:DATA]?

**Return Type**

Numeric

**Default**

0

---

**SENSe<cnum>:MULTiplexer<id>:TSET9:PORT1 <char> Superseded**

**Note:** This command is replaced with SENS:MULT:ALLPorts which sets ALL ports to the specified outputs.

(Read-Write) Switches Port 1 of the specified E5091A to one of the available outputs.

**Parameters**

- `<cnum>` Any existing channel number; if unspecified, value is set to 1.
- `<id>` Id of the E5091A test set. Choose from 1 or 2. [Learn how to set ID value](#).
- `<char>` Output port to be switched to. Choose from:
  - A
  - T1 - (If Port 2 already is connected to T1, then Port 2 will be switched to T2.)

**Examples**

SENS:MULT1:TSET9:PORT1 A

**Query Syntax**

SENSe<cnum>:MULTiplexer<id>:TSET9:PORT1?
SENSe<cnum>:MULTiplexer<id>:TSET9:PORT2 <char> Superseded

Note: This command is replaced with SENS:MULT:ALLPorts which sets ALL ports to the specified outputs.

(Read-Write) Switches Port 2 of the specified E5091A to one of the available outputs.

Parameters
- <cnum> Any existing channel number; if unspecified, value is set to 1.
- <id> Id of the E5091A test set. Choose from 1 or 2. Learn how to set ID value.
- <char> Output port to be switched to. Choose from:
  - T1 - If Port 1 already is connected to T1, then Port 1 will be switched to A.
  - T2

Examples

SENS:MULT1:TSET9:PORT2 T2

Query Syntax
SENSe<cnum>:MULTiplexer<id>:TSET9:PORT2?

Return Type Character
Default T1

SENSe<cnum>:MULTiplexer<id>:TSET9:PORT3 <char> Superseded

Note: This command is replaced with SENS:MULT:ALLPorts which sets ALL ports to the specified outputs.

(Read-Write) Switches Port 3 of the specified E5091A to one of the available outputs.

Parameters
- <cnum> Any existing channel number; if unspecified, value is set to 1.
- <id> Id of the E5091A test set. Choose from 1 or 2. Learn how to set ID value.
- <char> Output port to be switched to. Choose from:
  - R1 (R1+)
  - R2 (R2+)
R3 (R3+) If option 007 (7port), R2 is selected.

Examples
SENS: MULT1: TSET9: PORT3 R2

Query Syntax
SENSe<cnum>: MULTiplexer<id>: TSET9: PORT3?

Return Type
Character

Default
R1

SENSe<cnum>: MULTiplexer<id>: TSET9: PORT4 <char> Superseded

Note: This command is replaced with SENS: MULT: ALLPorts which sets ALL ports to the specified outputs.

(Read-Write) Switches Port 4 of the specified E5091A to one of the available outputs.

Parameters

<cnun> Any existing channel number; if unspecified, value is set to 1.
:id> Id of the E5091A test set. Choose from 1 or 2. Learn how to set ID value.
<char> Output port to be switched to. Choose from:

R1 (R1-)
R2 (R2-)
R3 (R3-) If option 007 (7port), R2 is selected.

Examples
SENS: MULT1: TSET9: PORT4 R2

Query Syntax
SENSe<cnum>: MULTiplexer<id>: TSET9: PORT4?

Return Type
Character

Default
R1

SENSe: MULTiplexer<id>: TYPe <name>
(Read-Write) Loads a configuration file for the specified type of external test set.

This command should be immediately followed by the `SENSe:MULT:ADDResS` command.

**Parameters**

- `<name>`: String  The name of the type of test set. Must be one of the items in the list returned by the `SENSe:MULT:CATalog?` query.

- `<id>`: Id of the external test set. Set by this command. Use consecutive values starting at 1.

**Examples**

```
SENSe:MULT1:TYPE "Z5623AK66"  ' use K66 test set, and reference it through ID 1
```

**Query Syntax**: `SENSe:MULTiplexer<id>:TYPe?`

**Return Type**: String

**Default**: Not Applicable

---

Last Modified:

- 11-Jan-2011  Minor edits
- 17-Aug-2007  Modified Data command for differences in active high and low
SENSe:NOISe Commands

Controls the Noise Figure / NFX configuration and calibration.

SENSe:NOISe:

| AVERage <num> |
| STATe <bool> |
| BWIDth <num> |

CALibration:

| METHOD <string> |
| RMETHod <string> |

ENR:FILename <string>

GAIN <num>

IMPedance:COUNt <num>

PMAP <in>,<out>

| INPut? |
| OUTPut? |

PULL[:STATe] <bool>

RECeiver <char>

SOURce:

| CKIT <string> |
| CONNector <string> |

TEMPerature:AMBient <num>

TUNer:

| ID <string> |
| INPut <string> |
| ORIent[:STATe] |
| OUTPut <string> |

Click on a blue keyword to view the command details.
Other Noise Figure SCPI commands

The calibration commands listed in this topic are supplemental to the Guided Cal commands.

- `CALC:CUSTom:DEFine` - creates a noise figure measurement.
- `CONTrol:NOISe:SOURce` or `OUTPut:MANual:NOISe[:STATe]` - turns the Noise Source ON and OFF.
- `SENSe:PATH:CONF:ELEMENT[:STATE]` - sets the port 1 and port 2 noise switches.
- `SYST:PREF:ITEM:SWIT:DEF` - Sets the default setting of the Noise Tuner switch
- `SENS:CORR:NOISe` commands - noise calibration
- `SENS:CORR:Guided` commands - performs most of noise cal.

See Also

- **Examples:**
  - [Create and Cal a Noise Figure Measurement](#)
  - [Create and Cal an NFX Measurement](#)
- [Learn about Noise Figure Application](#)
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

---

**SENSe<ch>:NOISe:AVERage[:COUNT] <num>**

*(Read-Write)* Set and read the averaging factor for the noise receiver. Learn more

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<num>` Averaging value. Choose any number from 1 to 99.

**Examples**

```
SENSe:NOISe:AVER 20
sense:noise:average:count 10
```

**Query Syntax**

`SENSe:NOISe:AVERage[:COUNT]?

**Return Type**

Numeric

**Default**

1
SENSe<ch>:NOISe:AVERage:STATe <bool>
(Read-Write) Turns noise averaging ON and OFF.

Parameters
<ch> Noise Figure channel number. If unspecified, value is set to 1.
<bool> Averaging state. Choose from

0 - Noise averaging OFF
1 - Noise averaging ON

Examples
SENSe:NOISe:AVER:STAT 0
sense:noise:average:state 1

Query Syntax
SENSe:NOISe:AVERage:STATe?

Return Type
Boolean

Default
O - OFF

SENSe<ch>:NOISe:BWIDth[:RESolution] <num>
(Read-Write) Set and read the bandwidth of the noise receiver. Learn more

Parameters
<ch> Noise Figure channel number. If unspecified, value is set to 1.
<num> Bandwidth value. Choose from:

For Sens:Noise:Receiver = NOISe (Opt 029) choose from: 800 KHz, 2 MHz, 4 MHz, 8 MHz, or 24 MHz or the numerical equivalent, such as 8e6 and so forth.

For Sens:Noise:Receiver = NORMal (Opt 028) choose from: 720 kHz or 1.2 MHz

If the value does not match one of these, it is rounded up to the next valid bandwidth value.

Examples
SENSe:NOISe:BWID 2e6
sense:noise:bwidth:resolution 8mhz

Query Syntax
SENSe:NOISe:BWIDth[:RESolution]?

Return Type
Numeric
**SENSe<ch>:NOISe:CALibration:METHod <string>**
(Read-Write) Set and read the method for performing a calibration on a noise channel.

**Parameters**
- <ch> Noise Figure channel number. If unspecified, value is set to 1.
- <string> Calibration method. NOT case-sensitive. Choose from:
  - "VectorFull" or "Vector"
  - "SParameter" (Not available for NFX measurements)
  - "ScalarFull" or "Scalar"

**Examples**
```
SENSe:NOISe:CAL:METH "Vector"
sense:noise:calibration:method "SParameter"
```

**Query Syntax**  SENSe:NOISe:CALibration:METHod?

**Return Type**  String

**Default**  "VectorFull"

---

**SENSe<ch>:NOISe:CALibration:RMEThod <string>**
(Read-Write) Set and read the method used to characterize the noise receivers. [Learn more.](#)

**Parameters**
- <ch> Noise Figure channel number. If unspecified, value is set to 1.
- <string> Receiver characterization method. NOT case-sensitive. Choose from:
  - "NoiseSource" - Use a noise source. This selection is NOT allowed when a standard PNA receiver is used as the noise receiver (SENSe:NOIS:REC NORM).
  - “PowerMeter” - Use a power meter.

**Examples**
```
SENSe:NOISe:CAL:RMET "PowerMeter"
sense:noise:calibration:rmethod "noisesource"
```

**Query Syntax**  SENSe:NOISe:CALibration:RMEThd?

---
SENSe<ch>:NOIS:ENR:FILename <string>

(Read-Write) Set and read the path and name of the ENR file associated with the noise source.

**Parameters**

<ch> Noise Figure channel number. If unspecified, value is set to 1.
<string> Full path, filename, and extension of the ENR file.

**Examples**

```
SENSe:NOIS:ENR:FIL "c:/ProgramFiles/Agilent/NetworkAnalyzer/Documents/ENR/346C.enr"
SENSe:NOIS:ENR:FIL "c:/ProgramFiles/Agilent/NetworkAnalyzer/Documents/ENR/346C.enr"
```

**Query Syntax**

SENSe:NOIS:ENR:FILename?

**Return Type** String

**Default** "NoiseSource"

SENSe<ch>:NOIS:GAIN <num>

(Read-Write) Set and read the amount of gain for the noise receiver. This setting is NOT used when Sens:Noise:Receiver = NORMal (Opt 028)

**Parameters**

<ch> Noise Figure channel number. If unspecified, value is set to 1.
<num> Gain value. Choose from:

- 0 - Low gain; select if the gain of your DUT is relatively high (>35 dB).
- 15 - Medium gain; select if the gain of your DUT is about average (20 dB to 45 dB)
- 30 - High gain; select if the gain of your DUT is relatively low (<30 dB).

Learn more about Noise Receiver Gain setting.

If the value does not match one of these, it is rounded up to the next legal value.

**Examples**

```
SENSe:NOIS:GAIN 15
SENSe:NOIS:GAIN 0
```
SENSe<ch>:NOISe:IMPedance:COUNt <num>

(Read-Write) Sets the number of impedance states to use during calibrated measurements.

**Parameters**

<ch> Noise Figure channel number. If unspecified, value is set to 1.

<num> Number of impedance states to use. Choose between 4 and the maximum number allowed by the noise tuner device. The more states that are used, the more accurate, and slower, the measurement. If the specified number exceeds the capability of the device, the measurement will use the maximum number of states the device allows.

**Examples**

SENSe:NOISe:IMP:COUNT 5

sense:noise:impedance:count 7

Query Syntax  SENSE:NOISE:IMPedance:COUNt?

Return Type  Numeric

Default  4

SENSe<ch>:NOISe:PMAP <in>,<out>

(Write-only) Set and read the DUT-to-PNA port mapping for Noise Figure. Port mapping is allowed ONLY when SENSe:NOISe:REC is set to NORM (standard PNA-X receiver).

**Parameters**

<ch> Any existing NF or NFX channel. If unspecified, value is set to 1.

<in> PNA port which is connected to the DUT input.

<out> PNA port which is connected to the DUT output.

**Examples**

SENSe:NOISe:PMAP 1,2

sense:noise:pmap 2,1

See example program

Query Syntax  Not Applicable

Default  1,2
SENSe<ch>:NOISe:PMAP:INPut?
(Read-only) Read the PNA port number to be connected to the DUT Input.

Use SENS:NOISE:PMap to set the port mapping.

Parameters

<ch> Any existing NF or NFX channel. If unspecified, value is set to 1.

Examples

SENSe:NOISe:PMAP:INPut?
sense:noise:pmap:input?

Return Type Numeric

Default 1

SENSe<ch>:NOISe:PMAP:OUTPut?
(Read-only) Read the PNA port number to be connected to the DUT Output.

Parameters

<ch> Any existing NF or NFX channel. If unspecified, value is set to 1.

Examples

SENSe:NOISe:PMAP:OUTPut?
sense:noise:pmap:output?

Return Type Numeric

Default 2

SENSe<ch>:NOISe:PULL[:STATe] <bool>
(Read-Write) Enables and disables the use of source pull technique to compute S22. Learn more.

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<bool>` Source pull technique state. Choose from:
  - **OFF** or **0** - Disable use of source pull technique.
  - **ON** or **1** - Enable use of source pull technique.

**Examples**

```
SENS:NOIS:PULL 0
sense2:noise:pull:state ON
```

**Query Syntax**

```
SENSe:NOISe:PULL[:STATe]?
```

**Return Type** Boolean

**Default** 0 - OFF

---

**SENSe<ch>:NOISe:RECeiver <char>**

(Read-Write) Sets and returns the noise receiver to use for noise measurements.

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<char>` Noise receiver. Choose from:
  - **NORMal** The standard PNA receiver. (Opt 028 and Opt 029)
  - **NOISe** The low-noise receivers. (Opt 029 only)

**Examples**

```
SENS:NOIS:REC NORM
sense2:noise:receiver noise
```

**Query Syntax**

```
SENSe:NOISe:RECeiver?
```

**Return Type** Character

**Default** NOISe

---

**SENSe<ch>:NOISe:SOURce:CKIT <string>**
(Read-Write) Set and read the Cal Kit that will be used for the Noise Source adapter.

An adapter is always necessary to connect a 346C Noise Source to the PNA port 2. Select a Cal Kit that is the same type and gender as the noise source connector.

If the Noise Source mates directly to PNA port 2, then set this type to "None".

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<string>` Cal Kit. Case sensitive.

To read possible cal kit strings for the adapter:

- Change the port connector type to that of the noise source using:
  ```
  SENS:CORR:COLL:GUID:CONN:PORT<n>
  ```
- Then read the possible cal kit strings for that port using:
  ```
  ```

**Examples**

```
SENS:NOIS:SOUR:CKIT "N4691-60004 ECal"
SENSe:NOIs:SOUs:CKIT
```

**Query Syntax**

```
SENSe:NOIS:SOURce:CKIT?
```

**Return Type**

String

**Default**

Not applicable

---

**SENSe<ch>:NOISe:SOURce:CONNector <string>**

(Read-Write) Set and read the Noise Source connector type and gender. The Agilent 346C has an "APC 3.5 male" connector.

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<string>` Noise source connector type and gender. Case sensitive.

Use **SENS:CORR:COLL:GUID:CONN:CAT?** to read possible connector strings.

**Examples**

```
SENS:NOIS:SOUR:CONN "APC 3.5 male"
SENSe:NOIs:SOUs:CONNector "APC 3.5 female"
```

**Query Syntax**

```
SENSe:NOISe:SOURce:CONNector?
```

**Return Type**

String
SENSe<ch>:NOISe:TEMPerature:AMBient <num>

(Read-Write) Sets the temperature at which the current noise measurement is occurring. Learn more

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<num>` Ambient temperature in Kelvin.

**Examples**

```
SENS:NOIS:TEMP:AMB 292
sense:noise:temperature 289
```

**Query Syntax**

SENSe:NOISe:TEMPerature:AMBient?

**Return Type**

Numeric

**Default**

295

---

SENSe<ch>:NOISe:TUNer:ID <string>

(Read-Write) Set and read the identity of the noise tuner. This is an ECal model and serial number string. To read the identities of the connected ECal modules, use SENSe:CORRection:CKIT:ECAL:LIST? and SENSe:CORRection:CKIT:ECAL<mod>:INFormation?

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<string>` ECal model and serial number string. The ECal module must be connected when this command is sent.

**Examples**

```
SENS:NOIS:TUN:ID "N4691-60004 ECal 02822"
sense:noise:tuner:id ""N4691-60004 ECal 02822"
```

**Query Syntax**

SENSe:NOISe:TUNer:ID?

**Return Type**

String

**Default**

Not applicable

---

SENSe<ch>:NOISe:TUNer:INPut <string>
(Read-Write) Sets and reads the port of the ECal noise tuner that is connected to the PNA SOURCE OUT.

Parameters

<ch> Noise Figure channel number. If unspecified, value is set to 1.
<string> ECal port identifier. Case sensitive.

Examples

SENSe:NOISe:TUNer:INPut "B"
sense:noise:tuner:input "A"

Query Syntax

SENSe:NOISe:TUNer:INPut?

Return Type

String

Default

"B"

SENSe<ch>:NOISe:TUNer:ORIent[:STATe] <bool>

(Read-Write) Sets the state of auto orientation for a noise tuner during Noise Figure for NFX.

Parameters

<ch> Noise Figure channel number. If unspecified, value is set to 1.
<bool> Auto-orientation state. Choose from:

**OFF** or 0 - Disable Auto-orientation

**ON** or 1 - Enable Auto-orientation

Examples

SENSe:NOISe:TUNer:ORIent 0
sense:noise:tuner:orient:state ON

Query Syntax

SENSe:NOISe:TUNer:ORIent[:STATe]?

Return Type

Boolean

Default

1 - ON

SENSe<ch>:NOISe:TUNer:OUTPut <string>
(Read-Write) Sets and reads the port of the ECal noise tuner that is connected to the CPLR THRU.

**Parameters**

- `<ch>` Noise Figure channel number. If unspecified, value is set to 1.
- `<string>` ECal port identifier. Case sensitive.

**Examples**

```
SENS:NOIS:TUN:OUTP "B"

sense:noise:tuner:output "A"
```

**Query Syntax**

`SENSe:NOISe:TUNer:OUTput?`

**Return Type**

String

**Default**

"A"

---

Last Modified:

- 9-Jun-2011 Added RMethod (A.09.41)
- 11-Jan-2011 Added count keyword to average factor
- 2-Mar-2010 Added Noise Receiver command (9.2)
- 6-Oct-2009 Added NFX commands (9.1)
- 21-Jun-2007 MX New topic
**Sense:Offset Commands Superseded**

*Note:* These commands are replaced by the `Sense:FOM` commands which include the features of the new FOM dialog. Although these old commands will continue to work, they cannot be mixed with the new commands.

Sets the offset frequency functions, causing the stimulus and response frequencies to be different.

![SENSe:OFFSet diagram](SENSe:OFFSet_diagram.png)

Click on a blue keyword to view the command details.

**See Also**

- [Example Programs](#)
- [Learn about Frequency Offset](#)
- [Synchronizing the PNA and Controller](#)
- [SCPI Command Tree](#)

---

**SENSe<nump>:OFFSet: CW <bool>**

*(Read-Write)* Turns stimulus CW Override mode ON or OFF. Use this setting to establish a fixed (CW) stimulus frequency while measuring the Response over a swept frequency range.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<bool>`
  - ON (or 1) - turns CW override ON.
  - OFF (or 0) - turns CW override OFF.

**Examples**

- `SENS:OFFS:CW ON`
- `sense2:offset:cw off`

**Query Syntax**

`SENSe<nump>:OFFSet: CW?`

**Return Type**

Boolean

**Default**

OFF

---

**SENSe<nump>:OFFSet:DIVisor <num>**
(Read-Write) Specifies (along with the multiplier) the value to multiply by the stimulus.

Parameters

<cnmu> Any existing channel number. If unspecified, value is set to 1
<num> Divisor value. Range is 1 to 1000

Examples
SENS:OFFS:DIV 3
sense2:offset:divisor 2

Query Syntax
SENSe<cnmu>:OFFSet:DIVisor?

Return Type
Numeric

Default
1

SENSe<cnmu>:OFFSet:MULTiplier <num>

(Read-Write) Specifies (along with the divisor) the value to multiply by the stimulus.

Parameters

<cnmu> Any existing channel number. If unspecified, value is set to 1
<num> Multiplier value. Range is +/- 1000. Negative multipliers cause the stimulus to sweep in decreasing direction. For mixer measurements, this would be for setups requiring the RF frequency to be less than LO frequency

Examples
SENS:OFFS:MULT 2
sense2:offset:multiplier 4

Query Syntax
SENSe<cnmu>:OFFSet:MULTiplier?

Return Type
Numeric

Default
1

SENSe<cnmu>:OFFSet:OFFSet <num>
Specifies an absolute offset frequency in Hz. For mixer measurements, this would be the LO frequency.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<num>`: Offset frequency. Range is +/- 1000 GHz. Offsets can be positive or negative

**Examples**

```
SENS:OFFS:OFFS 1GHz
sense2:offset:offset 1e9
```

**Query Syntax**

`SENSe<cnum>:OFFSet:OFFSet?`

**Return Type** Numeric

**Default** 0 Hz

---

`SENSe<cnum>:OFFSet:STARt?`

(Read-Only) Returns the response start frequency

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SENS:OFFS:STAR?
sense2:offset:start?
```

**Return Type** Numeric

**Default** Not applicable

---

`SENSe<cnum>:OFFSet:[STATe] <bool>`
(Read-Write) Enables Frequency Offset Mode on ALL measurements that are present on the active channel. This immediately causes the source and receiver to tune to separate frequencies. The receiver frequencies are specified with the other SENS:OFFSet commands. To make the stimulus settings use the SENS:FREQ commands.

Tip: To avoid unnecessary errors, first make other offset frequency settings, then set Frequency Offset ON.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<bool>` ON (or 1) - turns Frequency Offset ON.
  
  OFF (or 0) - turns Frequency Offset OFF.

**Examples**

```
SENS:OFFS ON
sense2:offset:state off
```

**Query Syntax**

SENSe<cnum>:OFFSet:[STATe]?

**Return Type** Boolean

**Default** OFF (0)

---

**SENSe<cnum>:OFFSet:STOP?**

(Read-Only) Returns the response stop frequency.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SENS:OFFS:STOP
sense2:offset:stop
```

**Return Type** Numeric

**Default** Not applicable

---

Last Modified:

23-Jul-2007  Superseded Note.
Sense:Path:Config Commands

Controls the path configuration settings.

SENSe:PATH:CONFig

| CATalog?  |
| COPY     |
| DELeTe   |
| DTEXt    |
| ELEMent  |
  | CATalog? |
  | [STATe]  |
  | VALue:CATalog |
| NAME     |
| SELect   |
| STORe    |

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Path Configuration
- Synchronizing the PNA and Controller
- SCPI Command Tree

SENSe:PATH:CONFig:CATalog?

(Read-only) Returns a list of configuration names stored in the PNA.

| Examples | SENS : PATH : CONF : CAT? |
| Return Type | Comma-separated list of double-quoted strings |
| Default | Not Applicable |

SENSe<ch>:PATH:CONFig:COPY <num>
Copies the mechanical switch and attenuator settings from the specified channel <num> to channel <ch>.

To avoid potential conflicts, all port couplings in the calling channel will be turned OFF and all port attenuator settings will be set to manual before copying the switch or attenuator settings. The two channels CAN be of different measurement classes.

Use SYSTem:MACRo:COPY:CHANnel to copy ALL settings from one channel to another.

Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;ch&gt;</td>
<td>Channel number to copy mechanical settings to. If unspecified, value is set to 1.</td>
</tr>
<tr>
<td>&lt;num&gt;</td>
<td>Channel number to copy mechanical settings from.</td>
</tr>
</tbody>
</table>

Examples

Copies mechanical settings from chan 2 to chan 1.

SENS1:PATH:CONF:COPY 2

Return Type Not Applicable

Default Not Applicable

SENSe:PATH:CONFig:DELete <string>

(Write-only) Deletes the specified configuration name from the PNA. The factory configurations cannot be deleted. This is the only method of distinguishing a factory configuration from a user-named configuration.

Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;string&gt;</td>
<td>Configuration name to be deleted.</td>
</tr>
</tbody>
</table>

Examples

SENS:PATH:CONF:DEL "MyMixer"

Return Type Not Applicable

Default Not Applicable

SENSe:PATH:CONFig:DTEXt <string>
(Read-Write) Write and read descriptive text associated with the configuration. This text is displayed in the path configuration dialog. Text is generally used to describe external connections that must be made manually to complete the configuration setup.

**Parameters**

<string> Descriptive text enclosed in quotes. Double quotes are not allowed within the descriptive text.

**Examples**

SENS:PATH:CONF:DTEX "Connect J1 jumper on the rear panel."

**Query Syntax**

SENSe<ch>:PATH:CONFig:DTEXt?

**Return Type**

String

**Default** Not Applicable

---

**SENSe<ch>:PATH:CONFig:ELEMENT:CATalog?**

(Read-only) Returns the names of configurable elements as a comma-delimited list of strings. See a list of configurable elements and settings for various PNA models.

**Parameters**

<ch> Any existing channel number; if unspecified, value is set to 1.

**Examples**

SENS:PATH:CONF:ELEM:CAT?

'returns

"Combiner", "Src1", "Src2"

**Default** Not Applicable

---

**SENSe<ch>:PATH:CONFig:ELEMENT[:STATE] <elem>, <setting>**
(Read-Write) Write or read the setting of a specified element in the current configuration. See a list of configurable elements and settings for various PNA models.

Parameters

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<elem>` Name of the element for which a setting is to be made.

Examples

```
SENS:PATH:CONF:ELEM "Combiner", "Normal"
```

Query Syntax

SENSe<ch>:PATH:CONFig:ELEMent? "Combiner"

Returns the current state of the Combiner element.

Return Type

String

Default

Not applicable

SENSe<ch>:PATH:CONFig:ELEMENT:VAL ue:CATalog? <element>

(Read-only) Returns the list of valid settings that can be used with the specified element. See a list of configurable elements and settings for various PNA models.

Parameters

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<element>` String. Element name for which to return valid settings.

Examples

```
  'returns "Normal", "Reversed"
```

Default

Not Applicable

SENSe<ch>:PATH:CONFig:NAME?
(Read-only) Returns the name of the current configuration only if NO individual element settings had been changed since selecting or storing a configuration. When element settings change, the path configuration name is cleared.

**Parameters**

<ch> Any existing channel number; if unspecified, value is set to 1.

**Examples**

SENS:PATH:CONF:NAME?
'returns  "Default"

**Return Type**

String

**Default**

Not Applicable

---

**SENSe<ch>:PATH:CONFig:SELect <string>**

(Write only) Loads the named configuration onto the specified channel.

Use SENSe:PATH:CONFig:CAT? to return the configuration names that are stored on the PNA.

**Parameters**

<ch> Any existing channel number; if unspecified, value is set to 1.

<string> Configuration name. "Default" is the default factory configuration.

**Examples**

SENS:PATH:CONF:SEL 'default'
sense2:path:CONFig:select "MyMixer"

**Query Syntax**

Not Applicable

**Default**

"Default"

---

**SENSe<ch>:PATH:CONFig:STORe <name>**
(Write only) Saves the path configuration currently associated with channel <ch> to the specified configuration name.

**Parameters**

- **<ch>** Any existing channel number; if unspecified, value is set to 1.
- **<name>** String. Configuration name. Factory configurations can NOT be overwritten. Specifying the name of a pre-defined factory configuration will result in an error.

**Examples**

```
SENS:PATH:CONF:STOR "MyMixer"
```

**Query Syntax** Not Applicable

**Default** Not Applicable

---

Last Modified:

- 3-May-2011  Added copy command
- 24-Mar-2010  Changed CONFiguration to CONFig
- 14-Dec-6  MX New Topic
Sense:Power Command

Learn about Receiver Attenuation

SENSe<nun>:POWer:ATTenuator <recvr>,<num>

(Read-Write) Sets the attenuation level for the specified receiver.

**Note:** Attenuation cannot be set with Sweep Type set to Power

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<recvr>`: Receiver to get attenuation. Choose from:
  - AREceiver - receiver A
  - BREceiver - receiver B
  - CREceiver - receiver C
  - DREceiver - receiver D

Receiver attenuation can NOT be set using logical receiver notation.

- `<num>`: Attenuation value in dB. To determine how many receiver attenuators, the maximum receiver attenuation, and attenuation step size, for a PNA model, see PNA Models and Options.

  If a number other than these is entered, the analyzer will select the next lower valid value. For example, if 19 is entered for the E8361A/C, then 10 dB attenuation will be selected.

**Examples**
- SENSe:POW:ATT AREC, 10
- sense2:power:attenuator breceiver, 30

**Query Syntax**
SENSe<nun>:POWer:ATTenuator? <rec>

**Return Type**
Numeric

**Default**
0

Last Modified:
- July 9, 2009   Edit syntax
- 25-Oct-2007   Edit range of values
Sense:Pulse Commands

Configures the 5 pulse generators in the PNA-X.

SENSe:PULSe
  | DELay
  | DINCrement
  | INVert
  | PERiod
  | STATe
  | SUBPointtrig
  | TPOLarity
  | TTYPe
  | WIDTH

Click on a blue keyword to view the command details.

See Also

- **SENS:SWEep:PULSE** - configures the channel for pulse measurements
- PNA-X IF Path Block diagram
- SENS:IF configuration commands
- Example Programs
- PNA-X Integrated Pulse Application
- Synchronizing the PNA and Controller
- SCPI Command Tree
**Pulse Definitions**

- **D** = Delay; the time before each pulse begins
- **W** = Width; the time the pulse is ON
- **P** = Period; one complete pulse cycle
- Duty Cycle = \( \frac{W}{P} \)

**Important:** If \( D + W \) is greater than \( P \), then undefined PNA behavior results. There is NO error message or warning.

**SENSe<ch>:PULSe<n>:DELay <value>**

*(Read-Write)* Sets the pulse delay. The amount of time before a new pulse begins.

See Pulse Definition diagram.

**Parameters**

- **<ch>** Any existing channel number; if unspecified, value is set to 1.
- **<n>** Pulse generator number. Choose from 0 to 4.
  - 0 is the generator that pulses the ADC.
- **<value>** Delay value in seconds. Choose a value from about 33ns to about 70 seconds.

**Examples**

```
SENS:PULS1:DEL .5
```

**Query Syntax**

```
SENSe<ch>:PULSe<n>:DELay?
```

**Return Type** Numeric

**Default** 0

**SENSe<ch>:PULSe<n>:DINCrement <value>**
(Read-Write) Sets the pulse delay increment. The delay increments with each pulse by the <value> amount.

For example, in this diagram the delay starts as 1. On the second pulse, delay=2. On the third pulse, delay=3.

**Important:** If D + W is greater than P, then undefined PNA behavior results. There is NO error message or warning. Delay includes the incremented value.

This is useful for pulse profiling.

See Pulse Definition diagram.

**Parameters**
- **<ch>** Any existing channel number; if unspecified, value is set to 1.
- **<n>** Pulse generator number. Choose from 0 to 4.
  - 0 is the generator that pulses the ADC.
- **<value>** Delay increment value in seconds.

**Examples**
- `SENS:PULS1:DINC .5`

**Query Syntax**
- `SENSe<ch>:PULSe<n>:DINCrement?`

**Return Type**
- Numeric

**Default**
- 0

`SENSe<ch>:PULSe<n>:INVert <bool>`
(Read-Write) Sets whether to invert the polarity of the pulse.

**Parameters**

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Pulse generator number. Choose from 0 to 4.
- `<bool>`  ON (or 1) - Invert the pulse generator polarity. This causes the pulse ON time to be active low and OFF be active high.
  
  OFF (or 0) - Do NOT Invert the pulse generator polarity.

**Examples**

```
SENS:PULS1:INV 1
```

**Query Syntax**

```
SENSe<ch>:PULSe:INVert?
```

**Return Type**

Boolean

**Default**

OFF (0)

---

**SENSe<ch>:PULSe:PERiod <value>**

(Read-Write) Sets the pulse-period (1/PRF) for ALL pulse generators.

The resolution of the period is 16.667nS.

See Pulse Definition diagram.

**Parameters**

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<value>` Pulse period in seconds. Choose a value from about 33ns to about 70 seconds.

**Examples**

```
SENS:PULS:PERiod .5
```

**Query Syntax**

```
SENSe<ch>:PULSe:PERiod?
```

**Return Type**

Numeric

**Default**

1e-3 sec

---

**SENSe<ch>:PULSe<n>[:STATe] <bool>**
(Read-Write) Turns the pulse output ON and OFF.

**Parameters**

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Pulse generator number. Choose from 0 to 4.
  
  0 is the generator that pulses the ADC.
- `<bool>` ON (or 1) - turns pulse output ON.
  
  OFF (or 0) - turns pulse output OFF.

**Examples**

```
SENS:PULS1 1
```

**Query Syntax**

```
SENSe<ch>:PULSe[:STATe]?
```

**Return Type**

Boolean

**Default**

OFF

---

**SENSe<ch>:PULSe<n>:SUBPointtrig <bool>**

(Read-Write) Enables / Disables subpoint triggering. When enabled and performing Point Averaging, each rising edge of P0 triggers a subpoint (one of N acquisitions in an N point average). Must also enable the P0 generator using SENS:PULS0:STAT.

**Learn more about the PNA-X pulse generators.**

**Parameters**

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Pulse generator number. **Must be 0** as this is the generator that triggers the ADC.
- `<bool>` ON (or 1) - turns subpoint triggering ON.
  
  OFF (or 0) - turns subpoint triggering OFF.

**Examples**

```
SENS:PULS0:SUBP 1
```

**Query Syntax**

```
SENSe<ch>:PULSe0:SUBPointtrig?
```

**Return Type**

Boolean

**Default**

OFF

---

**SENSe<ch>:PULSe:TPOLarity <char>**
Sets the polarity of the trigger signal to which the internal pulse generators will respond when being externally triggered at the PulseSyncIn pin.

**Note:** This feature requires DSP version: **4.0 FPGA: 34** or higher. [Learn more](#).

Learn more about the PNA-X pulse generators.

### Parameters

- **<ch>** Any existing channel number; if unspecified, value is set to 1.
- **<char>** Pulse polarity. Choose from:
  - **POSitive** - PNA responds to rising edge or HIGH level
  - **NEGative** - PNA responds to falling edge or LOW level.

Set Edge or Level triggering using `SENS:PULS:TTYPe`.

### Examples

```plaintext
SENS:PULS:TPOL NEG
```

### Query Syntax

`SENSe<ch>:PULSe<ch>:TTYPe`

### Return Type

Character

### Default

`POSitive` - Also the polarity used when the PNA-X does not have the required DSP hardware.

---

Sets the type of trigger signal to which the internal pulse generators will respond when being externally triggered at the PulseSyncIn pin.

**Note:** This feature requires DSP version: **4.0 FPGA: 34** or higher. [Learn more](#).

Learn more about the PNA-X pulse generators.

### Parameters

- **<ch>** Any existing channel number; if unspecified, value is set to 1.
- **<char>** Trigger type. Choose from:
  - **EDGE** - PNA responds to the edge (rising or falling) of a signal
  - **LEVelo** - PNA responds to the level (HIGH or LOW) of a signal

Set polarity using `SENS:PULS:TPOL`.

### Examples

```plaintext
SENS:PULS:TTYP EDGE
```

---

2760
**Query Syntax**  
SENSe<ch>:PULSe:TTYPe?

**Return Type**  
Character

**Default**  
LEVel - Also the type used when the PNA-X does not have the required DSP hardware.

---

**SENSe<ch>:PULSe<n>:WIDTh <value>**

*(Read-Write)* Sets the pulse width. The amount of time that the pulse is ON.

*See Pulse Definition diagram.*

**Parameters**

- `<ch>` Any existing channel number; if unspecified, value is set to 1.
- `<n>` Pulse generator number. Choose from 0 to 4.
- `<value>` Pulse width in seconds. Choose a value from about 33ns to about 70 seconds.

**Examples**  
**SENSe:PULS:WIDT 0.5**

**Query Syntax**  
SENSe<ch>:PULSe<n>:WIDTh?

**Return Type**  
Numeric

**Default**  
1e-4 sec

---

Last Modified:

- **11-Jan-2011**  
Minor edit
- **8-Oct-2010**  
Added Invert
- **9-Dec-2009**  
Added Trigger Polarity and Type (9.1.3)
- **20-Jul-2009**  
Added subpoint trig (8.55.09)
- **2-Jan-2007**  
MX New topic
**Sense:Roscillator Command**

Learn about the Reference Osc.

See the rear-panel 10 MHz connector.

**SENSe:ROSCillator:SOURce?**

(Read-only) Applying a 10 MHz signal to the Reference Oscillator connector automatically sets the Reference Oscillator to EXTernal. This command allows you to check that it worked.

- **EXT** is returned when a signal is present at the 10 MHz Reference Oscillator connector.
- **INT** is returned when NO signal is present at the 10 MHz Reference Oscillator connector.

**Examples**

```
SENS:ROSC:SOUR?
sense:roscillator:source?
```

**Return Type** Character

**Default** Not applicable
**Sense:Segment Commands**

Defines the segment sweep settings.

Enable segment sweep with `SENS:SWE:TYPE SEGment`.

```
SENSe:Segment
  | ADD
  | ARBitrary
  | BWIDth
    | [RESolution]
    | CONTrol
  | COUNt
  | DELeTe
    | ALL
  | FREQuency
    | CENTER
    | SPAN
    | STARt
    | STOP
  | LIST
  | POWer
    | [LEVel]
    | CONTrol
  | [STATe]
  | SWEep
    | POINts
    | TIME
      | CONTrol
  | X
```
See Also

- Example: Upload and Download a Segment List
- Learn about Segment Sweep
- Synchronizing the PNA and Controller
- SCPI Command Tree

### SENSE<cnun>:SEGMENT<snun>:ADD

*(Write-only)* Adds a segment.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<snun>` Segment number to add. If unspecified, value is set to 1. Segment numbers must be sequential. If a new number is added where one currently exists, the existing segment and those following are incremented by one.

**Examples**

Two segments exist (1 and 2). The following command will add a new segment (1). The existing (1 and 2) will become (2 and 3) respectively.

```
SENS:SEGM1:ADD
sense2:segment1:add
```

**Query Syntax**

Not applicable. Use Sense:Segment:Count to determine the number of segments in a trace.

**Default**

Not Applicable

### SENSE<cnun>:SEGMENT:ARBitrary <ON | OFF>
(Read-Write) Enables you to setup a segment sweep with arbitrary frequencies. The start and stop frequencies of each segment can overlap other segments. Also, each segment can have a start frequency that is greater than its stop frequency which causes a reverse sweep over that segment. Learn more about Arbitrary Segment Sweep.

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<ON | OFF>`
  - ON (or 1) - Allows the setup of arbitrary segment sweep.
  - OFF (or 0) - Prevents the setup of arbitrary segment sweep.

Examples

```
SENS:SEGM:ARB ON
sense2:segment:arbitrary off
```

Query Syntax

SENSe<cnum>:SEGMent:ARBitrary?

Return Type

Boolean (1 = ON, 0 = OFF)

Default

OFF

---

**SENSe<cnum>:SEGMent<snum>:BWIDth[:RESolution] <num>**

(Read-Write) Sets the IF Bandwidth for the specified segment. First set SENS:SEGM:BWIDth:CONTrol ON. All subsequent segments that are added assume the new IF Bandwidth value.

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<snum>` Segment number to modify. Choose any existing segment number.
- `<num>` IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. (Click to see the lists.) If an invalid number is specified, the analyzer will round up to the closest valid number.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See **SCPI Syntax** for more information.

Examples

```
SENS:SEGM:BWID 1KHZ
sense2:segment2:bwidth:resolution max
```

Query Syntax

SENSe<cnum>:SEGMent<snum>:BWIDth[:RESolution]?

Return Type

Numeric

Default

See **Preset IFBW** for your PNA model.

---

**SENSe<cnum>:SEGMent:BWIDth[:RESolution]:CONTrol <ON | OFF>**
(Read-Write) Specifies whether the IF Bandwidth resolution can be set independently for each segment.

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<ON | OFF>`: ON (or 1) - turns Bandwidth control ON. Bandwidth can be set for each segment. OFF (or 0) - turns Bandwidth control OFF. Use channel bandwidth setting

**Examples**
```
SENS:SEGM:BWID:CONT ON
sense2:segment:bwidth:control off
```

**Query Syntax**
```
SENSe<cnum>:SEGMent:BWIDth[:RESolution]:CONTrol?
```

**Return Type**
Boolean (1 = ON, 0 = OFF)

**Default**
OFF

---

**SENSe<cnum>:SEGMent:COUNT?**

(Read-only) Queries the number of segments that exist in the specified channel.

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.

**Examples**
```
SENS:SEG:M:COUN?
sense2:segment:count?
```

**Return Type**
Numeric

**Default**
1 segment

---

**SENSe<cnum>:SEGMent<snum>:DELe**

(Write-only) Deletes the specified sweep segment. When ALL segments are deleted, `SEN:SWE TYPE` is automatically set to Linear because there are no segments to sweep.

**Parameters**
- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<snum>`: Number of the segment to delete. If unspecified, value is set to 1.

**Examples**
```
SENS:SEG:M:DEL
sense2:segment2:delete
```

**Query Syntax**
Not applicable

**Default**
Not Applicable
SENSe<cnum>:SEGMen<tn>:DELete:ALL

(Write-only) Deletes all sweep segments. When this command is executed, SENS:SWE:TYPE is automatically set to Linear because there are no segments to sweep.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SENS:SEG:DEL:ALL
sense2:segment:delete:all
```

**Query Syntax**

Not applicable

**Default** Not Applicable

---

SENSe<cnum>:SEGMen<snum>:FREQuency:CENTer <num>

(Read-Write) Sets the Center Frequency for the specified segment. The Frequency Span of the segment remains the same. The Start and Stop Frequencies change accordingly.

**Note:** All previous segment's Start and Stop Frequencies that are larger than the new Start Frequency are changed to the new Start Frequency. All following segment's start and stop frequencies that are smaller than the new Stop Frequency are changed to the new Stop Frequency.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<snum>` Segment number to modify. Choose any existing segment number.
- `<num>` Center Frequency in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

**Note:** This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
SENS:SEGM:FREQ:CENT 1MHZ
sense2:segment2:frequency:center 1e9
```

**Query Syntax**

SENSe<cnum>:SEGMen<snum>:FREQuency:CENTer?

**Return Type** Numeric

**Default** Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

---

SENSe<cnum>:SEGMen<snum>:FREQuency:SPAN <num>

2767
(Read-Write) Sets the Frequency Span for the specified segment. The center frequency of the segment remains the same. The start and stop frequencies change accordingly.

**Note:** All previous segment's Start and Stop Frequencies that are larger than the new Start Frequency are changed to the new Start Frequency. All following segment's start and stop frequencies that are smaller than the new Stop Frequency are changed to the new Stop Frequency.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<snum>`: Segment number to modify. Choose any existing segment number.
- `<num>`: Frequency Span in Hz. Choose any number between the **minimum** and **maximum** frequency of the analyzer.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

**Examples**

```
SENS:SEG:SPAN 1MHZ
sense2:segment2:frequency:span max
```

**Query Syntax**

`SENSe<cnum>:SEGMent<snum>:FREQuency:SPAN?`

**Return Type**

Numeric

**Default**

If first segment, frequency span of the analyzer. Otherwise 0.

---

**SENSe<cnum>:SEGMent<snum>:FREQuency:START <num>**

(Read-Write) Sets the Start Frequency for the specified sweep segment.

**Notes**

All other segment Start and Stop Frequency values that are larger than this frequency are changed to this frequency.

To return the start and stop frequency of the entire sweep (all segments), Use **SENS:FREQ:START?** and **SENS:FREQ:STOP?**

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<snum>`: Segment number to modify. Choose any existing segment number.
- `<num>`: Start Frequency in Hz. Choose any number between the **minimum** and **maximum** frequency of the analyzer.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter.
See SCPI Syntax for more information.

Examples

SENSe<cnun>:SEGMent<snum>:FREQuency:STAR 1MHZ
sense2:segment2:frequency:start minimum

Query Syntax
SENSe<cnun>:SEGMent<snum>:FREQuency:STARt?

Return Type
Numeric

Default
Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

SENSe<cnun>:SEGMent<snum>:FREQuency:STOP <num>
(Read-Write) Sets the Stop Frequency for the specified sweep segment.

Notes
All other segment Start and Stop Frequency values that are larger than this frequency are changed to this frequency.

To return the start and stop frequency of the entire sweep (all segments), Use SENS:FREQ:STARt? and SENS:FREQ:STOP?

Parameters
<cnun> Any existing channel number. If unspecified, value is set to 1
<snum> Segment number to modify. Choose any existing segment number.
<num> Stop Frequency in Hz. Choose any number between the minimum and maximum frequency of the analyzer.

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples
SENSe<cnun>:SEGMent<snum>:FREQuency:STOP 1MHZ
sense2:segment2:frequency:stop maximum

Query Syntax
SENSe<cnun>:SEGMent<snum>:FREQuency:STOP?

Return Type
Numeric

Default
If first segment, stop frequency of the analyzer. Otherwise, start frequency of the segment.

SENSe<cnun>:SEGMent:LIST <char>,<numSegs>,<data>
(Read-Write) Reads or writes the entire list of values in the segment sweep table.

2769
Note: For binary data transfer, specify 64-bit instead of 32-bit using FORMat[:DATA]. This is because higher frequencies used on PNA exceed the maximum value that can be represented by a 32-bit floating point number.

When sending/receiving this data as binary (FORMat[:DATA] REAL,64), use FORMat:BORDer to specify the correct 'endianness' (byte ordering) corresponding to your programming environment / computer platform.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<char>` Choose from:
  - SSTOP - Frequency values are Start and Stop for each segment.
  - CSPAN - Frequency values are Center and Span for each segment.
- `<numSegs>` Total number of sweep segments being input. This allows the PNA to determine how many values per-each-segment are in the input `<data>` block.
- `<data>` A 2-dimensional array of Segment data as a single data block. The data elements within a segment are each represented as real floating-point numbers as follows, and the data block is formed by interleaving all the segments together consecutively:

  1. Segment state (Boolean 1 for ON and 0 for OFF)
  2. Number of Points in the segment
  3. Start Freq (when `<char>` is SSTOP), or Center Freq (when `<char>` is CSPAN)
  4. Stop Freq (when `<char>` is SSTOP), or Freq Span (when `<char>` is CSPAN)
  5. IFBW (optional for the Write)
  6. Dwell Time (optional for the Write)
  7. Power (optional for the Write) - see below.

The first four data elements must always be supplied. After those values, data must be supplied for successive optional elements. For example, to set dwell time values, you must also supply IFBW values, because IFBW (#5) precedes dwell time (#6) in the array order.

The IF Bandwidth, Sweep Time and Source Power Control settings do NOT affect the order in which elements are interpreted.

The number of elements to supply for Power depends on the following two
settings:

1. **Source Power Option** - ON allows segments to have independent power levels.
2. **Couple Ports** = Off allows different power levels for each test port.

<table>
<thead>
<tr>
<th>CouplePorts</th>
<th>SourcePowerOption</th>
<th>Number of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>False</td>
<td>False</td>
<td>Each port has its own channel-wide power setting, which is set using <code>SOURce:POWer[:LEVel]</code>. Provide exactly 7 elements per segment. The last element (power) is ignored.</td>
</tr>
<tr>
<td>False</td>
<td>True</td>
<td>Provide 6 elements + total number of ports. The first 7 elements are still interpreted the same. The remaining elements (in-order) are interpreted as the power levels to set on that segment for Ports 2 through N, where N is the total number of ports currently enabled for the PNA or for a PNA with multiport external test set.</td>
</tr>
<tr>
<td>True</td>
<td>False</td>
<td>Provide exactly 7 elements per segment. The last element (power) is ignored.</td>
</tr>
<tr>
<td>True</td>
<td>True</td>
<td>Provide exactly 7 elements per segment. The last element (power) is honored.</td>
</tr>
</tbody>
</table>

**Examples**

```
SENS:SEG:LIST SSTOP,1,1,201,10E6,26.5E9,1E3,0,—10 1 segment, state ON, 201 points, 10 MHz to 26.5 GHz, 1kHz IFBW, 0 dwell time, —10 dBm (port powers coupled)

sense2:segment:list? cspan
```

See [Upload and Download a Segment List](#) example program

**Query Syntax**

`SENSe<cnum>:SEGment:LIST? [char].`  
If unspecified, char is set to SSTOP.
The number of data elements per segment returned will be 6 + total number of source ports, regardless of the IF Bandwidth, Sweep Time and Source Power Control settings. For the N5264A, which has no source ports, the query will return just 6 values per segment. For all other PNA models, the last elements in each segment correspond to the power level for each port.

**Return Type**
Returns block data in the format specified by `FORM:[:DATA].`

**Default**
Not Applicable

---

**SENSe<cnm>:SEGMe nt<snm>:POW<port>[[:LEV]el] <num>**


When *port power is Coupled*, setting port power for one port will apply port power for all source ports.

All subsequent segments that are added assume the new Power Level value.

**Parameters**
- `<cnm>` Any existing channel number. If unspecified, value is set to 1
- `<snm>` Segment number to modify. Choose any existing segment number.
- `<port>` Port number of the source. If unspecified, value is set to 1.
- `<num>` Power level.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, send `SOUR:POW? MAX` and `SOUR:POW? MIN`. *(SOUR:POW:ATT:AUTO* must be set to ON).

Actual achievable leveled power depends on frequency.

**Examples**

```
SEN:S:SEG:M:POW 0
sense2:segment2:power1:level -10
```

**Query Syntax**

`SENSe<cnm>:SEGMe nt<snm>:POW<port>[[:LEV]el]?`

**Return Type**
Numeric

**Default**
0

---

**SENSe<cnm>:SEGMe nt:POW<port>[[:LEV]el]:CONTr ol <ON | OFF>**
(Read-Write) Specifies whether Power Level can be set independently for each segment.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>`: ON (or 1) - turns Power Level control ON. Power level can be set for each segment. OFF (or 0) - turns Power Level control OFF. Use the channel power level setting.

**Examples**

```
SENS:SEGM:POW:CONT ON
sense2:segment:power:level:control off
```

**Query Syntax**

```
SENSe<cnum>:SEGMent:POWer[:LEVel]:CONTrol?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

---

**SENSe<cnum>:SEGMent<snum>[:STATe] <ON | OFF>**

(Read-Write) Turns the specified sweep segment ON or OFF. At least ONE segment must be ON or Sweep Mode is automatically set to Linear.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<snum>`: Segment number to be turned ON or OFF
- `<ON | OFF>`: ON (or 1) - turns segment ON. OFF (or 0) - turns segment OFF.

**Examples**

```
SENS:SEGm ON
sense2:segment2:state off
```

**Query Syntax**

```
SENSe<cnum>:SEGMent<snum>[:STATe]?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

---

**SENSe<cnum>:SEGMent<snum>:SWEep:POINts <num>**
(Read-Write) Sets the number of data points for the specified sweep segment.

Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<snum>`: Any existing segment number. If unspecified, value is set to 1
- `<num>`: Number of points in the segment. The total number of points in all segments cannot exceed 20001. A segment can have as few as 1 point.

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See [SCPI Syntax](#) for more information.

**Examples**

```
SENS:SEGM:SWE:POIN 51
sense2:segment2:sweep:points maximum
```

**Query Syntax**

```
SENSe<cnum>:SEGMent<snum>:SWEep:POINts?
```

**Return Type**

Numeric

**Default**

21

---

**SENSe<cnum>:SEGMent<snum>:SWEep:TIME <num>**

(Read-Write) Sets the time the analyzer takes to sweep the specified sweep segment.

Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<snum>`: Any existing segment number.
- `<num>`: Sweep time in seconds. Choose a number between 0 and 100

**Note:** This command will accept **MIN** or **MAX** instead of a numeric parameter. See [SCPI Syntax](#) for more information.

**Examples**

```
SENS:SEGM:SWE:TIME 1ms
sense2:segment2:sweep:time .001
```

**Query Syntax**

```
SENSe<cnum>:SEGMent<snum>:SWEep:TIME?
```

**Return Type**

Numeric

**Default**

Not Applicable

---

**SENSe<cnum>:SEGMent:SWEp:TIME:CONTrol <ON | OFF>**
(Read-Write) Specifies whether Sweep Time can be set independently for each sweep segment.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>`
  - **ON** (or 1) - turns Sweep Time control ON. Sweep Time can be set for each segment.
  - **OFF** (or 0) - turns Sweep Time control OFF. Uses the channel Sweep Time setting.

**Examples**

```plaintext
SENS:SEGM:SWE:TIME:CONT ON
sense2:segment:sweep:time:control off
```

**Query Syntax**

`SENSe<cnum>:SEGMent<snum>:SWEep:TIME:CONTrol?`

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF

---

**SENSe<cnum>:SEGMent<snum>:X:SPACing <char>**

(Read-Write) Sets X-axis spacing ON or OFF

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<snum>` Any existing segment number. (This parameter is ignored)
- `<char>`
  - **LINear** - turns X-axis point spacing OFF
  - **OBASe** - turns X-axis point spacing ON

**Examples**

```plaintext
SENS:SEGM:X:SPACing LIN
sense2:segment1:spacing obase
```

**Query Syntax**

`SENSe<cnum>:SEGMent<snum>:X:SPACing?`

**Return Type**

Character

**Default**

LINear

---

Last Modified:

- 27-Oct-2009  Added Segment:List note
- 29-Apr-2009  Added List command (8.6)
- 13-May-2008  Fixed segment delete links
- 21-Jun-2007  Increased max number of points
**Sense:Sweep Commands**

Specifies the sweep functions of the analyzer.

```
SENSe:SWEep:
  DWELI
    | AUTO
    | SDELay
  GENeration
    | POINtsweep
  GROups
    | COUNt
  MODE
  POINts
  PULSe More commands
  SPEed
  SRCPort
  TIME
    | AUTO
  TRIGger
    | DELAY
    | MODE
    | POINt
  TYPE
    | FACW
```

Click on a blue keyword to view the command details.

**See Also**

- Example Programs
SENSe<cnm>:SWEep:DWEL <num>

{(Read-Write)} Sets the dwell time between each sweep point.

- Dwell time is **ONLY** available with SENSe:SWEep:GENeration set to **STEpped**; It is **Not** available in **ANALOG**.
- Sending dwell = 0 is the same as setting SENS:SWE:DWEL:AUTO **ON**. Sending a dwell time > 0 sets SENS:SWE:DWEL:AUTO **OFF**.

**Parameters**

<cnm> Any existing channel number. If unspecified, value is set to 1
<num> Dwell time in seconds.

This command will accept **MIN** or **MAX** instead of a numeric parameter. See **SCPI Syntax** for more information.

**Examples**

```
SENSe:SWE:DWEL .1
sense2:sweep:dwell min
```

**Query Syntax**

SENSe<cnm>:SWEep:DWEL?

**Return Type**

Numeric

**Default**

0 - (Note: dwell time set to 0 is the same as dwell:auto ON)

---

SENSe<cnm>:SWEep:DWEL:AUTO <ON | OFF>
(Read-Write) Specifies whether or not to automatically calculate and set the minimum possible dwell time. Setting Auto **ON** has the same effect as setting dwell time to **0**.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>`: ON (or 1) - turns dwell ON.
  OFF (or 0) - turns dwell OFF.

**Examples**

```
SENS:SWE:DWEL:AUTO ON
sense2:sweep:dwell:auto off
```

**Query Syntax**

SENSe<cnun>:SWEep:DWELl:AUTO?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**SENSe<cnun>:SWEep:DWELl:SDELay <num>**

(Read-Write) Specifies the time to wait just before acquisition begins for each sweep. This delay is in addition to **Dwell Time** and **External Trigger** delay if enabled.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<num>`: Sweep delay in seconds.

This command will accept **MIN** or **MAX** instead of a numeric parameter. See **SCPI Syntax** for more information.

**Examples**

```
SENS:SWE:DWEL:SDEL .1
sense2:sweep:dwell:sdelay .5
```

**Query Syntax**

SENSe<cnun>:SWEep:DWELl:SDELay?

**Return Type**

Numeric

**Default**

0

---

**SENSe<cnun>:SWEep:GENeration <char>**
(Read-Write) Sets sweep as Stepped or Analog.

**Parameters**

| <cnum> | Any existing channel number. If unspecified, value is set to 1 |
| <char> | Choose from: |

**STEPped** - source frequency is CONSTANT during measurement of each displayed point. More accurate than ANALog. Dwell time can be set in this mode.

**ANALog** - source frequency is continuously RAMPING during measurement of each displayed point. Faster than STEPped. Sweep time (not dwell time) can be set in this mode.

**Examples**

```
SENSe<sweep>:GENeration
sense2:sweep:generation analog
```

**Query Syntax**

```
SENSe<cnum>:SWEep:GENeration?
```

**Return Type**

Character

**Default**

Analog

---

**SENSe<cnum>:SWEep:GENeration:POINtsweep <bool>**

(Read-Write) Turns ON and OFF point sweep mode. When enabled, the PNA measures both the forward and reverse parameters at each frequency point before stepping to the next frequency. [Learn more.](#)

**Parameters**

| <cnum> | Any existing channel number. If unspecified, value is set to 1 |
| <char> | Choose from: |

**ON** or (1) - Enable point sweep mode.

**OFF** or (0) - Disable point sweep mode.

**Examples**

```
SENSe:SWEep:GENeration
SENSe2:sweep:generation:pointsweep
```

**Query Syntax**

```
SENSe<cnum>:SWEep:GENeration:POINtsweep?
```

**Return Type**

Boolean

**Default**

OFF
SENSe<cnum>:SWEep:GROups:COUNt <num>

(Read-Write) Sets the trigger count (groups) for the specified channel. Set trigger mode to group after setting this count.

Parameters

-  <cnum> Any existing channel number. If unspecified, value is set to 1
-  <num> Count (groups) number. Choose any number between: 1 and 2e6 (1 is the same as single trigger)

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:SWE:GRO:COUN 10</td>
<td>Sets the trigger count to 10 groups for channel 1</td>
</tr>
<tr>
<td>sense2:sweep:groups:count 50</td>
<td>Sets the trigger count to 50 groups for channel 2</td>
</tr>
</tbody>
</table>

Query Syntax

SENSe<cnum>:SWEep:GROups:COUNt?

Return Type

Numeric

Default

1

SENSe<cnum>:SWEep:MODE <char>

(Read-Write) Sets the number of trigger signals the specified channel will ACCEPT.

See Triggering the PNA Using SCPI.

Parameters

-  <cnum> Any existing channel number. If unspecified, value is set to 1
-  <char> Trigger mode. Choose from:

<table>
<thead>
<tr>
<th>Trigger Mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>HOLD</td>
<td>channel will not trigger</td>
</tr>
<tr>
<td>CONTinuous</td>
<td>channel triggers indefinitely</td>
</tr>
<tr>
<td>GROups</td>
<td>channel accepts the number of triggers specified with the last SENS:SWE:GRO:COUN &lt;num&gt;. This is one of the PNA overlapped commands. Learn more.</td>
</tr>
<tr>
<td>SINGle</td>
<td>channel accepts ONE trigger, then goes to HOLD.</td>
</tr>
</tbody>
</table>

Note: Beginning with 7.50, the SINGle argument makes it no longer necessary to use SENS:SWE:GRO:COUN 1.

Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:SWE:MODE CONT</td>
<td>Sets the trigger mode to continuous for channel 1</td>
</tr>
<tr>
<td>sense2:sweep:mode hold</td>
<td>Sets the trigger mode to hold for channel 2</td>
</tr>
</tbody>
</table>

Query Syntax

SENSe<cnum>:SWEep:MODE?
SENSe<cn}>{SWEep:POINts <num>
(Read-Write) Sets the number of data points for the measurement.

Parameters
<cn>
Any existing channel number. If unspecified, value is set to 1
<num>
Choose any number between 1 and the PNA maximum number of points.

This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.

Examples
SENSe:SWEep:POINts 51
SENSe2:SWEep:POINts MAX

Query Syntax
SENSe<cn>:SWEep:POINts?

Return Type
Character
Default
CONTinuous

SENSe<cn>:SWEep:SRCPort <1 | 2>
Superseded
This command is superseded. The Calc:Par:Def:Ext and Calc:Par:Mod:Ext can now optionally include the source port.

(Read-Write) Sets the source port when making non S-parameter measurements. Has no effect on S-parameter measurements.

Parameters
<cn>
Any existing channel number. If unspecified, value is set to 1
<1 | 2>
1 - Source power comes out Port 1
2 - Source power comes out Port 2

Examples
SENSe:SWEep:SRCPort 1
SENSe2:SWEep:SRCPort 2

Query Syntax
SENSe<cn>:SWEep:SRCPort?

Return Type
Character
Default
1
SENSe<cnm>:SWEep:SPEed <ch> 
(Read-Write) Sets and returns the state of Fast Sweep mode. Learn more about Fast Sweep.

Parameters

<cnm> Any existing channel number. If unspecified, value is set to 1
<ch> Fast Sweep mode. Choose from:

- **FAST** - turns Fast Sweep Mode ON
- **NORMAL** - turns Fast Sweep Mode OFF (Normal Mode).

Examples

```
SENSe:SWE:SPE NORM
sense2:sweep:speed fast
```

Query Syntax

SENSe<cnm>:SWEep:SPEed?

Return Type
Character

Default
NORMAL

SENSe<cnm>:SWEep:TIME <nm>
(Read-Write) Sets the time the analyzer takes to complete one sweep. If sweep time accuracy is critical, use ONLY the values that are attained using the up and down arrows next to the sweep time entry box. See Sweep Time.

Parameters

<cnm> Any existing channel number. If unspecified, value is set to 1
<nm> Sweep time in seconds. Choose a number between 0 and 86,400 (24hrs),

To select the fastest sweep speed, either send MIN as an argument to this command, or send SENS:SWE:TIME:AUTO 1.

This command will accept **MIN** or **MAX** instead of a numeric parameter. See SCPI Syntax for more information.

Examples

```
SENSe:SWE:TIME 1ms
sense2:sweep:time .001
```

Query Syntax

SENSe<cnm>:SWEep:TIME?

Return Type
Numeric

Default
NA
SENSe<nump>:SWEep:TIME:AUTO <ON | OFF>

(Read-Write) Turns the automatic sweep time function ON or OFF.

**Parameters**

<nump> Any existing channel number. If unspecified, value is set to 1

<ON | OFF> ON (or 1) - turns the automatic sweep time ON.

OFF (or 0) - turns the automatic sweep time OFF.

**Examples**

SENSe:SWEep:TIME:AUTO
sense2:sweep:time:auto off

**Query Syntax**

SENSe<nump>:SWEep:TIME:AUTO?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

SENSe<nump>:SWEep:TRIGger:DELay <num>

(Read-Write) Sets and reads the trigger delay for all measurements in the specified CHANNEL. This delay is only applied while TRIG:SOURce EXTERNAL and TRIG:SCOP CURREnT. After an external trigger is applied, the start of the sweep is delayed for the specified delay value plus any inherent latency.

To apply a trigger delay for all channels (Global), use TRIG:DEL.

**Parameters**

<nump> Any existing channel number. If unspecified, value is set to 1

<num> Trigger delay value in seconds. Range is from 0 to 107 seconds

**Examples**

SENSe:SWEep:TRIGger:DELay .003
sense2:sweep:trigger:delay 1

**Query Syntax**

SENSe<nump>:SWEep:TRIGger:DELay?

**Return Type**

Numeric

**Default**

0

SENSe<nump>:SWEep:TRIGger:MODE <char>
(Read-Write) Sets and reads the trigger mode for the specified channel. This determines what EACH signal will trigger. Learn more.

**Note:** Setting Point and Sweep mode forces **Trigger:SCOpe** = CURRent

### Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<char>`: Trigger mode. Choose from:
  - **CHANNEL**: Each trigger signal causes **ALL traces** in that channel to be swept.
  - **Sweep**: Each Manual or External trigger signal causes **ALL traces that share a source port** to be swept.
  - **POINT**: Each Manual or External trigger signal causes one data point to be measured.
  - **TRACE**: Allowed ONLY when **SENS:SWE:GEN:POIN** is enabled. Each trigger signal causes two identical measurements to be triggered separately - one trigger signal is required for each measurement. Other trigger mode settings cause two identical parameters to be measured simultaneously.

### Examples

```
SENS:SWE:TRIG:MODE SWEEP
sense2:sweep:trigger:mode point
```

### Query Syntax

`SENSe<cnum>:SWEep:TRIGger:MODE?`

### Return Type

Character

**Default**

Channel

`SENSe<cnum>:SWEep:TRIGger:POINt <ON | OFF> Superseded`
This command is replaced with `SENS:SWE:TRIG:MODE POINT`

(Read-Write) Specifies whether the specified channel will measure one point for each trigger or all of the measurements in the channel. Setting any channel to POINt mode will automatically set the TRIGger:SCOPe = CURRent.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>`: ON (or 1) - Channel measures one data point per trigger.
  
  OFF (or 0) - All measurements in the channel made per trigger.

**Examples**

- `SENS:SWE:TRIG:POIN ON`
- `sense2:sweep:trigger:point off`

**Query Syntax**

`SENSe<cnum>:SWEep:TRIGger:POINt`?

**Return Type**

Boolean (1 = Point, 0 = Measurement)

**Default**

0 - Measurement

---

`SENSe<cnum>:SWEep:TYPE <char>`

(Read-Write) Sets the type of analyzer sweep mode. First set sweep type, then set sweep parameters such as frequency or power settings.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<char>`: Choose from:
  
  LINear | LOGarithmic | POWer | CW | SEGment | PHASE

**Note:** SWEep TYPE cannot be set to SEGment if there are no segments turned ON. A segment is automatically turned ON when the analyzer is started.

**Examples**

- `SENS:SWE:TYPE LIN`
- `sense2:sweep:type segment`

**Query Syntax**

`SENSe<cnum>:SWEep:TYPE?`

**Return Type**

Character

**Default**

LINear

---

`SENSe<cnum>:SWEep:TYPE:FACW <num>`
(Read-Write) Enables Fast CW sweep and sets the number of data points for the channel. **Sweep Type** must already be set to CW and FIFO must already be enabled.

**See Also**

FIFO commands

Example program

N5264A Measurement Receiver

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Number of data points to measure in Fast CW mode. This setting overwrites the standard number of points setting for the channel.

Set to 0 to disable Fast CW.

**Examples**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:SWE:TYPE:FACW 1e6</td>
<td>SENSE2:sweep:type facw 1e3</td>
</tr>
</tbody>
</table>

**Query Syntax**

SENSe<cnum>:SWEep:TYPE:FACW?

**Return Type**

Numeric

**Default**

0 - Disabled

---

Last Modified:

- 3-Dec-2010 Added sweep type Phase
- 6-May-2010 Added sweep type note
- 1-Mar-2010 Added Sweep Delay
- 10-Oct-2008 Added FACW and point sweep
- 26-Aug-2008 Added sweep speed
- 21-Jun-2007 Increased max number of points
- 18-Jun-2007 Added Single to Mode
- 24-Apr-2007 Clarified Sweep mode
X Values Command

**SENSe<cnm>:X[:VALues]?**

*(Read-only)* Returns the stimulus values for the specified channel. If the channel is sweeping the source backwards, the values will be in descending order.

**Note:** For channels that can have different traces with different x-axis values (like GCA, SMC and VMC), this command (SENs:X?) returns only the channel's default x-axis values. To get x-axis values for specific traces, use **CALC:X**?

### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;cnm&gt;</td>
<td>Any existing channel number; if unspecified, value is set to 1.</td>
</tr>
</tbody>
</table>

### Examples

- **SENs:X?**
- **sense2:x:values?**

### Return Type

Depends on **FORM:DATA** command

- **Default** Not applicable

---

Last Modified:

9-Apr-2010 Added Note
Source Commands

Controls the power delivered to the DUT.

<table>
<thead>
<tr>
<th>SOURce:</th>
</tr>
</thead>
<tbody>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>PHASE - More commands</td>
</tr>
<tr>
<td>POWER</td>
</tr>
<tr>
<td>ALC:MODE</td>
</tr>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>REceiver</td>
</tr>
<tr>
<td>FAST</td>
</tr>
<tr>
<td>IFBW</td>
</tr>
<tr>
<td>ITERation</td>
</tr>
<tr>
<td>LSPC</td>
</tr>
<tr>
<td>OFFSet</td>
</tr>
<tr>
<td>RATio</td>
</tr>
<tr>
<td>REFerence</td>
</tr>
<tr>
<td>SAFE</td>
</tr>
<tr>
<td>MAX</td>
</tr>
<tr>
<td>MIN</td>
</tr>
<tr>
<td>STEP</td>
</tr>
<tr>
<td>TOLerance</td>
</tr>
<tr>
<td>ATTenuation</td>
</tr>
<tr>
<td>AUTO</td>
</tr>
<tr>
<td>CENTer</td>
</tr>
<tr>
<td>CORRection - More commands</td>
</tr>
<tr>
<td>COUple</td>
</tr>
<tr>
<td>DETector</td>
</tr>
<tr>
<td>[LEVEL]</td>
</tr>
<tr>
<td>[IMMediate][AMPLitude]</td>
</tr>
<tr>
<td>SLOPe</td>
</tr>
<tr>
<td>STATe</td>
</tr>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>PORT</td>
</tr>
<tr>
<td>START</td>
</tr>
<tr>
<td>STOP</td>
</tr>
<tr>
<td>SPAN</td>
</tr>
<tr>
<td>START</td>
</tr>
<tr>
<td>STOP</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.
SOURce<cnum>:CATalog?

(Read-Write) Returns a list of valid port names that can be controlled. Some ports only have string names, NOT numbers. All commands that require a port argument have provisions for specifying either a port number OR a string name.

See also: Remotely Specifying a Source Port.

**Parameters**

**Examples**

```
SOUR:CAT?
sour:catalog

'Some PNA-X models return
"Port 1,Port 2,Port 3,Port 4,Port 1 Src2"
```

**Return Type** Comma-separated list of strings.

**Default** Not applicable

SOURce<cnum>:POWer<port>:ALC[:MODE] <char>, [src]
Sets and returns the ALC mode for the specified channel and port. Use `SOUR:POW:ALC:MODE:CAT?` to return a list of valid ALC modes for the PNA.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port number of the PNA. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<char>` ALC Mode.

For the PNA-X choose from:

- **INTernal** Standard ALC loop
- **OPENloop** No ALC loop


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

**Examples**

```
SOUR:POW:ALC INT
source2:power2:alc:mode openloop
source:power:alc:mode openloop,"Port 1 Src2"
```

**Query Syntax** `SOURce<cnum>:POWer<port>:ALC:MODE? [src]`

**Return Type** Character

**Default** INTernal

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:CATalog? [src]
```
(Read-only) Returns a list of valid ALC modes for the specified channel and port number. Use the returned values to set SOUR:POW:ALC:MODE.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port number of the PNA. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW:ALC:CAT?
source2:power2:alc:mode:catalog?
source:power:alc:mode:catalog? "Port 1 Src2"
```

**Return Type** Comma-separated list of strings.

**Default** Not applicable

---

**SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver <bool>, [src]**

(Read-Write) Sets and returns the state of Receiver Leveling for the specified source port.

**Learn more about Receiver Leveling**

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<bool>` Receiver Leveling state.

**ON or 1** - Receiver Leveling ON

**OFF or 0** - Receiver Leveling OFF

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW:ALC:REC 1
source2:power2:alc:mode:receiver on
source:power:alc:mode:receiver off,"Port 1 Src2"
```

See ReceiverLeveling example

**Query Syntax**

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver? [src]
```

**Return Type**

Boolean

**Default**

OFF

**SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:FAST <bool>, [src]**

*(Read-Write)* Sets and returns the state of a separate IFBW setting for leveling sweeps. ON allows a higher (faster) IFBW than the measurement sweep. It also causes leveling sweeps to be noisier.

Learn more about Receiver Leveling

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<bool>` Separate IFBW setting state.
  - **ON** or **1** - Separate IFBW setting. Specify IFBW using
    SOUR:POW:ALC:MODE:REC:IFBW
  - **OFF** or **0** - Same IFBW as the measurement sweep. Specify IFBW using
    Sens:BWID

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both
Arguments are specified, [src] takes priority.

### Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOUR:POW:ALC:REC:FAST 1</td>
<td>Source 2 power 2 ALC mode receiver fast off</td>
</tr>
<tr>
<td>source2:power2:alc:mode:receiver:fast off</td>
<td>&quot;Port 1 Src2&quot;</td>
</tr>
<tr>
<td>source:power:alc:mode:receiver:fast off,&quot;Port 1 Src2&quot;</td>
<td>See ReceiverLeveling example</td>
</tr>
</tbody>
</table>

### Query Syntax

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:FAST? [src]

Return Type: Boolean

Default: ON

### SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:IFBW <value>, [src]

(Read-Write) Sets and returns the IFBW to be used for leveling sweeps. Enable separate IFBW for leveling sweeps using SOUR:POW:ALC:MODE:REC:FAST 1

Learn more about Receiver Leveling

#### Parameters

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<port>** Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- **<value>** IFBW for leveling sweeps in Hz. The list of valid IF Bandwidths is different depending on the PNA model. (Click to see the lists.) If an invalid number is specified, the analyzer will round up to the closest valid number.

This parameter supports MIN and MAX as arguments. Learn more.


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

#### Examples

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOUR:POW:ALC:REC:IFBW 100E3</td>
<td>Source 2 power 2 ALC mode receiver ifbw 100kHz</td>
</tr>
<tr>
<td>source2:power2:alc:mode:receiver:ifbw 100kHz</td>
<td>&quot;Port 1 Src2&quot;</td>
</tr>
<tr>
<td>source:power:alc:mode:receiver:ifbw 70e3,&quot;Port 1 Src2&quot;</td>
<td>&quot;Port 1 Src2&quot;</td>
</tr>
</tbody>
</table>
**Query Syntax**

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:ITERation <value>, [src]

(Read-Write) Sets and returns the maximum iterations to be used in order to achieve the tolerance setting.

**Learn more about Receiver Leveling**

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<port>`: Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<value>`: Max iterations. Choose a value between 1 and 25.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an **external source**, **true mode balanced port**, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

SOUR:POW:ALC:REC:ITER 5

source2:power2:alc:mode:receiver:iteration 10

source:power:alc:mode:receiver:iteration 7,"Port 1 Src2"

**Query Syntax**

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:ITERation? [src]

**Return Type**

Numeric

**Default**

5

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:LSPC <bool>, [src]
(Read-Write) Sets and returns the state of **Use Last Result for Source Power Cal**. When Leveling Mode is switched back to Internal, this feature turns Source Power Cal correction ON using the latest receiver leveling correction data.

**Learn more about Receiver Leveling**

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<bool>` **State of Use Last Result for Source Power Cal.**
  - **ON** or **1** - When Leveling Mode is switched back to Internal, Source Power Cal correction is turned ON using the latest receiver leveling correction data.
  - **OFF** or **0** - When Leveling Mode is switched back to Internal, Source Power Cal correction is NOT turned ON.
- `[src]` **String.** (NOT case sensitive). Source port. Optional. Use **SOUR:CAT?** to return a list of valid port names.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

**Examples**

```
SOUR:POW:ALC:REC:LSPC 1
```

```
source2:power2:alc:mode:receiver:lspc off
```

```
source:power:alc:mode:receiver:lspc off,"Port 1 Src2"
```

**See ReceiverLeveling example**

**Query Syntax**  SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:LSPC? [src]

**Return Type**  Boolean

**Default**  OFF
(Read-Write) Sets and returns the power level offset value.

Learn more about Receiver Leveling

Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<port>`: Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<value>`: Power level offset in dB. Choose a value between +200 and -200.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

Examples

```
SOUR:POW:ALC:REC:OFFS 10
source2:power2:alc:mode:receiver:offset 5
source:power:alc:mode:receiver:offset 7,"Port 1 Src2"
```

See ReceiverLeveling example

Query Syntax

`SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:OFFSet? [src]`

Return Type

Numeric

Default

0

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:RATio? [src]
```
Returns the receiver ratio to be used with receiver leveling. This receiver ratio parameter is the same as the one set in SOUR:PHAS:PARameter.

Learn more about Receiver Leveling

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<value>`

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

Examples

```
SOUR:POW:ALC:REC:RAT 10
source2:power2:alc:mode:receiver:ratio "a1/a3,3"
source:power:alc:mode:receiver:ratio "R1/R3,3","Port 1 Src2"
```

See ReceiverLeveling example

Query Syntax

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:REFerence <rec>, [src]
```

The receivers and paired port are separated by a comma.

Return Type **String**

**Default** "a1/a3,3"
(Read-Write) Sets and returns the reference receiver to be used with Receiver Leveling.

Learn more about Receiver Leveling

Parameters

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<rec>` (String) PNA receiver. Choose the PNA physical receiver that works with the source port `<port>`.

For example: `<port 1> = "R1"`


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an **external source, true mode balanced port**, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

Examples

```
SOUR:POW:ALC:REC:REF 'r1'
source2:power2:alc:mode:receiver:reference 'r2'
source:power1:alc:mode:receiver:reference "r1","Port 1 Src2"
```

See ReceiverLeveling example

Query Syntax

`SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:REFerence? [src]`

Return Type

**String** - Name of the reference receiver.

Default

OFF

`SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE <bool>, [src]`
(Read-Write) Sets and returns the state of Safe Mode.

Learn more about Receiver Leveling

Parameters

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<port>** Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- **<bool>** Safe mode state.
  - **ON** or **1** - Safe mode ON
  - **OFF** or **0** - Safe mode OFF

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

Examples

SOUR:POW:ALC:REC:SAFE 1

source2:power2:alc:mode:receiver:safe on

source:power:alc:mode:receiver:safe off,"Port 1 Src2"

See ReceiverLeveling example

Query Syntax

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE? [src]

Return Type

Boolean

Default

OFF

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE:MAX <value>, [src]
(Read-Write) Sets and returns the maximum power level for Safe Mode.

**Learn more about Receiver Leveling**

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<port>** Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- **<value>** Maximum power level in dB.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```plaintext
SOUR:POW:ALC:REC:SAFE:MAX 10
source2:power2:alc:mode:receiver:safe:max 20
source:power:alc:mode:receiver:safe:max 15,"Port 1 Src2"
```

See ReceiverLeveling example

**Query Syntax**

`SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE:MAX? [src]`

**Return Type** Numeric

**Default** 30 dB
(Read-Write) Sets and returns the minimum power level for Safe Mode.

Learn more about Receiver Leveling

Parameters

- **<cnum>**  Any existing channel number. If unspecified, value is set to 1
- **<port>**  Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- **<value>** Minimum power level in dB.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

Examples

```
source2:power2:alc:mode:receiver:safe:min -80
source:power:alc:mode:receiver:safe:min -40,"Port 1 Src2"
```

See ReceiverLeveling example

Query Syntax

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE:MIN?

Return Type  Numeric

**Default**  -95 dB

SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE:STEP <value>, [src]
(Read-Write) Sets and returns the maximum step power level for Safe Mode.

Learn more about Receiver Leveling

Parameters

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1.
- `<port>`: Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<value>`: Maximum Step power level in dB.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

Examples

```
SOUR:POW:ALC:REC:SAFE:STEP 2
source2:power2:alc:mode:receiver:safe:step 1.5
source:power:alc:mode:receiver:safe:min 2,"Port 1 Src2"
```

See ReceiverLeveling example

Query Syntax

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:SAFE:STEP?
```

Return Type: Numeric

**Default**: 1 dB

```
SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:TOLerance <value>, [src]
```
(Read-Write) Sets and returns the tolerance value for leveling sweeps.

**Learn more about Receiver Leveling**

**Parameters**

- `<cnum>`  Any existing channel number. If unspecified, value is set to 1
- `<port>`  Source port being used for Receiver Leveling. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<value>`  Tolerance level in dB.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

- `SOUR:POW:ALC:REC:TOL .01`
- `source2:power2:alc:mode:receiver:tolerance .5`
- `source:power:alc:mode:receiver:tolerance .2,"Port 1 Src2"

See ReceiverLeveling example

**Query Syntax**  
`SOURce<cnum>:POWer<port>:ALC[:MODE]:RECeiver:TOLerance?` [src]

**Return Type**  Numeric

**Default**  .1 dB

---

**SOURce<cnum>:POWe<port>:ATTenuation <num>, [src]**

(Read-Write) Sets the attenuation level for the selected channel. Sending this command turns automatic attenuation control (`SOUR:POW:ATT:AUTO`) to OFF. If the ports are coupled, changing the attenuation on one port will also change the attenuation on all other ports. To turn port coupling OFF use `SOURce:POWe:COUPlE OFF`.

**Note:** Attenuation cannot be set with **Sweep Type** set to **Power**

See `Sens:Power:ATT` to change receiver attenuation.

**Parameters**
<cnum>  Any existing channel number. If unspecified, value is set to 1

<port>  Source port number of the PNA. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.

<num>  Attenuation value. The range of settable values depends on the PNA model. To determine the valid settings, do one of the following:

- See PNA models and options to see the range and step size for each model / option.

- Perform a query using MAX, then MIN, as an argument. Example:
  SOURce:POWer:ATT? Max  However, this will not tell you the attenuation step size.

If an invalid attenuation setting is entered, the PNA will select the next lower valid value. For example, if 19 is entered, then for an E8361A, 10 dB attenuation will be selected.

Note: This command will accept MIN or MAX instead of a numeric parameter. See SCPI Syntax for more information.


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

Examples

SOUR:POW:ATT 10
source2:power2:attenuation maximum
source:power:att 20, "Port 1 Src2"

Query Syntax  SOURce<cnum>:POWer<port>:ATTenuation? [min/max] [src]
              [min/max,src]

Return Type  Numeric

Default  0

SOURce<cnum>:POWer<port>:ATTenuation:AUTo <bool>, [src]
(Read-Write) Turns automatic attenuation control ON or OFF. Setting an attenuation value (using SOURce:POWer:ATTenuation <num>) sets AUTO OFF.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<port>` Source port number of the PNA. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<bool>` **ON** (or 1) - turns coupling ON. The analyzer automatically selects the appropriate attenuation level to meet the specified power level.
  
  **OFF** (or 0) - turns coupling OFF. Attenuation level must be set using SOURce:POWer:ATTenuation <num>.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW2:ATT:Auto On
source2:power:attentuation:auto off
sour:pow:att:auto 1, "Port 1 Src2"
```

**Query Syntax**

SOURce<cnum>:POWer:ATTenuation:Auto? [src]

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

**SOURce<cnum>:POWer:CENTer <num>**
(Read-Write) Sets the power sweep center power. Must also set: SENS:SWE:TYPE POWer and SOURce:POWer:SPAN <num>.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<num>`: Center power. Actual achievable leveled power depends on frequency.

**Examples**

```
SOUR:POW:CENT -15
source2:power:center -7
```  

**Query Syntax**

SOURce<cnum>:POWer:CENTer?

**Return Type**

Numeric

**Default**

0 dBm

---

**SOURce<cnum>:POWer:COUPle <ON | OFF>**

(Read-Write) Turns Port Power Coupling ON or OFF.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<ON | OFF>`: ON (or 1) - turns coupling ON. The same power level is used for both source ports.  
  OFF (or 0) - turns coupling OFF. Power level can be set individually for each source port.

**Examples**

```
SOUR:POW:COUP ON
source2:power:couple off
```  

**Query Syntax**

SOURce<cnum>:POWer:COUPle?

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

ON

---

**SOURce<cnum>:POWer:DETector <char> OBSOLETE**
The PNA models with external leveling are now OBSOLETE.

Sets the source leveling loop as Internal or External.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<char>`: `INTERNAL` - Internal leveling is applied to the source
  
  `EXTERNAL` - External leveling is applied to the source through a rear-panel connector. ONLY provided on 3 GHz, 6 GHz, and 9 GHz PNA models.

**Examples**

- `SOUR:POW:DET INT`  
- `source2:power:detector` `EXTERNAL`  

**Query Syntax**

- `SOURce<cnum>:POWer:DETector?`

**Return Type**

- Character

**Default**

- `INTERnal`

---

**SOURce<cnum>:POWer<port>[:LEVEL][:IMMediate] [:AMPLitude] <num>, [src]**

(Read-Write) Sets the RF power output level.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<port>`: Source port number of the PNA. If unspecified, value is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<num>`: Source power in dBm.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, perform a query using MAX, then MIN, as an argument. (SOUR:POW:ATT:AUTO must be set to ON) Example: SOURce:POWer? Max

Actual achievable leveled power depends on frequency.


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both
arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW1 5
source2:power:level:immediate:amplitude maximum
sour:pow 5, "Port 1 Src2"
```

**Query Syntax**

```
SOURce<cnum>:POWer[:LEVel][:IMMediate][:AMPLitude]? [src]
```

**Return Type**

Numeric

**Default**

0 dBm

---

**SOURce<cnum>:POWer[:LEVel]:SLOPe <num>**

*(Read-Write)* Sets the RF power slope value.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<num>`: Slope value in dB/GHz. Choose any integer between -2 and 2 (0 is no slope).

**Examples**

```
SOUR:POW:SLOP 2
source2:power:slope -2
```

**Query Syntax**

```
SOURce<cnum>:POWer[:LEVel]:SLOPe?
```

**Return Type**

Numeric

**Default**

0

---

**SOURce<cnum>:POWer[:LEVel]:SLOPe:STATe <ON | OFF>**

*(Read-Write)* Turns Power Slope ON or OFF.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<ON|OFF>`: ON (or 1) - turns slope ON.
  - OFF (or 0) - turns slope OFF.

**Examples**

```
SOUR:POW:SLOP:STAT ON
source2:power:slope:state off
```

**Query Syntax**

```
SOURce<cnum>:POWer[:LEVel]:SLOPe:STATe?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF
SOURce<cnum>:POWer<port>:MODE <state>, [src]

(Read-Write) Sets the state of PNA source for the specified port.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port number of the PNA. If unspecified, `<port>` is set to 1. To make settings for ports that are not simple numbers, use the [src] argument.
- `<state>` Source state. Choose from:
  - **AUTO** Source power is turned ON when required for a measurement.
  - **ON** Source power is always ON regardless of the measurement.
  - **OFF** Source power is always OFF regardless of the measurement.

While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW:MODE ON
source2:power4:mode OFF
sour:pow:mode on, "Port 1 Src2"
```

**Query Syntax** SOURce<cnum>:POWer<port>:MODE? [src]

**Return Type** Character

**Default** Auto

---

SOURce<cnum>:POWer<port>:PORT:STARt <num>, [src]
(Read-Write) Sets and reads the power sweep start power value for a specific port. This allows uncoupled forward and reverse power sweep ranges. Must also set SENS:SWE:TYPE POWer and SOUR:POW:COUPle OFF.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port number of the PNA. If unspecified, `<port>` is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<num>` Start power in dBm.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, perform a query using MAX, then MIN, as an argument. (SOUR:POW:ATT:AUTO must be set to ON) Example: SOURce:POWer:STARt? MIN

Actual achievable leveled power depends on frequency.


While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

**Examples**

SOUR:POW1:PORT:STAR -15
source2:power:port:start 5, "bal port 1"

**Query Syntax** SOURce<cnum>:POWer<port>:PORT:STARt? [src]

**Return Type** Numeric

**Default** -10 dBm

**SOURce<cnum>:POWer<port>:PORT:STOP <num>, [src]**
(Read-Write) Sets and reads the power sweep stop power value for a specific port. This allows uncoupled forward and reverse power sweep ranges. Must also set SENS:SWE:TYPE POWer and SOUR:POW:COUPle OFF.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Source port number of the PNA. If unspecified, `<port>` is set to 1. To make settings for ports that are not simple numbers, use the `[src]` argument.
- `<num>` Stop power in dBm.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, perform a query using MAX, then MIN, as an argument. (SOUR:POW:ATT:AUTO must be set to ON) Example: SOURce:POWer:STARt? MIN

Actual achievable leveled power depends on frequency.


  While this argument can be used to make settings for ALL ports, it is designed to access ports such as an external source, true mode balanced port, or one of the Source 2 outputs on the 2-port 2-source PNA-X model such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.

**Examples**

SOUR:POW1:PORT:STOP -15  
source2:power:port:stop 5, "bal port 1"

**Query Syntax** SOURce<cnum>:POWer<port>:PORT:STOP? [src]

**Return Type** Numeric

**Default** 0 dBm

---

**SOURce<cnum>:POWer:SPAN <num>**
(Read-Write) Sets the power sweep span power. Must also set:

SENS:SWE:TYPE POWer and SOURce:POWer:CENTer <num>.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<num>` Span power. Actual achievable leveled power depends on frequency.

**Examples**

```plaintext
SOUR:POW:SPAN -15
source2:power:span -7
```

**Query Syntax** SOURce<cnum>:POWer:SPAN?

**Return Type** Numeric

**Default** 0 dBm

---

**SOURce<cnum>:POWer:STARt <num>**

(Read-Write) Sets the power sweep start power for ALL ports being used by the specified channel. Must also set:

SENS:SWE:TYPE POWer and SOURce:POWer:STOP <num>.

To set start power for a specific port, use SOUR:POW:PORT:STARt.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Start power.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, perform a query using MAX, then MIN, as an argument. (SOUR:POW:ATT:AUTO must be set to ON) Example: SOURce:POWer:STARt? MIN

**Examples**

```plaintext
SOUR:POW:STAR -15
source2:power:start -7
```

**Query Syntax** SOURce<cnum>:POWer:STARt?

**Return Type** Numeric

**Default** 0 dBm
**SOURCE<cnum>::POWER::STOP <num>**

(Read-Write) Sets the power sweep stop power for ALL ports being used by the specified channel. Must also set: **SENS:SWE:TYPE POWER** and **SOURCE::POWER::START <num>**.

To set start power for a specific port, use **SOURCE::POWER::PORT::STOP**.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<num>` Stop power.

**Note:** The range of settable power values depends on the PNA model and if source attenuators are installed. To determine the range of values, perform a query using MAX, then MIN, as an argument. (**SOURCE::POWER::ATT::AUTO** must be set to ON)  Example: **SOURCE::POWER::STOP? MAX**

Actual achievable leveled power depends on frequency.

**Examples**

```
SOUR::POW::STOP -15
source2:power:stop -7
```

**Query Syntax**  
**SOURCE<cnum>::POWER::STOP?**  

**Return Type**  
Numeric

**Default**  
0 dBm

Last modified:

- 8-Jul-2011   Edit start and stop sweep commands
- 11-Jan-2011   Many minor edits
- 3-Dec-2010   Added link to source phase
- 16-Sep-2010   Added LSPC
- 17-Jun-2010   Fixed error in receiver mode - reference
- 19-Oct-2009   Fixed many broken links
- 15-Apr-2009   Fixed sour:pow:mode example
- 12-Feb-2009   Added receiver leveling
- 31-Jul-2008   Fix sour:pow unit in example
- 23-May-2008   Added uncoupled power sweep commands
- 25-Oct-2007   Edit test for source and rec attenuators commands
- 27-Jun-2007   Edited wording on Source:Cat?
- 10/18/06   MQQ Added Mode command

2813
Source:Power:Correction Commands

Used to perform source power calibration on internal and external sources.

**Note:** Only ONE Source Power Cal can be performed at a time.

```
SOURce:POWer:CORRection

COLLect
| ABORt
| ACQuire
| AVERage
  | [COUNT]
  | NTOLerance
| DISPlay
  | [STATE]
| FCHeck
  | [STATE]
| ITERation
  | [COUNT]
  | NTOLerance
| METHOD
| SAVE
| SENSor
  | RCFactor
  | [FRANge]
  | SEllect
| TABLe
  | DATA
  | FREQuency
  | LOSS
    | [STATE]
  | POINts?
  | [SEllect]
DATA
| PRIor
LEVel
OFFSet
  | [MAGNitude]
[STATE]
```

Click on a blue keyword to view the command details.

Red commands are superseded.
See Also

- Example program using these commands.
- Template for creating your own Power Meter Driver
- Learn about Source Power Cal
- Synchronizing the PNA and Controller
- SCPI Command Tree

Note: The SOURce:POWer:CORRection:COLLect:ACQuire command, used to step the PNA and read a power meter, cannot be sent over the GPIB unless the power meter is connected to a different GPIB interface. See the alternative methods described in the command details.

**SOURce<cnum>:POWer<port>:CORRection:COLLect:ABORt**

(Write-only) Aborts a source power calibration sweep that is in progress.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Port number to correct for source power. If unspecified, value is set to 1.

**Examples**

SOUR:POW:CORR:COLL:ABOR
source1:power2:correction:collect:abort

**Query Syntax** Not Applicable

**Default** Not Applicable

**SOURce<cnum>:POWer<port>:CORRection:COLLect[:ACQuire] <char>,<id>[,src][,sync]**

**Note:** With PNA Rev. 6.2, a new "id" argument has been added to this command, replacing SOUR:POW:CORR:COLL:METH.

(Write-only) Initiates a source power cal acquisition sweep using the power sensor attached to the specified channel (A or B) on the power meter, using a USB power sensor, or using the specified PNA receiver.

**Note:** This command, when used with a power meter, can NOT be sent over the GPIB unless the power meter is connected to a different GPIB interface or USB. Use one of the following methods to perform this command or its equivalent:

- If present, use the GPIB dedicated controller port.
Connect the power meter to the PNA using a USB / GPIB interface (Agilent 82357A).

SCPI programming of the PNA using a LAN Client interface (see example).

Send SCPI commands through the COM interface using the SCPI String Parser object.

Directly control the Power Meter and PNA to step frequency; then acquire and store the Power reading. (see example).

Configure the Power Meter/Sensor as a PMAR Device. Learn how, See SCPI commands.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Port number to correct for source power. If unspecified, value is set to 1.
- `<char>` Acquisition Choose from:
  - **PMETer** - Power Meter is used for all readings.
  - **PMReceiver** - Power meter for the first iteration; then use the reference receiver for remaining readings if necessary (same as “fast iteration” box checked on dialog box)
  - **RECeiver** - Use PNA measurement receiver for all readings.
- `<id>` String (Not case sensitive). The power sensor or PNA receiver to use for measuring power.
  
  For **PMETer** or **PMRECeiver**, choose from:
  
  - "ASENSOR" or "BSENSOR". For U series USB sensors, always specify "ASENSOR"

  For **RECeiver**, choose from:
  
  - Any PNA receiver to acquire readings using physical or logical receiver notation.
  - Any configured PMAR device name. Learn more about PMAR Devices. See PMAR commands.


While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, `[src]` takes priority.
If this argument is specified, must also specify [src].

Choose from:

- SYNChronous - Blocks SCPI commands during standard measurement (default behavior).
- ASYNchnous - Does NOT block SCPI commands during standard measurement.

Examples

SOUR:POW:CORR:COLL PMET,"ASEN","Port 1",ASYN  'acquires power meter readings using the A sensor, source port 1, asynchronous.

source1:power2:correction:collect:acquire receiver,"a1"  'acquires source cal readings using the reference receiver for port 1.

Query Syntax  Not Applicable

Default  Not Applicable

SOURce:POWer:CORRection:COLLect:AVERage[:COUNt] <num>

(Read-Write) This command, along with SOUR:POW:CORR:COLL:AVER:NTOLerance, allows for settling of the power sensor READINGS.

Specifies the maximum number of power readings that are taken at each stimulus point to allow for measurement settling. Each reading is averaged with the previous readings at that stimulus point.

When this average meets the Average:NTOLerance value or this number of readings has been made, the average is returned as the valid reading.

This setting is not necessary when using a PNA receiver (SOUR:POW:CORR:COLL REC) to make the measurement.

Learn more.

Parameters

- <num> Maximum number of readings to make to allow for settling. Choose any number between 3 and 25.

Examples

SOUR:POW:CORR:COLL:AVER 2
source:power:correction:collect:average:count 3

Query Syntax  SOURce:POWer:CORRection:COLLect:AVERage[:COUNt]?

Return Type  Numeric
SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance <num>

(Read-Write) This command, along with SOUR:POW:CORR:COLL:AVER:COUNT, allows for settling of the power sensor READINGS.

Each power reading is averaged with the previous readings at each stimulus point. When the average meets this nominal tolerance value or the max number of readings has been made, the average is returned as the valid reading.

This setting is not necessary when using a PNA receiver (SOUR:POW:CORR:COLL REC) to make the measurement.

Learn more.

Parameters

<num>  Power measurement settling tolerance value in dB. Choose any number between 0 and 5.

Examples

SOUR:POW:CORR:COLL:AVER:NTOL .05
source1:power2:correction:collect:average:ntolerance .003

Query Syntax

SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance?

Return Type  Numeric

Default  .050 dBm

SOURce<cnum>:POWer:CORRection:COLLect:DISPlay[:STATe] <ON | OFF>

(Read-Write) Enables and disables the display of power readings on the PNA screen. Send this command BEFORE you begin a source power cal acquisition. After the source power cal data is acquired, this setting is reset to ON.

Parameters

<cnun>  Any existing channel number. If unspecified, value is set to 1

<ON|OFF>  

ON (1)  Source power calibration dialog box is shown on the PNA screen. Power readings are plotted against the Tolerance value as limit lines.

OFF (0) - Source power calibration dialog box is NOT shown on the PNA screen.

Examples

SOUR:POW:CORR:COLL:DISP ON
source1:power2:correction:collect:display:state off
**Query Syntax**
```
SOURce:POWer:CORRection:COLLect:DISPlay[:STATe]?
```

**Return Type**
Boolean (1 = ON, 0 = OFF)

**Default**
ON (1)

**SOURce<cnum>:POWer:CORRection:COLLect:FCHeck[:STATe] <ON | OFF>**
(Read-Write) Enables and disables frequency checking of source power cal acquisition sweeps. ONLY use when you have more than one power sensor.

**Parameters**
- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<ON|OFF>**
  - **ON (1)** turns source power cal frequency checking ON. A requested acquisition will only succeed for those frequency points which fall within a frequency range specified for the power sensor being used. An acquisition will pause in mid-sweep if the frequency is about to exceed the maximum frequency limit specified for that sensor. When the sweep is paused in this manner, a sensor connected to the other channel input of the power meter can be connected to the measurement port in place of the previous sensor, and used to complete the sweep. However, the maximum frequency specified for the second sensor would need to be sufficient for the sweep to complete. Frequency limits are specified using the `SOUR:POW:CORR:COLL:SEN` command.
  - **OFF (0)** - turns source power cal frequency checking OFF. An acquisition will use just one power sensor for the entire sweep, regardless of frequency.

**Examples**
```
SOUR:POW:CORR:COLL:FCH ON
source1:power2:correction:collect:fcheck:state off
```

---

**SOURce:POWer:CORRection:COLLect:ITERation[:COUNt] <num>**
This command, along with `SOUR:POW:CORR:COLL:ITER:NTOL`, describes the number of adjustments to make to the source power.

Sets the maximum number of readings to take at each data point for iterating the source power. Power READINGS (performed by `SOUR:POW:CORR:COLL:AVER:COUNT` and `SOUR:POW:CORR:COLL:AVER:NTOLerance`) will continue to be made, and source power adjusted, until the measurement is within the iteration tolerance value or this max number of measurements has been met. The last value is the valid measurement for that data point.

**Parameters**

- `<num>` Maximum number of readings. Choose any number between 1 and 100.

**Examples**

```
SOUR:POW:CORR:COLL:ITER 2
source:power:correction:collect:iteration 3
```

**Query Syntax**

`SOURce:POWer:CORRection:COLLect:ITERation[:COUNt]?
```

**Return Type** Numeric

**Default** 1

---

This command, along with `SOUR:POW:CORR:COLL:ITER:COUNt`, describes the number of adjustments to make to the source power.

Sets the maximum desired deviation from the sum of the test port power and the offset value. Power READINGS (performed by `SOUR:POW:CORR:COLL:AVER:COUNT` and `SOUR:POW:CORR:COLL:AVER:NTOLerance`) will continue to be made, and source power adjusted, until a measurement is within this tolerance value or the max number of measurements has been met. The last value is the valid measurement for that data point.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<port>` Port number to correct for source power. If unspecified, value is set to 1.
- `<num>` Tolerance value in dBm. Choose any number between 0 and 5

**Examples**

```
SOUR:POW:CORR:COLL:ITER:NTOL .005
source:power:correction:collect:iteration:ntolerance .1
```

**Query Syntax**

`SOURce:POWer:CORRection:COLLect:ITERation:NTOLerance?`
**Return Type**  Numeric

**Default**  .05

---

**SOURce<cnum>:POWer<port>:CORRection:COLLect:METHod <char> Superseded**

This command is replaced with **SOUR:POW:CORR:COLLect[:ACQuire]** which now specifies the method and the device. The only parameter required by that command was either ASEN sor or BSEN sor which are still supported but not documented.

*(Read-Write)* Selects the calibration method to be used for the source power cal acquisition.

**Parameters**

- `<cnum>`  Any existing channel number. If unspecified, value is set to 1
- `<port>`  Port number to correct for source power. If unspecified, value is set to 1.
- `<char>`  Choose from:
  - **NONE** - No Cal method
  - **PMETer** - Power Meter is used for all readings. (same as "fast iteration" box not checked on dialog box)
  - **PMReceiver** - Power meter for the first iteration; then use the reference receiver for remaining readings if necessary (same as "fast iteration" box checked on dialog box)

**Examples**  SOUR:POW:CORR:COLL:METH PMET

source1:power2:correction:collect:method pmreceiver

**Query Syntax**  SOURce:POWer:CORRection:COLLect:METHod?

**Return Type**  Character

**Default**  NONE

---

**SOURce:POWer:CORRection:COLLect:SAVE [<RREC>]**
(Write-only) Applies the array of correction values after a source power calibration sweep has completed. The source power correction will then be active on the specified source port for channel <cnum>. This command does NOT save the correction values. To save correction values, save an instrument / calibration state (*.cst file) after performing a source power cal.

Parameters

<RREC> Optional argument.

**RRECeiver** In addition to a source Power Cal, perform a calibration of the reference receiver used in the measurement. ONLY the Reference Receiver calibration is then saved to a Cal Set or Cal Register as specified by the current setting of SENS:CORR:PREF:CSET:SAVE.

This argument only applies to standard S-parameter channels.

**Examples**


**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SOURce<cnum>:POWer:CORRection:COLLect:<pmChan>SENsor[:FRANge] <num1>,<num2>**

(Read-Write) Specifies the frequency range over which the power sensors connected to the specified channels (A and B) of the power meter can be used (minimum frequency, maximum frequency). If the power meter has only a single channel, that channel is considered channel A.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1

<pmChan> Power Meter channel. Choose from:

A - Channel A

B - Channel B

<num1> Minimum frequency for the sensor. If a frequency unit is not specified, Hz is assumed. No limits are placed on this value.

<num2> Maximum frequency for the sensor. If a frequency unit is not specified, Hz is assumed. No limits are placed on this value.

**Examples**

SOUR:POW:CORR:COLL:ASEN 100E3, 3E9

source1:power2:correction:collect:bsensor:frange 10 MHz, 18 GHz
**Query Syntax**

SOURce:POWer:CORRection:COLLect:ASENsor[:FRANge]?

SOURce:POWer:CORRection:COLLect:BSENsor[:FRANge]?

**Return Type**

Numeric

**Default**

0,0

---

**SOURce<cnm>:POWer:CORRection:COLLect:<pmChan>SENsor:RCFactor <num>**

(Read-Write) Specifies the reference cal factor for the power sensor connected to channel A or B of the power meter. If the power meter has only a single channel, that channel is considered channel A.

**Note:** If the sensor connected to the specified channel of the power meter contains cal factors in EPROM (such as the Agilent E-series power sensors), those will be the cal factors used during the calibration sweep. The reference cal factor value associated with this command, and any cal factors entered into the PNA for that sensor channel, will not be used.

**Parameters**

- `<cnm>` Any existing channel number. If unspecified, value is set to 1
- `<pmChan>` Power Meter channel. Choose from:
  - A - Channel A
  - B - Channel B
- `<num>` Reference cal factor in percent. Choose any number between 1 and 150.

**Examples**

```
SOUR:POW:COR:COLL:ASEN:RCF 98.7
source1:power2:correction:collect:bsensor:rcfactor 105
```

**Query Syntax**

SOURce:POWeR:CORRection:COLLect:ASENsor:RCFactor?

SOURce:POWeR:CORRection:COLLect:BSENsor:RCFactor?

**Return Type**

Numeric

**Default**

100

---

**SOURce<cnm>:POWeR:CORRection:COLLect:<pmChan>SENsor:SELect**
Sets and returns the power sensor channel (A or B) to be used. This performs the same function as the Use this sensor only checkbox in the Power Sensor Settings dialog.

**Note:** This write portion of this command is only necessary when performing an SMC calibration.

**Parameters**

- `<cnum>`: Any existing channel number. If unspecified, value is set to 1
- `<pmChan>`: Power Meter channel. Choose from:
  - **A**: Channel A
  - **B**: Channel B

**Examples**

```plaintext
SOUR:POW:CORR:COLL:<pmChan>SEN:SEL 'Write
source1:power2:correction:collect:bsensor:select? 1e9 'Read
```

**Query Syntax**

- SOURce:POWer:CORRection:COLLect:ASENsor:SELect? <Frequency>
- SOURce:POWer:CORRection:COLLect:BSENsor:SELect? <Frequency>

Returns a boolean 1 or 0 (ON or OFF) indicating whether the sensor is to be used at the specified frequency.

If frequency checking is OFF, then the `<Frequency>` parameter is ignored. The query returns if the sensor is selected for ALL frequencies.

**Return Type**

Numeric

**Default**

Not Applicable

---

```
SOURce<cnum>:POWer:CORRection:COLLect:TABLE:DATA <data>
```
(Read-Write) Read or write data into the selected table. Each table can contain up to 100 segments. Use SOUR:POW:CORR:COLL:TABL:SESelect to select a table.

- When the power sensor table is selected, the data is interpreted as cal factors in percent.
- When the loss table is selected, POSITIVE values in dB are interpreted as LOSS. To compensate for gain, use negative values. Learn more about Power Loss Compensation.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<data>` Data to write into the selected table.

**Examples**

```bash
SOURce:POWer:CORRection:COLLect:TABLE:DATA 0.12, 0.34, 0.56
```

**Query Syntax**

SOURce<cnum>:POWer:CORRection:COLLect:TABLE:DATA?

If the selected table is currently empty, no data is returned.

**Return Type**

Numeric - one number per table segment.

**Default**

Not Applicable

---

**SOURce<cnum>:POWer:CORRection:COLLect:TABLE:FREQuency <data>**

(Read-Write) Read or write frequency values for the selected table (cal factor table for a power sensor, or the loss compensation table). Use SOUR:POW:CORR:COLL:TABL:SESelect to select a table.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1
- `<data>` Frequency data to write into the selected table.

**Examples**

```bash
SOURce:POWer:CORRection:COLLect:TABLE:FREQuency 10E6, 1.5E9, 9E9
```

**Query Syntax**

SOURce<cnum>:POWer:CORRection:COLLect:TABLE:FREQuency?

If the selected table is currently empty, no data is returned.

**Return Type**

Numeric - one number per table segment

**Default**

Not Applicable

---

**SOURce<cnum>:POWer:CORRection:COLLect:TABLE:LOSS[:STATe] <ON | OFF>**
(Read-Write) Indicates whether or not to adjust the power readings using the values in the loss table during a source power cal sweep. Learn more about Power Loss Compensation.

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<ON|OFF>** ON (or 1) - turns use of the loss table ON.
  
  OFF (or 0) - turns use of the loss table OFF.

**Examples**

```
SOUR:POW:CORR:COLL:TABL:LOSS ON
source1:power2:correction:collect:table:loss:state off
```

**Query Syntax**

```
SOURce:POWer:CORRection:COLLect:TABLe:LOSS[:STATe]?
```

**Return Type**

Boolean (1 = ON, 0 = OFF)

**Default**

OFF (0)

---

### SOURce<cnum>:POWer:CORRection:COLLect:TABLe:POINts?

(Read-only) Returns the number of segments that are currently in the selected table.

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1

**Examples**

```
SOUR:POW:CORR:COLL:TABL:POIN?
source1:power2:correction:collect:table:points?
```

**Return Type**

Numeric

**Default**

0

---

### SOURce<cnum>:POWer:CORRection:COLLect:TABLE[:SELect] <char>

2827
(Read-Write) Selects which table you want to write to or read from. Read or write using
SOURce:POWer:CORRection:COLLect:TAble:FREQuency and
SOURce:POWer:CORRection:COLLect:TAble:DATA

Parameters

<cnun> Any existing channel number. If unspecified, value is set to 1
<char> Choose from:

NONE - No table selected

ASEnsor - Cal Factor table for Power Sensor A

BSeNsor - Cal Factor table for Power Sensor B

LOSS - Loss compensation table

Examples

SOUR:POW:CORR:COLL:TabL ASEN
source1:power2:correction:collect:table:select bsensor

Query Syntax

SOURce:POWer:CORRection:COLLect:TAble[>:SELect]?

Return Type

Character

Default

NONE

SOURce<cnun>:POWer<port>:CORRection:DATA <data>[,src]

(Read-Write) Writes and reads source power calibration data.

The effect from this command on the channel is immediate. Do NOT send
SOUR:POW:CORR:COLL:SAVE after this command as it may invalidate the uploaded data.

When querying source power calibration data, if no source power cal data exists for the specified
channel and source port, then no data is returned.

If a change in the instrument state causes interpolation and/or extrapolation of the source power cal,
the correction data associated with this command correspond to the new instrument state (interpolated
and/or extrapolated data).

If the channel is sweeping the source backwards, then the first data point is the highest frequency
value; the last data point is the lowest. Use the SENS:X:VALues? command to return the X-axis
values in the displayed order.

Parameters

<cnun> Any existing channel number. If unspecified, value is set to 1
Port number to correct for source power. If unspecified, value is set to 1.

Correction Data


While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

Examples SOURce1:POWer2:CORRection:DATA 0.12, -0.34, 0.56

Query Syntax SOURce<cnum>:POWer<port>:CORRection:DATA? [src]

Return Type Depends on FORMat:DATA command

Default Not Applicable

SOURce<cnum>:POWer<port>:CORRection:DATA:PRIor <data>,[src]

(Read-Write) Writes and reads power correction values from the previous iteration of the source power cal. Data for which the first power meter reading were within the tolerance limit, the ‘prior’ correction value is 0.

In all other respects, this command is the same as SOUR:POW:CORR:DATA.

This command can be used to determine the final power reading at each point of the power cal, for a cal that did not pass tolerance limits. The formula for determining the power reading (in dB):

Power reading = Target power at the source port + specified power cal offset value + ‘prior’ iteration corr value – actual power corr value.

The "actual" value in this equation is returned with SOUR:POW:CORR:DATA?

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1

<port> Port number to correct for source power. If unspecified, value is set to 1.

<data> Correction Data


While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise,
the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

SOURc1:POWer2:CORRection:DATA:PRIor 0.12, -0.34, 0.56

**Query Syntax**

SOURce<cnum>:POWer<port>:CORRection:DATA:PRIor? [src]

**Return Type**

Depends on FORMat:DATA command

**Default**

Not Applicable

**SOURce<cnum>:POWer<port>:CORRection:LEVel[:AMPLitude] <num>[,src]**

*(Read-Write)* Specifies the power level that is expected at the desired reference plane (DUT input or output). This is not used for **segment sweep with independent power levels** or **power sweeps**.

**Note:** Although this command still works, it is recommended that you specify cal power by setting the **test port power** and **offset** value.

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<port>** Port number to correct for source power. If unspecified, value is set to 1.
- **<num>** Cal power level in dBm. Because this could potentially be at the output of a device-under-test, no limits are placed on this value here. It is realistically limited by the specifications of the device (power sensor) that will be used for measuring the power. The power delivered to the PNA receiver must never exceed PNA specifications for the receiver!
- **[src]** **String.** (NOT case sensitive). Source port. Optional. Use SOUR:CAT? to return a list of valid port names.

While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

SOUR:POW:CORR:LEV 10
sourcel:power2:correction:level:amplitude 0 dbm

**Query Syntax**

SOURce:POWer:CORRection:LEVel[:AMPLitude]? [src]

**Return Type**

Numeric

**Default**

0 dBm

**SOURce<cnum>:POWer<port>:CORRection:OFFSet[:MAGNitude] <num>[,src]**
Sets or returns a power level offset from the PNA test port power. This can be a gain or loss value (in dB) to account for components you connect between the source and the reference plane of your measurement. For example, specify 10 dB to account for a 10 dB amplifier at the input of your DUT.

Cal power is the sum of the test port power setting and this offset value. Following the calibration, the PNA power readouts are adjusted to the cal power.

**Parameters**

- **<cnum>** Any existing channel number. If unspecified, value is set to 1
- **<port>** Port number to correct for source power. If unspecified, value is set to 1.
- **<num>** Gain or loss value in dB. Choose a value between -200 and 200

While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise, the <port> argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW:CORR:OFFS 10
source1:power2:correction:offset:magnitude -3
```

**Query Syntax** SOURce:POWer:CORRection:OFFSet[:MAGNitude]? [src]

**Return Type** Numeric

**Default** 0 dB

```
SOURce<cnum>:POWer<port>:CORRection[:STATe] <bool>[,src]
```
(Read-Write) Enables and disables source power correction for the specified port on the specified channel.

**Parameters**

- `<cnum>` Any existing channel number. If unspecified, value is set to 1.
- `<port>` Port number to correct for source power. If unspecified, value is set to 1.
- `<bool>` ON (or 1) turns source power correction ON.
  
  OFF (or 0) - turns source power correction OFF.
  

While this argument can be used to make settings for ALL ports, it is designed to access ports that are not simple numbers, such as "Port 1 Src2". Otherwise, the `<port>` argument performs the same function. If both arguments are specified, [src] takes priority.

**Examples**

```
SOUR:POW:CORR ON
source1:power2:correction:state off
```

**Query Syntax** SOURce:POWer:CORRection[:STATe]? [src]

**Return Type** Boolean (1 = ON, 0 = OFF)

**Default** OFF (0)

---

**Last modified:**

- 10-Feb-2011 Added PMAR to ACQuire command
- 25-Oct-2010 Added immediate note to DATA command
- 17-Mar-2010 Added PRIor command (9.2)
- 24-Feb-2009 Added sync to ACQ, removed old arguments
- 2-Jun-2008 Clarified Loss compensation data
- 17-Apr-2007 Removed ch and port arguments for 4 settling and accuracy commands.
- 12-Sept-2006 MQ Modified for receiver only SPC
Status Register Commands

The status registers enable you to query the state of selected events that occur in the analyzer.

Note: This documentation requires familiarity with the "Standard Status Data Structure - Register Model" as defined in IEEE Std 488.2-1992. Also, first read Learn about Status Registers

Click on a blue keyword to view the command details.

See Also

- Example Programs
- Learn about Status Registers
- Synchronizing the PNA and Controller
- SCPI Command Tree

Notes:

- Any bit not shown in the registers is not used but may be reserved for future use.
- The SCPIStringParser can NOT be used with SCPI Status Reporting. However, the *OPC? will work.
**Status Byte Register**

Summarizes the states of the other registers and monitors the PNA output queue. It also generates service requests. The Enable register is called the Service Request Enable Register.

<table>
<thead>
<tr>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*CLS</td>
<td>Clears ALL &quot;event&quot; registers and the SCPI Error / Event queue. The corresponding ENABLE registers are unaffected.</td>
</tr>
<tr>
<td>*STB?</td>
<td>Reads the value of the analyzer's status byte. The byte remains after being read.</td>
</tr>
<tr>
<td>*SRE?</td>
<td>Reads the current state of the Service Request Enable Register.</td>
</tr>
<tr>
<td>*SRE &lt;num&gt;</td>
<td>Sets bits in the Service Request Enable register. The current setting of the SRE register is stored in non-volatile memory. Use *SRE 0 to clear the enable.</td>
</tr>
</tbody>
</table>

<num> Combined value of the weights for bits to be set.

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4</td>
<td>Error / Event queue Summary (EAV)</td>
<td>the Error / Event queue is not empty. To read the error message, use SYST:ERR?</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>Questionable Register Summary</td>
<td>any enabled bit in the questionable event status register is set to 1</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>Message Available</td>
<td>the output queue is not empty</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>Standard Event Register Summary</td>
<td>any enabled bit in the standard event status register is set to 1</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>Request Service</td>
<td>any of the other bits in the status byte register is set to 1 (used to alert the controller of a service request within the analyzer). This bit cannot be disabled.</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>Operation Register Summary</td>
<td>any enabled bit in the standard operation event status register is set to 1</td>
</tr>
</tbody>
</table>
**STATus:QUESTionable:<keyword>**

Summarizes conditions that monitor the quality of measurement data.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:CONDition?</td>
<td>STAT:QUES:COND?</td>
</tr>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:QUES:ENAB 1024</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:QUES?</td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td>STAT:QUES:NTR 1024</td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td>STAT:QUES:PTR 0</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>512</td>
<td>Integrity Reg summary</td>
<td>any enabled bit in the <strong>Integrity</strong> event register is set to 1</td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>Limit Registers summary</td>
<td>any enabled bit in the <strong>Limit</strong> event registers is set to 1</td>
</tr>
<tr>
<td>11</td>
<td>2048</td>
<td>Define Registers summary</td>
<td>any enabled bit in the <strong>Define</strong> event registers is set to 1</td>
</tr>
</tbody>
</table>

---

**STATus:QUESTionable:INTegrity <keyword>**

Summarizes conditions in the Measurement Integrity register.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:CONDition?</td>
<td>STAT:QUES:INT:COND?</td>
</tr>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:QUES:INT:ENAB 1024</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:QUES:INT?</td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td>STAT:QUES:INT:NTR 1024</td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td>STAT:QUES:INT:PTR 0</td>
</tr>
<tr>
<td>Bit</td>
<td>Weight</td>
</tr>
<tr>
<td>-----</td>
<td>--------</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
</tr>
</tbody>
</table>

**STATus:QUEStionable:INTegrity:HARDware<keyword>**

Monitors the status of hardware failures.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>[:EVENt]?</td>
<td>STAT:QUES:INT:HARD?</td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td>STAT:QUES:INT:HARD:NTR 1024</td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td>STAT:QUES:INT:HARD:PTR 0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>Phase Unlock</td>
<td>the source has lost phaselock, possibly caused by a reference channel open or a hardware failure.</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>Unleveled</td>
<td>the source power is unleveled. This could be caused by a source set for more power than it can deliver at the tuned frequency. Or it could be caused by a hardware failure.</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>Not used</td>
<td>N/A</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>EE Write Failed</td>
<td>an attempted write to the EEPROM has failed, possibly caused by a hardware failure.</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>Not used</td>
<td>N/A</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>Ramp Cal Failed</td>
<td>the analyzer was unable to calibrate the analog ramp generator due to a possible hardware failure.</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>Not used</td>
<td>N/A</td>
</tr>
</tbody>
</table>
**STATus:QUEStionable:INTegrity:MEASurement<n> <keyword>**

**Note:** This register can be used ONLY with standard S-parameter measurements.

Monitors the lag between changing a channel setting and when the data is ready to query.

When you change the channel state (start/stop freq, bandwidth, and so forth), then the questionable bit for that channel is set. This indicates that your desired channel state does not yet match the data you would get if querying a data trace. When the next sweep is complete (without aborting in the middle), and the data trace matches the channel state that produced it, the bit is cleared for that channel.

<n> Measurement register number. Choose from 1 to 3

<keyword>

| [:EVENt]? | STAT:QUES:INT:MEAS3? |
| :NTRansition <bits> | STAT:QUES:INT:MEAS2:NTR 1024 |
| :PTRansition <bits> | STAT:QUES:INT:MEAS1:PTR 0 |

---

### Measurement Register <n>

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>1</td>
<td></td>
<td></td>
<td>Summary from Meas Reg 3 a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>2</td>
<td>15</td>
<td>29</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>3</td>
<td>16</td>
<td>30</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>4</td>
<td>17</td>
<td>31</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>5</td>
<td>18</td>
<td>32</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>6</td>
<td>19</td>
<td></td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>7</td>
<td>20</td>
<td></td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>8</td>
<td>21</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>-----</td>
<td>---</td>
<td>----</td>
<td>-----------------------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>256</td>
<td>9</td>
<td>22</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>512</td>
<td>10</td>
<td>23</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>11</td>
<td>24</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>2048</td>
<td>12</td>
<td>25</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>4096</td>
<td>13</td>
<td>26</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>8192</td>
<td>14</td>
<td>27</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>16384</td>
<td>Summary from Meas Reg 2</td>
<td>28</td>
<td>a setting change on this channel has occurred and the data does not yet reflect that change.</td>
<td></td>
</tr>
</tbody>
</table>

**STATus:QUESTionable:LI Mit<n> <keyword>**

Monitors and summarizes the status of limit line failures. When a trace fails, the representative bit is set to 1.

Bit 0 is used to summarize failures in the registers that follow. For example, Limit Register 3, bit 0, summarizes the failures from registers 4 through 42.

All enable bits are set to 1 by default.

To find the measurement number, use **Calc:Par:Mnum**

<n> Limit register: Choose from 1 to 42.

<keyword> Example

:CONDition?

STAT:QUES:LI M4:COND?

:ENABle <bits>

STAT:QUES:LI M1:ENAB 1024

[:EVENt]?

STAT:QUES:LI M3?

:NTRansition <bits>

STAT:QUES:LI M2:NTR 1024

:NTRansition?

STAT:QUES:LI M1:NTR?
| Bit | Weight | 1  | 2  | 3  | 4  | 5  | 6  | 7  | 8  | ... | 41 | 42 | Bit is set to 1 when the following conditions exist:
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>2-</td>
<td>3-</td>
<td>4-</td>
<td>5-</td>
<td>6-</td>
<td>7-</td>
<td>8-</td>
<td>9-</td>
<td>...</td>
<td>42</td>
<td>42</td>
<td>Summary Bit - If any bit from that register fails, it propagates to the previous register, bit 0.</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>1</td>
<td>15</td>
<td>29</td>
<td>43</td>
<td>57</td>
<td>71</td>
<td>85</td>
<td>99</td>
<td>...</td>
<td>561</td>
<td>575</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>2</td>
<td>16</td>
<td>30</td>
<td>44</td>
<td>58</td>
<td>72</td>
<td>86</td>
<td>100</td>
<td>...</td>
<td>562</td>
<td>576</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>3</td>
<td>17</td>
<td>31</td>
<td>45</td>
<td>59</td>
<td>73</td>
<td>87</td>
<td>101</td>
<td>...</td>
<td>563</td>
<td>577</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>4</td>
<td>18</td>
<td>32</td>
<td>46</td>
<td>60</td>
<td>74</td>
<td>88</td>
<td>102</td>
<td>...</td>
<td>564</td>
<td>578</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>5</td>
<td>19</td>
<td>33</td>
<td>47</td>
<td>61</td>
<td>75</td>
<td>89</td>
<td>103</td>
<td>...</td>
<td>565</td>
<td>579</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>6</td>
<td>20</td>
<td>34</td>
<td>48</td>
<td>62</td>
<td>76</td>
<td>90</td>
<td>104</td>
<td>...</td>
<td>566</td>
<td>580</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>7</td>
<td>21</td>
<td>35</td>
<td>49</td>
<td>63</td>
<td>77</td>
<td>91</td>
<td>105</td>
<td>...</td>
<td>567</td>
<td>--</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>8</td>
<td>256</td>
<td>8</td>
<td>22</td>
<td>36</td>
<td>50</td>
<td>64</td>
<td>78</td>
<td>92</td>
<td>106</td>
<td>...</td>
<td>568</td>
<td>--</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>9</td>
<td>512</td>
<td>9</td>
<td>23</td>
<td>37</td>
<td>51</td>
<td>65</td>
<td>79</td>
<td>93</td>
<td>107</td>
<td>...</td>
<td>569</td>
<td>--</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>10</td>
<td>24</td>
<td>38</td>
<td>52</td>
<td>66</td>
<td>80</td>
<td>94</td>
<td>108</td>
<td>...</td>
<td>570</td>
<td>--</td>
<td>any point on trace fails the limit test</td>
</tr>
<tr>
<td>11</td>
<td>2048</td>
<td>11</td>
<td>25</td>
<td>39</td>
<td>53</td>
<td>67</td>
<td>81</td>
<td>95</td>
<td>109</td>
<td>...</td>
<td>571</td>
<td>--</td>
<td>any point on trace fails the limit test</td>
</tr>
</tbody>
</table>
To determine Register, Bit number, and Weight for trace numbers between 113 and 560 (not shown in the above table) use the following calculations.

The limit status for trace numbers higher than 580 can NOT be tracked.

The following example calculates the Register, Bit number, and Bit Weight for trace # 400:

- To determine Limit **Register** number, use $$((\text{Trace} \# - 1) / 14) + 1$$.
- To determine Limit **Bit Number**, use the **remainder** +1 of the above calculation.
- $$((400-1)/14) + 1 = \text{Register}# \ r+1\text{Bit}$$
  - $$399/14 = 28 r7$$
  - $$28+1= \text{Register} 29$$
  - $$7+1= \text{Bit number} 8$$
- To determine Limit **Bit Weight**: Use above table. For example: Bit 8 = 256

---

**STATus:QUESTionable:DEFine<keyword>**

Summarizes conditions in the Questionable:Define:User<1|2|3> event registers.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:CONDition?</td>
<td>STAT:QUES:DEF:COND?</td>
</tr>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:QUES:DEF:ENAB 1024</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:QUES:DEF?</td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td>STAT:QUES:DEF:NTR 1024</td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td>STAT:QUES:DEF:PTR 0</td>
</tr>
</tbody>
</table>
### Bit | Weight | Description | Bit is set to 1 when the following conditions exist:
--- | --- | --- | ---
1 | 2 | USER1 | any bit in the USER1 event register is set to 1
2 | 4 | USER2 | any bit in the USER2 event register is set to 1
3 | 8 | USER3 | any bit in the USER3 event register is set to 1

#### STATus:QUEStionable:DEFine:USER<1|2|3><keyword>

Monitors conditions that you define and map in any of the three QUES:DEF:USER event registers.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:QUES:DEF:USER1:ENABle 1024</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:QUES:DEF:USER1?</td>
</tr>
<tr>
<td>:MAP &lt;bit&gt;,&lt;error&gt;</td>
<td>STAT:QUES:DEF:USER1:MAP 0,-113 'when error -113 occurs, bit 0 in USER1 will set to 1.'</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>8</td>
<td>256</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>9</td>
<td>512</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>11</td>
<td>2048</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>12</td>
<td>4096</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>13</td>
<td>8192</td>
<td>for user</td>
<td>user defined</td>
</tr>
<tr>
<td>14</td>
<td>16384</td>
<td>for user</td>
<td>user defined</td>
</tr>
</tbody>
</table>


**Standard Event Status Register**

Monitors "standard" events that occur in the analyzer. This register can only be cleared by:

- a Clear Command (*CLS).
- reading the Standard Enable Status Register (*ESE?).
- a power-on transition. The analyzer clears the register and then records any transitions that occur, including setting the Power On bit (7).

<table>
<thead>
<tr>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*ESE?</td>
<td>Reads the settings of the standard event ENABLE register.</td>
</tr>
<tr>
<td>*ESE &lt;bits&gt;</td>
<td>Sets bits in the standard event ENABLE register. The current setting is saved in non-volatile memory.</td>
</tr>
<tr>
<td>&lt;bits&gt;</td>
<td>The sum of weighted bits in the register. Use *ESE 0 to clear the enable register.</td>
</tr>
<tr>
<td>*ESR?</td>
<td>Reads and clears the EVENT settings in the Standard Event Status register.</td>
</tr>
<tr>
<td>*OPC</td>
<td>Sets bit 0 when the overlapped command is complete. (see Understanding Command Synchronization / OPC).</td>
</tr>
<tr>
<td>*OPC?</td>
<td>Operation complete query - read the Operation Complete bit (0).</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>Operation Complete</td>
<td>the two following events occur <strong>in order:</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1. the *OPC command is sent to the analyzer</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2. the analyzer completes all pending overlapped commands</td>
</tr>
<tr>
<td>1</td>
<td>NA</td>
<td>Request Control</td>
<td>Not Supported - the analyzer application is not configured to control GPIB operation</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>Query Error</td>
<td>a query error is detected indicating:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>- an attempt to read data from the output queue when no data was present <strong>OR</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>- data in the output queue was lost, as in an overflow</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>Execution Error</td>
<td>an execution error is detected indicating:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>- a &lt;PROGRAM DATA&gt; element was outside the legal range or inconsistent with the operation of the analyzer <strong>OR</strong></td>
</tr>
</tbody>
</table>
- the analyzer could not execute a valid command due to some internal condition

<table>
<thead>
<tr>
<th>Command Error</th>
<th>a command error is detected indicating that the analyzer received a command that:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• did not follow proper syntax</td>
</tr>
<tr>
<td></td>
<td>• was misspelled</td>
</tr>
<tr>
<td></td>
<td>• was an optional command it does not implement</td>
</tr>
</tbody>
</table>

7 128 Power ON
Power to the analyzer has been turned OFF and then ON since the last time this register was read.

**STATus:OPERation<keyword>**
Summarizes conditions in the Averaging and Operation:Define:User<1|2|3> event registers.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:CONDition?</td>
<td><strong>STAT:OPER:COND?</strong></td>
</tr>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td><strong>STAT:OPER:ENAB 1024</strong></td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td><strong>STAT:OPER?</strong></td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td><strong>STAT:OPER:NTR 1024</strong></td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td><strong>STAT:OPER:PTR 0</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>256</td>
<td>Averaging summary</td>
<td>either enabled bit in the <strong>Averaging summary</strong> event register is set to 1</td>
</tr>
<tr>
<td>9</td>
<td>512</td>
<td>User Defined summary</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>Device summary</td>
<td>either enabled bit in the <strong>Device summary</strong> event register is set to 1</td>
</tr>
</tbody>
</table>
STATus:OPERation:AVERaging<n> <keyword>

Monitors and summarizes the status of Averaging on traces 1 to 580. When averaging for a trace is complete, the representative bit is set to 1.

Bit 0 is used to summarize the status in the registers that follow. For example, Average Register 3, bit 0, summarizes the status from registers 4 through 42.

All enable bits are set to 1 by default.

To find the measurement number, use Calc:Par:Mnum.

<n> Averaging Register. Choose from 1 to 42

Example

:CONDition? STAT:OPER:AVer1:COND?
:ENABle <bits> STAT:OPER:AVer1:ENAB 1024
[:EVENt]? STAT:OPER:AVer1?
:NTRansition <bits> STAT:OPER:AVer1:NTR 1024
:PTRansition <bits> STAT:OPER:AVer1:PTR 0

<table>
<thead>
<tr>
<th>Averaging Register &lt;n&gt;</th>
<th>Bit</th>
<th>Weight</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>...</th>
<th>41</th>
<th>42</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bit</td>
<td></td>
<td></td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>...</td>
<td>41</td>
<td>42</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td></td>
<td>2-42</td>
<td>3-42</td>
<td>4-42</td>
<td>5-42</td>
<td>6-42</td>
<td>7-42</td>
<td>8-42</td>
<td>...</td>
<td>42</td>
<td>--</td>
<td></td>
</tr>
<tr>
<td>Bit is set to 1 when the following conditions exist:</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
| Summary Bit - If any bit from that register fails, it propagates to the previous register, bit 0.

Trace Numbers

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>...</th>
<th>561</th>
<th>575</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>1</td>
<td>15</td>
<td>29</td>
<td>43</td>
<td>57</td>
<td>71</td>
<td>85</td>
<td>99</td>
<td>...</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>2</td>
<td>16</td>
<td>30</td>
<td>44</td>
<td>58</td>
<td>72</td>
<td>86</td>
<td>100</td>
<td>...</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>3</td>
<td>17</td>
<td>31</td>
<td>45</td>
<td>59</td>
<td>73</td>
<td>87</td>
<td>101</td>
<td>...</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>4</td>
<td>18</td>
<td>32</td>
<td>46</td>
<td>60</td>
<td>74</td>
<td>88</td>
<td>102</td>
<td>...</td>
</tr>
<tr>
<td>Averaging on this trace is complete</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2844</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
To determine Register, Bit number, and Weight for trace numbers between 113 and 560 (not shown in the above table) use the following calculations.

The averaging status for trace numbers higher than 580 can NOT be tracked.

The following example calculates the Register, Bit number, and Bit Weight for trace # 400:

- To determine **Register** number, use \(((\text{Trace #} - 1) / 14) + 1\).
- To determine **Bit Number**, use the **remainder** +1 of the above calculation.
- \(((400-1)/14) + 1 = \text{Register# } r+1\text{Bit}
  
  - 399/14 = 28 r7
  - 28+1= Register 29
  - 7+1= Bit number 8
To determine **Bit Weight**: Use above table. For example: Bit 8 = 256

### STATus:OPERation:DEFine<keyword>
Summarizes conditions in the OPERation:Define:User<1|2|3> event registers.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:CONDition?</td>
<td>STAT:OPER:DEF:COND?</td>
</tr>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:OPER:DEF:ENAB &lt;bits&gt;</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:OPER:DEF?</td>
</tr>
<tr>
<td>:NTRansition &lt;bits&gt;</td>
<td>STAT:OPER:DEF:NTR &lt;bits&gt;</td>
</tr>
<tr>
<td>:PTRansition &lt;bits&gt;</td>
<td>STAT:OPER:DEF:PTR &lt;bits&gt;</td>
</tr>
</tbody>
</table>

#### Bit Weight

<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>USER1</td>
<td>any bit in the USER1 event register is set to 1</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>USER2</td>
<td>any bit in the USER2 event register is set to 1</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>USER3</td>
<td>any bit in the USER3 event register is set to 1</td>
</tr>
</tbody>
</table>

### STATus:OPERation:DEFine:USER<1|2|3><keyword>
Monitors conditions that you define and map in any of the three OPER:DEF:USER event registers.

<table>
<thead>
<tr>
<th>&lt;keyword&gt;</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>:ENABle &lt;bits&gt;</td>
<td>STAT:OPER:DEF:USER1:ENAB &lt;bits&gt;</td>
</tr>
<tr>
<td>[:EVENt]?</td>
<td>STAT:OPER:DEF:USER1?</td>
</tr>
<tr>
<td>:MAP &lt;bit&gt;,&lt;error&gt;</td>
<td>STAT:OPER:DEF:USER1:MAP &lt;bit&gt;,&lt;error&gt;</td>
</tr>
</tbody>
</table>

> *when error -113 occurs, bit 0 in USER1 will set to 1.*
Bit | Weight | Description                     | Bit is set to 1 when the following conditions exist:
--- | --- | --- | ---
0  | 1  | for user                        | user defined
1  | 2  | for user                        | user defined
2  | 4  | for user                        | user defined
3  | 8  | for user                        | user defined
4  | 16 | for user                        | user defined
5  | 32 | for user                        | user defined
6  | 64 | for user                        | user defined
7  | 128| for user                       | user defined
8  | 256| for user                        | user defined
9  | 512| for user                        | user defined
10 | 1024| for user                        | user defined
11 | 2048| for user                        | user defined
12 | 4096| for user                        | user defined
13 | 8192| for user                        | user defined
14 | 16384| for user                        | user defined

**STATus:OPERation:DEVice<keyword>**

Summarizes conditions in the OPERation:DEVice event registers.

*<keyword>*

- :CONDition?
  - Example: STAT:OPER:DEV:COND?
- :ENABle <bits>
  - Example: STAT:OPER:DEV:ENAB 16
- [:EVENt]?
  - Example: STAT:OPER:DEV?
- :NTRansition <bits>
  - Example: STAT:OPER:DEV:NTR 16
- :PTRansition <bits>
  - Example: STAT:OPER:DEV:PTR 0
<table>
<thead>
<tr>
<th>Bit</th>
<th>Weight</th>
<th>Description</th>
<th>Bit is set to 1 when the following conditions exist:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>Sweep Completed</td>
<td>When sweep is complete</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>64</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>256</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>512</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1024</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>2048</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>4096</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>8192</td>
<td>Unused</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>16384</td>
<td>Unused</td>
<td></td>
</tr>
</tbody>
</table>

Last modified:

- 13-Jan-2010  Changes to Limit and Average register
- 16-Jul-2009  Added note to Stat:Ques:Int:Meas register
- 16-Jul-2009  Fixed typo in STAT:QUES:INT register
- 9/19/06       MQ Modified for unlimited windows.
System Commands

Controls and queries settings that affect the PNA system.

<table>
<thead>
<tr>
<th>SYSTem:</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTive</td>
</tr>
<tr>
<td>CHANnel</td>
</tr>
<tr>
<td>MEASurement</td>
</tr>
<tr>
<td>CHANnels</td>
</tr>
<tr>
<td>CATalog?</td>
</tr>
<tr>
<td>COUPle</td>
</tr>
<tr>
<td>DELete</td>
</tr>
<tr>
<td>HOLD</td>
</tr>
<tr>
<td>RESume</td>
</tr>
<tr>
<td>COMMunicate</td>
</tr>
<tr>
<td>GPIB</td>
</tr>
<tr>
<td>PMETer</td>
</tr>
<tr>
<td>ADDRess</td>
</tr>
<tr>
<td>RDEVice</td>
</tr>
<tr>
<td>CLOSe</td>
</tr>
<tr>
<td>OPEN</td>
</tr>
<tr>
<td>READ?</td>
</tr>
<tr>
<td>RESet</td>
</tr>
<tr>
<td>WBINary</td>
</tr>
<tr>
<td>WBLock</td>
</tr>
<tr>
<td>WRITe</td>
</tr>
<tr>
<td>LAN:HOSTname</td>
</tr>
<tr>
<td>PSENsor</td>
</tr>
<tr>
<td>TCPip:CONTrol?</td>
</tr>
<tr>
<td>USB:PMETer:CAT?</td>
</tr>
<tr>
<td>CONFigure</td>
</tr>
<tr>
<td>REVision</td>
</tr>
<tr>
<td>CPU?</td>
</tr>
<tr>
<td>DSP?</td>
</tr>
<tr>
<td>DSPFpga?</td>
</tr>
<tr>
<td>CONFiguration:EDEVice</td>
</tr>
<tr>
<td>CORRection</td>
</tr>
<tr>
<td>INTerpolate:LINear</td>
</tr>
<tr>
<td>WIZard</td>
</tr>
<tr>
<td>ERRor?</td>
</tr>
<tr>
<td>COUNt?</td>
</tr>
<tr>
<td>REPort</td>
</tr>
<tr>
<td>SUNLeveled</td>
</tr>
<tr>
<td>FIFO</td>
</tr>
<tr>
<td>FPReset</td>
</tr>
<tr>
<td>MACRO:COPY</td>
</tr>
<tr>
<td>CHANnel</td>
</tr>
</tbody>
</table>

2849
Click on a blue keyword to view the command details.

See Also

- Learn about PNA Preferences
- Example Programs
- Synchronizing the PNA and Controller
- SCPI Command Tree

SYSTem:ACTive:CHANnel?
(Read-only) Returns the number of the active channel or an error message if there is no active channel. The active channel is the channel number that contains the active measurement.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:PRES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SYST:ACT:CHAN?</td>
</tr>
<tr>
<td></td>
<td>'Returns 1</td>
</tr>
</tbody>
</table>

**Return Type** Integer

**Default** Not Applicable

---

**SYSTem:ACTive:MEASurement?**

(Read-only) Returns the name of the active measurement or an error message if there is no active measurement. While looking at the PNA display, the active measurement is the trace that has an indented **Trace Status button** and a label in the upper-left corner of the display. Only displayed measurements can be active.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:PRES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SYST:ACT:MEAS?</td>
</tr>
<tr>
<td></td>
<td>'Returns &quot;CH1_S11_1&quot;</td>
</tr>
</tbody>
</table>

**Return Type** String

**Default** Not Applicable

---

**SYSTem:CHANnels:CATalog?**

(Read-only) Returns the channel numbers currently in use.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:CHAN:CAT?</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>system:channels:catalog?</td>
</tr>
<tr>
<td></td>
<td>'Returns:</td>
</tr>
<tr>
<td></td>
<td>&quot;1,2,3&quot;</td>
</tr>
</tbody>
</table>

**Return Type** String of comma-separated numbers

**Default** Not Applicable

---

**SYSTem:CHANnels:COUPle[:STATe] <bool>**
(Read-Write) Sets and reads the state of channel coupling. This causes the PNA to emulate Agilent 8720 channel coupling.

When set to ON, all existing S-parameter channels receive the stimulus settings of the active channel. Subsequent changes made to any coupled channel are changed on all coupled channels.

Channels with applications such as SMC, VMC, GCA, Noise, IMD are not affected.

Coupling is primarily aimed at stimulus settings (such as start, stop, points, power) but also applies to many trigger settings and to Cal Set pointers.

**Parameters**

<bool> ON (or 1) Channels are coupled

OFF (or 0) Channels are NOT coupled

**Examples**

SYST:CHAN:COUP 1

system:channels:couple:state OFF

**Query Syntax**

SYSTem:CHANnels:COUPle[:STATe]?

**Default**

OFF

**SYSTem:CHANnels:DELete <value>**

(Write-only) Deletes the specified channel.

**Parameters**

<value> Channel number to delete

**Examples**

SYST:CHAN:DEL 2

**Query Syntax**

Not Applicable

**Default**

Not Applicable

**SYSTem:CHANnels:HOLD**

(Write-only) Places all channels in hold mode. To place a single channel in hold mode, use SENS:SWE:MODE.

**Examples**

SYST:CHAN:HOLD

**Query Syntax**

Not Applicable

**Default**

Not Applicable
SYSTem:CHANnels:RESuMe

(Write-only) Resumes the trigger mode of all channels that was in effect before sending SYSTem:CHANnels:HOLD (must be sent before SYST:CHAN:RESuMe).

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:CHAN:RES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query Syntax</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

SYSTem:COMMunicate:GPIB:PMETer[:ADDRes] <num> Superseded

**Note:** This command is replaced with SYST:COMM:PSENsor

(Read-Write) Specifies the GPIB address of the power meter to be used in a source power calibration. When performing a source power cal, the PNA will search VISA interfaces that are configured in the Agilent IO Libraries on the PNA.

Use

**Parameters**

<num> GPIB address of the power meter. Choose any integer between 0 and 30.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:COMM:GPIB:PMET 13</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>system:communicate:gpib:pmeter:address 14</td>
</tr>
</tbody>
</table>

| Query Syntax | SYSTem:COMMunicate:GPIB:PMETer[:ADDRes]?
| Return Type  | Numeric |
| Default      | 13 |

SYSTem:COMMunicate:GPIB:RDEVice:CLOSE<e> <ID>

(Write only) Closes the remote GPIB session. This command should be sent when ending every successful OPEN session.

**Parameters**

<ID> Session identification number that was returned with the OPEN? command.

<table>
<thead>
<tr>
<th>Examples</th>
<th>See an example program</th>
</tr>
</thead>
</table>

| Query Syntax | Not Applicable |
| Default      | Not Applicable |
**SYSTem:COMMunicate:GPIB:RDEVice:OPEN**  <bus>, <addr>, <timeout>

(Read-Write) Initiates a GPIB pass-through session. First send this OPEN command, then send the OPEN query to read the session ID number. An existing GPIB pass-through session remains open after an instrument preset.

To learn more about GPIB pass-through capability, see the [example program](#).

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;bus&gt;</td>
<td>Bus ID number.</td>
</tr>
<tr>
<td></td>
<td>You can find the USB-GPIB adapter bus number by looking at the dialog that appears when the USB-GPIB device is connected. Error 1073 indicates the bus or address number is incorrect. Use 0 (zero) when connected using a GPIB cable to the PNA controller port.</td>
</tr>
<tr>
<td>&lt;addr&gt;</td>
<td>GPIB Address of the device to be controlled</td>
</tr>
<tr>
<td>&lt;timeout&gt;</td>
<td>The amount of time (in milliseconds) to wait for a response from the remote device after sending a command. A &quot;timeout&quot; error is displayed after this time has passed without a response.</td>
</tr>
</tbody>
</table>

**Examples**

See an example program

**Query Syntax**

SYSTem:COMMunicate:GPIB:RDEVice:OPEN?

Returns the session identification number that is used when communicating with this device.

**Return Type**

Numeric

**Default**

Not Applicable

---

**SYSTem:COMMunicate:GPIB:RDEVice:READ?**  <ID>

(Read-only) Returns data from the GPIB pass-through device.

**Parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;ID&gt;</td>
<td>Session identification number that was returned with the OPEN? command.</td>
</tr>
</tbody>
</table>

**Examples**

See an example program

**Return Type**

String

**Default**

Not Applicable
**SYSTem:COMMunicate:GPIB:RDEVice:RESet**

(Write-only) Performs the same function as SYST:COMM:GPIB:RDEV:CLOS except that ALL pass-through sessions are closed.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:COMM:GPIB:RDEV:RES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query Syntax</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Default</td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

**SYSTem:COMMunicate:GPIB:RDEVice:WBINary <ID>,<data>**

(Write-only) Sends data to a GPIB pass-through device. This command requires a header that specifies the size of the data to be written. The header (described below) is not passed along to the device.

Use this command if too many embedded quotes prevent you from using SYST:COMM:GPIB:RDEV:WRIT.

Use SYST:COMM:GPIB:RDEV:OPEN to open the pass through session.

**Parameters**

- **<ID>** Session identification number that was returned with the OPEN? command.
- **<data>** Data to be sent to the GPIB pass-through device. Use the following syntax:

  ```
  #<num_digits><byte_count><data_bytes><NL><END>
  ```

  - **<num_digits>** specifies how many digits are contained in **<byte_count>**
  - **<byte_count>** specifies how many data bytes will follow in **<data_bytes>**

**Examples**

| SYSTem:COMMunicate:GPIB:RDEVice:WBINary 101,#17ABC+XYZ<nl><end> |

- # - always sent before data.
- 1 - specifies that the byte count is one digit (7).
- 7 - specifies the number of data bytes that will follow, not counting <NL><END>.
- ABC+XYZ - Data block
- <nl><end> - always sent at the end of block data.

The following example sends a line feed at the end.
**SYSTem:COMMunicate:GPIB:RDEVice:WBIN** <ID>,<data>

*(Write-only)* Same as **SYSTem:COMM:GPIB:RDEV:WBIN** (above) but the header **IS** passed along to the device.

Use this command if too many embedded quotes prevent you from using **SYST:COMM:GPIB:RDEV:WRITe**.

**Parameters**

- **<ID>**  
  Session identification number that was returned with the **OPEN?** command.
- **<data>**  
  Data to be sent to the GPIB pass-through device. See previous command.

**Examples**  
See previous example.

**Query Syntax**  
Not Applicable

**Default**  
Not Applicable

---

**SYSTem:COMMunicate:GPIB:RDEVice:WRITe** <ID>,<string>

*(Write-only)* Sends ASCII string data to the GPIB pass-through device.

A line feed is NOT appended to the string data. To send a line feed, see the example in **SYST:COMM:GPIB:RDEV:WBIN**.

**Parameters**

- **<ID>**  
  Session identification number that was returned with the **OPEN?** command.
- **<string>**  
  Commands to be sent to the GPIB pass-through device.

**Examples**  
See an example program

**Query Syntax**  
Not Applicable

**Default**  
Not Applicable

---

**SYSTem:COMMunicate:LAN HOSTname?**
(Read-only) Returns the LAN hostname that is visible in the Help, About Network Analyzer dialog box. Learn more. This is the same information that is visible on the LXI compliance dialog.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>None</th>
</tr>
</thead>
<tbody>
<tr>
<td>Example</td>
<td>SYST:CONF:LAN:HOSTname?</td>
</tr>
<tr>
<td>Return Type</td>
<td>String</td>
</tr>
<tr>
<td>Default</td>
<td>Not applicable</td>
</tr>
</tbody>
</table>

**SYStem:COMMunicate:PSENsor <char>, <string>**

This command replaces SYST:COMM:GPIB:PMET:ADDR.

(Read-Write) Specifies the type and location of the power meter to be used in a source power calibration.

**Parameters**

- **<char>** Type of power meter/ sensor. Choose from:
  - **GPIB** GPIB power meter
  - **USB** USB power sensor or USB power sensor
  - **LAN** LAN enabled power meter

- **<string>** For **GPIB**, address of the power meter. Choose any integer between 0 and 30.
  For **USB**, the ID string of the power meter or power sensor. Use SYST:COMM:USB:PMET:CAT? to see a list of ID strings of connected power meters and sensors.
  For **LAN**, the hostname or IP address of the power meter.

**Examples**

- SYST:COMM:PSEN gpib, "14"
- syst:comm:psen lan, "mymeter.agilent.com"

**Query Syntax** SYSTem:COMMunicate:PSENo?  
**Return Type** Character / String  
**Default** GPIB
**SYSTem:COMMunicate:USB:PMETer:CATalog?**

*(Read-only)* Returns the ID string of power meters / sensors that are connected to the PNA USB. Use the list to select a power sensor for a **source power cal**.

These meter/sensor ID strings can NOT be used as the resource string for configuring a USB-based PMAR (**SYST:CONF:EDEV:IOConfig**).

<table>
<thead>
<tr>
<th>Parameters</th>
<th></th>
</tr>
</thead>
</table>
| **Examples** | **SYST:COMM:USB:PMET:CAT?**  
**system:communicate:usb:pmeter:catalog?** |

**Return Type** Comma-delimited strings. Two power sensor strings are separate by a semicolon.

**Default** Not applicable

**SYSTem:COMMunicate:TCPip:CONTrol?**

*(Read-only)* Queries the TCP/IP port number to use for opening a TCP/IP socket control connection to the PNA. The control connection is used for two purposes:

1. To perform a Device Clear operation on the PNA
2. To detect when a Service Request (SRQ) event occurs on the PNA.

The port number can range from 5000 to 5099. The PNA will skip over 5025 as it is being used for the primary socket connection.

To detect an SRQ, your program sends the appropriate commands via the regular socket connection to set up for a SRQ event to occur – the same sequence of commands as if you were sending them via GPIB. You write your program so that while your program is doing SCPI transactions on the standard socket connection, a second thread of execution in your program detects the SRQ on the control connection and responds to the event. When the SRQ event occurs, the PNA sends a “SRQ +xxx/n” message on the control connection (where ‘/n’ is linefeed character, ASCII value 10 decimal). The xxx value in the “SRQ +xxx/n” string is the IEEE 488.2 status byte at the time the SRQ was generated. So ‘listening’ for that on the control connection is how your program detects the event. If for your socket communication you’re using a software API that provides for asynchronous communication via a callback mechanism (for example, if you’re using Microsoft’s ‘winsock’ API, or their .NET ‘Socket’ class as in the example program below), in that case your ‘listener’ execution thread is created implicitly for you so your program doesn’t have to create one explicitly.

**Note:** If this SCPI query is sent to the PNA via a SCPI parser other than a TCP/IP socket connection (for example, if sent via GPIB), the query is not applicable in that case and will return value of 0.
**Parameters**  None

**Example**  See example program

**Return Type**  Integer

**Default**  Not applicable

---

**SYSTem:CONFigure <model>,<address>**

(Write-only) Restarts as an "N-port" PNA using the specified multiport test set.

Learn more about PNA Multiport capability.

See other commands to configure multiport test sets.

**Parameters**

- **<model>**  String - Model of the test set with which to restart.
  
  Use "Native" to restart without a test set.

  To see a list of supported test sets, use **SENS:MULT:CAT?**

- **<address>**  Numeric - GPIB Address of the test set. Ignored when model = "Native".

**Examples**

```
SYST:CONF "NATIVE",0  
system:configure "N44xx",18
```

**Query Syntax**  Not Applicable

**Default**  Not Applicable

---

**SYSTem:CONFigure:REVision:CPU?**
(Read-only) Returns a number that corresponds to the PNA CPU speed that is visible in the Help, About Network Analyzer dialog box. Learn more.

Use the following table to learn the clock speed using the returned value.

<table>
<thead>
<tr>
<th>Reported CPU version</th>
<th>Clock speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>266 MHz</td>
</tr>
<tr>
<td>2.0</td>
<td>500 MHz</td>
</tr>
<tr>
<td>3.0</td>
<td>1100 MHz</td>
</tr>
<tr>
<td>4.0</td>
<td>1600 MHz</td>
</tr>
<tr>
<td>5.0</td>
<td>2000 MHz</td>
</tr>
</tbody>
</table>

### Parameters
- None

### Example
```
SYST:CONF:REV:CPU?
```

### Return Type
String

### Default
Not applicable

**SYSTem:CONFigure:REVision:DSP?**

(Read-only) Returns the DSP Revision number that is visible in the Help, About Network Analyzer dialog box. Learn more.

### Parameters
- None

### Example
```
SYST:CONF:REV:DSP?
```

### Return Type
String

### Default
Not applicable

**SYSTem:CONFigure:REVision:DSPFpga?**
(Read-only) Returns the DSP FPGA Revision number that is visible in the Help, About Network Analyzer dialog box. Learn more.

**Parameters**
None

**Example**
SYST:CONF:REV:DSPF?

**Return Type**
String

**Default**
Not applicable

**SYSTem:CORRection:WIZard[:IMMediate] <char>**

(Write-only) Launches either the Calibration Wizard or the Version 2 Calibration Kit File Manager dialog box.

Remote operation returns immediately after the dialog is launched. This is done to avoid timeout issues with I/O protocols such as VISA. Although it is possible to send commands to the PNA while the dialog is open, this is not encouraged. Application programs should wait until the dialog is closed before resuming remote operations.

**Parameters**

<char> Choose from:

- **MAIN** - Launches the Calibration Wizard which matches the current channel, such as standard S-params, NoiseFigure, GCA, and so forth.
- **CKIT** - Launches the Version 2 Calibration Kit File Manager dialog box.

Both display on the PNA screen.

**Examples**

SYST:CORR:WIZ MAIN
system:correction:wizard:immediate ckit

**Query Syntax**
Not Applicable

**Default**
Not Applicable

**SYStem:ERRor?**
(Read-only) Returns the next error in the error queue. Each time the analyzer detects an error, it places a message in the error queue. When the SYSTEM:ERROR? query is sent, one message is moved from the error queue to the output queue so it can be read by the controller. Error messages are delivered to the output queue in the order they were received. The error queue is cleared when any of the following conditions occur:

- When the analyzer is switched ON.
- When the *CLS command is sent to the analyzer.
- When all of the errors are read.

If the error queue overflows, the last error is replaced with a "Queue Overflow" error. The oldest errors remain in the queue and the most recent error is discarded.

See list of all SCPI Errors.

**Examples**

<table>
<thead>
<tr>
<th>Example</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYST:ERR?</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>system:error?</td>
<td></td>
</tr>
</tbody>
</table>

**SYSTem:ERRor:COUNt?**

(Read-only) Returns the number of errors in the error queue. Use SYST:ERR? to read an error.

See list of all SCPI Errors.

**Examples**

<table>
<thead>
<tr>
<th>Example</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYST:ERR:COUN?</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>system:error:count?</td>
<td></td>
</tr>
</tbody>
</table>

**SYSTem:ERRor:REPort:SUNLeveled <bool>**
(Read-Write) Specifies whether or not to report Source Unleveled errors to the SCPI system error buffer.

This setting will revert to the default (OFF) setting on Instrument Preset.

**Parameters**

- ON (or 1) Report Source Unleveled Errors. Read errors from the system error buffer using `SYST:ERR?`
- OFF (or 0) Do NOT report Source Unleveled Errors.

**Examples**

```
SYST:ERR:REP:SUNL 1
system:error:report:sunleveled ON
```

**Query Syntax**

```
SYSTem:ERRor:REPort:UNLeveled?
```

**Default**

OFF

---

**SYSTem:FPReset**

(Write-only) Performs a standard Preset, then deletes the default trace, measurement, and window. The PNA screen becomes blank.

**Examples**

```
SYST:FPR
system:fpreset
```

**Default**

Not applicable

---

**SYSTem:MACRo:COPY:CHANnel<cnum>[:TO] <num>**

(Write-only) Copies ALL settings from <cnum> channel to <num> channel. Learn more about copy channels.

Use `SENS:PATH:CONF:COPY` to copy ONLY mechanical switch and attenuator settings.

**Parameters**

- <cnum> Channel number to copy settings from. If unspecified, value is set to 1.
- <num> Channel number to copy settings to.

**Examples**

```
SYST:MACR:COPY:CHAN1 2
system:macro:copy:channel2:to 3
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable
SYSTem:MACRo:COPY:CHANnel<fromChan>:SOURce <fromPort>,<toChan>,<toPort>

(Write-only) Copies and applies an existing Source Power Calibration to another channel. Learn more about source power calibration.

Parameters

<fromChan> Channel number of the existing source power correction.
<fromPort> Port number of the existing source power correction.
<toChan> Channel number to which the source power correction will be copied.
<toPort> Port number to which the source power correction will be applied.

Examples

```
SYST:MACR:COPY:CHAN1:SOUR 1,2,1
```

```
system:macro:copy:channel2:source 2,1,2
```

Query Syntax Not Applicable

Default Not Applicable

SYSTem:MCLass:CATalog?

(Read-only) Returns measurement classes available on the PNA. Learn more about Measurement Classes.

Parameters None

Examples

```
SYST:MClass:CAT?
```

Return Type String of comma-separated measurement class names. See the complete list of measurement class names.

Default Not Applicable

SYSTem:MEASurement:CATalog? [chan]
(Read-only) Returns ALL measurement numbers, or measurement numbers from a specified channel.

**Parameters**

[chann] Optional. Channel number to catalog. If not specified, all measurement numbers are returned.

**Examples**

'Returns all measurement names

SYST:MEAS:CAT?

'Returns the measurement names on channel 2

system:measurement:catalog? 2

**Return Type**

String of comma-separated numbers

For example: "1,2"

**Default** Not Applicable

---

**SYSTem:MEASurement<n>:NAME?**

(Read-only) Returns the name of the specified measurement.

**Parameters**

<n> Measurement number for which to return the measurement name. If unspecified, value is set to 1.

**Examples**

'Returns the name of measurement 2

SYST:MEAS2:NAME?

**Return Type** String

**Default** Not Applicable

---

**SYSTem:MEASurement<n>:TRACe?**
(Read-only) Returns the trace number of the specified measurement number. Trace numbers restart for each window while measurement numbers are always unique.

Parameters

<n> Measurement number for which to return the trace number. If unspecified, value is set to 1.

Examples

'Seconds the trace number of measurement 1
SYST:MEAS1:TRAC?

Return Type Numeric
Default Not Applicable

SYSTem:MEASurement<n>:WINDow?

(Read-only) Returns the window number for the specified measurement number.

Parameters

<n> Measurement number for which to return the window number. If unspecified, value is set to 1.

Examples

'Seconds the window number of measurement 2
SYST:MEAS2:WIND?

Return Type Numeric
Default Not Applicable

SYSTem:POWer<pnum>:LIMit <value>

(Read-Write) Sets and returns the power limit for the specified port. Learn more about Power Limit.

Parameters

<pnum> Port number. Choose any PNA port.
<value> Power limit in dBm

Examples

SYST:POW1:LIM 5
system:power2:limit 0

Query Syntax SYSTem:POWer<pnum>:LIMit?

Return Type Numeric
Default 100 dBm
**SYSTem:POWER:LIMit:LOCK <bool>**

(Read-Write) Enables or disables the ability to change the power limit values through the user interface. Learn more about Power Limit.

**Parameters**

- **<bool>** Power limit lock. Choose from:
  - **ON or 1** - Enables the ability to change the power limit values from the user interface.
  - **OFF or 0** - Disables the ability to change the power limit values from the user interface.

**Examples**

```
SYST:POW:LIM:LOCK 1
system:power:limit:lock OFF
```

**Query Syntax**

SYSTem:POWer:LIMit:LOCK?

**Return Type**

Boolean

**Default**

OFF

---

**SYSTem:POWER<pnum>:LIMit:STATe <bool>**

(Read-Write) Enables or disables the power limit for the specified port. Learn more about Power Limit.

**Parameters**

- **<pnum>** Port number. Choose any PNA port.
- **<value>** Power limit state. Choose from:
  - **ON or 1** Enables the power limit for the port<pnum>
  - **OFF or 0** Disables the power limit for the port<pnum>

**Examples**

```
SYST:POW1:LIM:STAT ON
system:power2:limit:state 0
```

**Query Syntax**

SYSTem:POWer<pnum>:LIMit:STATe?

**Return Type**

Boolean

**Default**

OFF
**SYSTem:PRESet**

(Write-only) Deletes all traces, measurements, and windows. In addition, resets the analyzer to factory defined default settings and creates a S11 measurement named "CH1_S11_1". For a list of default settings, see Preset.

Regardless of the state of the User Preset Enable checkbox, the SYST:PRESet command will always preset the PNA to the factory preset settings, and SYST:UPReset will always perform a User Preset.

If the PNA display is disabled with DISP:ENAB OFF then SYST:PRES will NOT enable the display.

This command performs the same function as *RST with one exception: Syst:Preset does NOT reset Calc:FORMAT to ASCII as does *RST.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:PRES system:reset</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default</td>
<td>Not applicable</td>
</tr>
</tbody>
</table>

**SYSTem:SECurity[:LEVel] <char>**

(Read-Write) Sets and returns the display of frequency information on the PNA screen and printouts.

Learn more about security level.

**Parameters**

<char> Choose from:

- **NONE** - ALL frequency information is displayed.
- **LOW** - NO frequency information is displayed. Frequency information can be redisplayed using the Security Setting dialog box or this command.
- **HIGH** - LOW setting plus GPIB console is disabled. Frequency information can be redisplayed ONLY by performing a Preset, recalling an instrument state with None or Low security settings, or using this command.
- **EXTRa** - HIGH setting plus:
  - ASCII data saving is disabled. Same method to redisplay frequency information as HIGH setting.
  - Mixer setup files (*.mxr) can NOT be saved.

<table>
<thead>
<tr>
<th>Examples</th>
<th>SYST:SEC LOW system:security:level high</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query Syntax</td>
<td>SYSTem:SECurity[:LEVel]?</td>
</tr>
</tbody>
</table>
**Return Type**  Character

**Default**  None

---

**SYSTem:SET <block>**

(Read-Write) Sends a definite-length binary block Instrument state and sets the PNA with those settings. This command does the same as saving a *.sta file to the PNA (MMEM:STOR STATE) and then MMEM:TRAN to transfer the file to the computer.

**Parameters**

- `<block>`  The Instrument state file as definite-length arbitrary binary block.

**Examples**  SYST:SET <block>

**Query Syntax**  SYSTem:SET?  (This saves the instrument state file to the remote computer.)

**Return Type**  Definite-length arbitrary binary block.

**Default**  Not Applicable

---

**SYSTem:SHORTcut<n>:ARGuments<string>**

(Read-Write) Reads and writes the arguments for the specified macro. On the Edit Macro Dialog, this is called the "Macro run string parameters".

**Parameters**

- `<n>`  Numeric. Number of the macro that is stored in the PNA.

To find the number of a macro, either open the Macro Setup dialog and count the line number of the desired macro, or query the titles of all 12 macros for the desired macro title.

- `<string>`  Arguments for the specified macro.

**Examples**  SYST:SHORT1:ARG


**Query Syntax**  SYSTem:SHORTcut<n>:ARGuments?

**Default**  Not Applicable

---

**SYSTem:SHORTcut<n>:DELeTe**
(Write-only) Removes the specified macro from the list of macros in the PNA. Does not delete the macro executable file.

**Parameters**

<n> Numeric. Number of the macro that is stored in the PNA.

To find the number of a macro, either open the Macro Setup dialog and count the line number of the desired macro, or query the titles of all 12 macros for the desired macro title.

**Examples**

SYST:SHOR1:DEL

**Query Syntax** Not Applicable

**Default** Not Applicable

**SYSTem:SHORtcut<n>:EXECute**

(Write-only) Executes (runs) the specified Macro (shortcut) that is stored in the PNA.

**Parameters**

<n> Numeric. Number of the macro that is stored in the PNA.

To find the number of a macro, either open the Macro Setup dialog and count the line number of the desired macro, or query the titles of all 12 macros for the desired macro title.

**Examples**

SYST:SHOR1:EXEC

**Query Syntax** Not Applicable

**Default** Not Applicable

**SYSTem:SHORtcut<n>:PATH <string>**
(Read-Write) Defines a Macro (shortcut) by linking a path and file name to the Macro number. To be executed, the executable file must be put in the PNA at the location indicated by this command.

**Parameters**

- <n> Numeric. Number of the macro to be stored in the analyzer. If the index number already exists, the existing macro is replaced with the new macro.
- <string> Full path, file name, and extension, of the existing macro "executable" file.

To find the number of a macro, either open the Macro Setup dialog and count the line number of the desired macro, or query the titles of all 12 macros for the desired macro title.

**Examples**

```
SYST:SHOR1:PATH "C:/Program Files/Agilent/Network Analyzer/Documents/unguideMultiple.vbs"
```

**Query Syntax**

```
SYSTem:SHORtcut<n>:PATH?
```

**Default** Not Applicable

---

**SYSTem:SHORtcut<n>:TITLe<string>**

(Read-Write) Reads and writes the name of the specified macro.

**Parameters**

- <n> Numeric. Number of the macro that is stored in the PNA.

To find the number of a macro, either open the Macro Setup dialog and count the line number of the desired macro, or query the titles of all 12 macros for the desired macro title.

- <string> The name to be assigned to the macro.

**Examples**

```
SYST:SHOR1:TITL "Guided 4-Port Cal"
```

**Query Syntax**

```
SYSTem:SHORtcut<n>:TITLe?
```

**Default** Not Applicable

---

**SYSTem:TOUChscreen[:STATe] <bool>**
(Read-Write) Enables and disables the PNA-X touchscreen.

This setting remains until changed again from the front-panel or remotely, or until the hard drive is changed or reformatted.

**Parameters**

<bool> Choose from:

- **ON (1)** Enables the touchscreen.
- **OFF (0)** Disables the touchscreen.

**Examples**

```
SYST:TOUC 1
system:touchscreen:state OFF
```

**Query Syntax**

SYSTem:TOUCHscreen[:STATe]?

**Return Type**

Boolean

**Default**

ON when shipped from factory.

**SYSTem:UPReset**

(Write-only) Performs a User Preset. There must be an active User Preset state file (see **Load** and **Save**) or an error will be returned. Learn more about User Preset.

Regardless of the state of the User Preset Enable checkbox, the SYST:PRESet command will always preset the PNA to the factory preset settings, and **SYST:UPReset** will always perform a User Preset.

**Examples**

```
SYST:UPReset
system:upreset
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

**SYSTem:UPReset:FPANel[:STATe] <bool>**
(Read-Write) 'Checks' and 'clears' the enable box on the User Preset dialog box. This only affects subsequent Presets from the front panel user interface.

Regardless of the state of the User Preset Enable checkbox, the `SYST:PRESet` command will always preset the PNA to the factory preset settings, and `SYST:UPReset` will always perform a User Preset.

**Parameters**

<bool> Front Panel User Preset State. Choose from:

0 User Preset OFF

1 User Preset ON

**Examples**

```
SYST:UPR:FPAN 1
system:upreset:fpanel:state 0
```

**Query Syntax**

`SYSTem:UPReSet:FPANel[:STATe]`?

**Return Type**

Boolean

**Default**

0

---

**SYSTem:UPReSet:LOAD[:FILE] <file>**

(Write-only) Loads an existing instrument state file (.sta or .cst) to be used for User Preset. Subsequent execution of `SYSTem:UPReSet` will cause the PNA to assume this instrument state.

Regardless of the state of the User Preset Enable checkbox, the `SYST:PRESet` command will always preset the PNA to the factory preset settings, and `SYST:UPReSet` will always perform a User Preset.

**Learn more about User Preset.**

**Parameters**

<file> String - Name of the file to be loaded. The default folder "C:/Program Files/Agilent/Network Analyzer/Documents" is used if unspecified. Change the default folder name using `MMEMory:CDIRectory`.

**Examples**

```
SYST:UPR:LOAD '1MHzto20GHzUserPreset.cst'
```

system:upreset:load:file 'C:/Documents and Settings/Administrator/My Documents/NewUserPreset.cst'

**Query Syntax**

Not Applicable

**Default**

Not Applicable
**SYSTem:UPReset:SAVE[:STATe]**

*(Write-only)* Saves the current instrument settings as UserPreset.sta. Subsequent execution of `SYSTem:UPReset` will cause the PNA to assume this instrument state.

Regardless of the state of the User Preset Enable checkbox, the `SYST:PRESet` command will always preset the PNA to the factory preset settings, and `SYST:UPReset` will always perform a User Preset. Learn more about User Preset.

**Examples**

- `SYST:UPR:SAVE`
- `system:upreset:save:state`

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SYSTem:WINDows:CATalog?**

*(Read-only)* Returns the numbers of window that are currently being used.

**Examples**

- `SYST:WIND:CAT?`
- `system:windows:catalog?`

**Return Type** String of comma-separated numbers.

For example: "1,2"

**Default** Not Applicable

---

**Last modified:**

- **11-Jan-2011**  Minor edit
- **4-Nov-2010**  Security for external sources (9.33)
- **14-Oct-2010**  Added note to PMET:CAT?
- **16-Sep-2010**  Added channel delete (A.09.30)
- **9-Apr-2010**  Added Preset note to 'unleveled' command
- **30-Nov-2009**  Added Help About read commands(9.1)
- **30-Jul-2009**  Added syst:conf:edev and RTOF (9.0)
- **24-Feb-2009**  Added Chan:Coup; Replaced True/False
- **4-Nov-2008**  Added FIFO (8.33)
- **30-Oct-2008**  Added several meas/trace/window query commands (8.33.x)
- **29-Sep-2008**  Removed rev from Psensor example
- **17-Sep-2008**  Added Syst:Pres vs *RST note
<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>28-Aug-2008</td>
<td>Updated Launch Cal Wiz command</td>
</tr>
<tr>
<td>11-Feb-2008</td>
<td>Added Noise switch preference (8.2)</td>
</tr>
<tr>
<td>5-Feb-2007</td>
<td>Added Extra security and USB power meter commands</td>
</tr>
<tr>
<td>23-Feb-2007</td>
<td>Added touchscreen command</td>
</tr>
<tr>
<td>15-Nov-2006</td>
<td>Added Unleveled Error reporting</td>
</tr>
<tr>
<td>31-Oct-2006</td>
<td>Added PSRTrace command</td>
</tr>
</tbody>
</table>
External Device Commands

Configures and makes settings for an external device.

**SYST:CONF:EDEVice:**
- ADD
- CAT?
- DRIver
- DTYPe
- IOConfig
- IOENable
- REMove
- STATe
- TOUT

<table>
<thead>
<tr>
<th>PMAR</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLIMit</td>
</tr>
<tr>
<td>FMAXimum</td>
</tr>
<tr>
<td>FMINimum</td>
</tr>
<tr>
<td>READING</td>
</tr>
<tr>
<td>COUNT</td>
</tr>
<tr>
<td>NTOLerance</td>
</tr>
<tr>
<td>SENSor</td>
</tr>
<tr>
<td>TABLe</td>
</tr>
<tr>
<td>CFAC:</td>
</tr>
<tr>
<td>DATA</td>
</tr>
<tr>
<td>FREQuency</td>
</tr>
<tr>
<td>LOSS</td>
</tr>
<tr>
<td>DATA</td>
</tr>
<tr>
<td>FREQuency</td>
</tr>
<tr>
<td>STATe</td>
</tr>
<tr>
<td>RFACtor</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SOURce</th>
</tr>
</thead>
<tbody>
<tr>
<td>DPP</td>
</tr>
<tr>
<td>TMODE</td>
</tr>
<tr>
<td>TPORT</td>
</tr>
</tbody>
</table>

Click on a blue keyword to view the command details.

See Also
Learn about: Configure an External Source
Learn about: Configure a PMAR Device
Example: Configure an External Source
Example: Configure a PMAR Device

**SYST:PREF:ITEM:EDEV:DPOL** - Determines whether External Devices remain activated or are de-activated when the PNA is Preset or when a Instrument State is recalled.

Synchronizing the PNA and Controller

SCPI Command Tree

---

**SYSTem:CONFigure:EDEVice:ADD <name>**

*(Write-only)* Adds an external device to the list of configured devices. This is the same as pressing **New** on the **Select an External Device** dialog.

Upon creation, all settings on the new device are set to the defaults. The device is not active until set using **SYST:CONF:EDEV:STAT**

**Parameters**

- `<name>` String - Model and type of the external device.

To see a list of configured external devices, use **SYST:CONF:EDEV:CAT?**

**Examples**

```
SYST:CONF:EDEV:ADD "myDevice"
```

```
system:configure:edev:add "myDevice"
```

**Query Syntax** Not Applicable

**Default** Not Applicable

---

**SYSTem:CONFigure:EDEVice:CAT?**
(Read-only) Returns a list of names of all configured devices. These are devices that appear in the External Devices dialog.

Use `SENS:FOM:CAT?` to report all active devices.
Use `Source:CAT?` to report all active sources.

**Parameters**

None

**Example**

```
SYST:CONF:EDEV:CAT?
```
```
system:configure:edevice:cat
```

**Return Type**

String of comma-separated devices. "name,name"

**Default**

Not applicable

---

**SYSTem:CONFigure:EDEVice:DRIVer <name>,<value>**

(Read-Write) Sets and returns the external device driver (model).

**Parameters**

- `<name>` String - Name of the device.
- `<value>` String - External device driver (model). Choose from the following:
  - `AGPM` for all power meters.
  - See a list of supported external source drivers.

**Examples**

```
SYST:CONF:EDEV:DRIV "myDevice","AGPM"
```
```
system:configure:edevice:driver "myDevice","AGESG"
```

**Query Syntax**

```
SYSTem:CONFigure:EDEVice:DRIVer? <name>
```

**Return Type**

String

**Default**

"AGGeneric"

---

**SYSTem:CONFigure:EDEVice:DTYPE <name>,<type>**
**SYSTem:CONFigure:EDEVice:DTYPe <name>,<type>**

*(Read-Write)* Sets and returns the Device Type for the external device.

**Parameters**

- `<name>`: String - Name of the device to modify.
- `<type>`: String - Device type - not case sensitive. Choose from:
  - "Source" - external source
  - "Power Meter" - power meter

**Examples**

- `SYST:CONF:EDEV:DTYP "myDevice","Power Meter"

**Query Syntax**

- `SYSTem:CONFigure:EDEVice:DTYPe? <name>`

**Return Type**

- String

**Default**

- None

---

**SYSTem:CONFigure:EDEVice:IOConfig <name>,<value>**

*(Read-Write)* Sets and return the configuration path for the specified external device.

**Parameters**

- `<name>`: String - Name of the device.
- `<value>`: String - Configuration path. Any valid VISA resource shown in the IO Configuration field of the External Devices dialog, enclosed in quotes.
  
  Do NOT use the ID string of a PMAR USB power sensor as the resource string. The ID string is returned by `SYST:COMM:USB:PMET:CAT`?

**Examples**

- `SYST:CONF:EDEV:IOC "myDevice","GPIB0::13::INSTR"

**Query Syntax**

- `SYSTem:CONFigure:EDEVice:IOConfig? <name>`

**Return Type**

- String

**Default**

- " " Empty String

---

**SYSTem:CONFigure:EDEVice:IOENable <name>,<value>**
Enable or disable communication with an external device. When attempting to enable a device, the PNA-X will verify communication with the device. An error will result if communication cannot be verified. Generally, an external device should only be enabled after all other configuration settings have been made. Learn more.

**Parameters**

- `<name>` String - Name of the device.
- `<value>` Boolean - Choose from:
  - OFF or 0 - Device communication enabled
  - ON or 1 - Device communication disabled

**Examples**

```bash
SYST:CONF:EDEV:IOEN "myDevice", ON
system:configure:edevice:ioenable "myDevice", 0
```

**Query Syntax**

SYSTem:CONFigure:EDEVice:IOENable? <name>

**Return Type**

Boolean

**Default**

ON

---

**SYSTem:CONFigure:EDEVice:REMove <name>**

(Write-only) Removes the specified device from the list of configured devices. If the device is a Source and both Active and I/O Enabled is checked (ON), then the RF power state is set to OFF. Learn more.

**Parameters**


**Examples**

```bash
SYST:CONF:EDEV:REM "myDevice"
system:configure:edevice:remove "myDevice"
```

**Query Syntax**

Not Applicable

**Default**

Not Applicable

---

**SYSTem:CONFigure:EDEVice:STATe <name>,<value>**


(Read-Write) Set and return the state of activation of the device. Learn more.
Send this command AFTER sending other External Device settings (especially SYST:CONF:EDEV:DTYP) to avoid communicating with the device before it has been fully configured.
See Also: SYST:PREF:ITEM:EDEV:DPOL - Determines whether External Devices remain activated or are de-activated when the PNA is Preset or when a Instrument State is recalled.

Parameters

- `<name>`: String - Name of the device.
- `<value>`: Boolean - Choose from:
  - OFF or 0 - Device is NOT activated
  - ON or 1 - Device is activated.

Examples

SYST:CONF:EDEV:STAT "myDevice", ON
system:configure:edevice:state "myDevice", 0

Query Syntax

SYSTem:CONFigure:EDEVice:STATe? <name>

Return Type

Boolean

Default

OFF - When configured using the front panel user interface, the device is ON (activated) by default.

SYSTem:CONFigure:EDEVice:TOUT <name>,<value>

(Read-Write) Set and return the time out value for the specified external device. This is the time allowed for communication with the device before an error is generated.

Parameters

- `<name>`: String - Name of the device.
- `<value>`: Time out value in seconds.

Examples

SYST:CONF:EDEV:TOUT "myDevice", 2
system:configure:edevice:tout "myDevice", 5

Query Syntax

SYSTem:CONFigure:EDEVice:TOUT? <name>

Return Type

Numeric
**SYSTem:CONFigure:EDEVice:SOURce:DPP <name>,<value>**

*(Read-Write)* Sets and returns the amount of time the PNA should wait after for an external source to settle before making a measurement at each data point. This setting applies to all channels that use this external source.

**Parameters**

- `<name>`  String - Name of the device.
- `<value>` Dwell time in seconds.

**Examples**

```
sYST:CONF:EDEV:SOUR:DPP "myDevice",2
system:configure:edevice:source:dpp "myDevice",.1
```

**Query Syntax**

SYSTem:CONFigure:EDEVice:SOURce:DPP? <name>

**Return Type** Numeric

**Default** 3.114 e-3

---

**SYSTem:CONFigure:EDEVice:SOURce:TMODe <name>,<value>**

*(Read-Write)* Sets and returns the trigger mode for an external source. Learn more.

**Parameters**

- `<name>`  String - Name of the device.
- `<value>` Trigger Mode. Choose from:
  - CW - Software CW mode
  - HW - Hardware list mode

**Examples**

```
system:configure:edevice:source:tmode HW,"myDevice"
```

**Query Syntax**

SYSTem:CONFigure:EDEVice:SOURce:TMODe? "myDevice"

**Return Type** Character
**SYSTem:CONFigure:EDEVice:SOURce:TPORt <name>,<value>**

(Read-Write) Sets and returns the PNA port through which an external source is to be triggered.

**Parameters**

- **<name>** String - Name of the device.
- **<value>** Trigger Port. Choose from:
  - For PNA 'C' models, choose `bnc1`
  - For PNA-X models, choose from `aux1` or `aux2`

**Examples**

```
SYST:CONF:EDEV:SOUR:TPOR "myDevice",bnc1
system:configure:edevice:source:tmode "myDevice",aux1
```

**Query Syntax**

```
SYSTem:CONFigure:EDEVice:SOURce:TMODe? <name>
```

**Return Type** Character

**Default** For PNA 'C' models - `bnc1`
For PNA-X models - `aux1`

---

**SYSTem:CONFigure:EDEVice:PMAR:FLIMit <name>,<value>**

(Read-Write) Enable or disable the power meter min and max frequencies.

**Parameters**

- **<name>** String - Name of the power meter.
- **<value>** Boolean. State of min and max frequency. Choose from:
  - `OFF` or `0` - Min and max frequencies disabled.
  - `ON` or `1` - Min and max frequencies enabled.

**Examples**

```
SYST:CONF:EDEV:PMAR:FLIM "myDevice", 0
system:configure:edevice:pmar:flimit "myDevice", ON
```

See example program
SYSTem:CONFigure:EDEVice:PMAR:FMAXimum <name>,<value>

(Read-Write) Set and return the maximum frequency of the power meter.

Parameters

- `<name>` String - Name of the power meter.
- `<value>` Numeric - Max frequency in Hz.

Examples

```
SYST:CONF:EDEV:PMAR:FMAX "myDevice", 1e10
system:configure:edev:pmar:fmaximum "myDevice", 3e9
```

See example program

Query Syntax

SYSTem:CONFigure:EDEVice:PMAR:FMINimum? <name>
(Read-Write) Set and return the minimum frequency of the power meter.

**Parameters**

- `<name>` String - Name of the power meter.
- `<value>` Numeric - Min frequency in Hz.

**Examples**

```
SYST:CONF:EDEV:PMAR:FMIN "myDevice", 1e10
system:configure:edevice:pmar:fminimum "myDevice", 3e9
```

See example program

**Query Syntax**

```
SYSTem:CONFigure:EDEVice:PMAR:FMAXimum? <name>
```

**Return Type**

Numeric

**Default** Not Applicable

---

**SYSTem:CONFigure:EDEVice:PMAR:READing:COUNt <name>,<value>**

(Read-Write) This command, along with SYST:CONF:EDEV:PMAR:READ:NTOL, allows for settling of the power sensor READINGS.

Set and return the maximum number of power readings that are taken at each stimulus point to allow for measurement settling. Each reading is averaged with the previous readings at that stimulus point.

When this average meets the Average:NTOLerance value or this number of readings has been made, the average is returned as the valid reading.

**Parameters**

- `<name>` String - Name of the power meter.
- `<value>` Number of readings. Choose a value between 1 and 25

**Examples**

```
```

See example program

**Query Syntax**

```
SYSTem:CONFigure:EDEVice:PMAR:READing:COUNt? <name>
```

**Return Type**

Numeric
SYSTem:CONFigure:EDEVice:PMAR:READing:NTOLerance <name>,<value>

(Read-Write) This command, along with SYST:CONF:EDEV:PMAR:READ:COUN, allows for settling of the power sensor READINGS.
Each power reading is averaged with the previous readings at each stimulus point. When the average meets this nominal tolerance value, or the max number of readings has been made, the average is returned as the valid reading.

Parameters

<name>  String - Name of the power meter.
<value>  Power measurement settling tolerance value in dB. Choose any number between 0 and 5.

Examples
system:configure:edevice:pmar:reading:ntolerance "myDevice", .01
See example program

SYSTem:CONFigure:EDEVice:PMAR:SENSor <name>,<value>

Query Syntax SYSTem:CONFigure:EDEVice:PMAR:READing:NTOLerance? <name>
Return Type Numeric

Default .05
(Read-Write) Sets and returns the power sensor channel (1 or 2) to be used. This performs the same function as the Use this sensor only checkbox.

**Parameters**

- **<name>** String - Name of the power meter.
- **<value>** Power Meter channel.
  - 1 - Channel A
  - 2 - Channel B

**Examples**

```markdown
SYST:CONF:EDEV:PMAR:SENS "myDevice",2
system:configure:edevice:pmar:sensor "myDevice",1
```

See example program

**Query Syntax**

```markdown
SYSTem:CONFigure:EDEVice:PMAR:TABLe:CFAC:DATA <name>,<value>[,.value]
```

**Return Type** Numeric

**Default** 1

---

**SYSTem:CONFigure:EDEVice:PMAR:TABLE:CFAC:DATA <name>,<value>[,.value]**

(Read-Write) Sets and returns the cal factor data for the power sensor.

**Parameters**

- **<name>** String - Name of the power meter.
- **<value>[,.value]** Cal factor data in percent. For each frequency used with SYST:CONF:EDEV:PMAR:TABLE:CFAC:FREQ, enter a cal factor number between 1 and 100.

**Examples**

```markdown
system:configure:edevice:pmar:table:cfac:data "myDevice", 97,97,97
```

See example program

**Query Syntax**

```markdown
SYSTem:CONFigure:EDEVice:PMAR:TABLE:CFAC:DATA? <name>
```

**Return Type** Numeric - one number per table segment.
SYSTem:CONFigure:EDEVice:PMAR:TABLE:CFAC:FREQuency <name>,<value> [,value]

(Read-Write) Sets and returns the cal factor frequencies for the power sensor.

**Parameters**

*<name>*  
String - Name of the power meter.

*<value> [,value]*  
Cal factor frequencies in Hz.

**Examples**

```
SYST:CONF:EDEV:PMAR:TAB:CFAC:FREQ "myDevice", 1e7, 1e8, 1e9
```

```
SYSTEM:CONFIGURE:EDevice:PMAR:TABLE:CFAC:FREQuency "myDevice", 5e7, 5e8, 5e9
```

See example program

**Query Syntax**

SYStem:CONFigure:EDEVice:PMAR:TABLE:CFAC:FREQuency? <name>

**Return Type**

Numeric - one number per table segment.

**Default**

Not Applicable

---

SYSTem:CONFigure:EDEVice:PMAR:TABLE:LOSS:DATA <name>,<value> [,value]

(Read-Write) Sets and returns the power loss data for the power sensor.

**Parameters**

*<name>*  
String - Name of the power meter.

*<value> [,value]*  
Loss data in dB. POSITIVE values in dB are interpreted as LOSS. To compensate for gain, use negative values.

For each frequency used with SYST:CONF:EDEV:PMAR:TABLE:LOSS:FREQ, enter a cal factor number between 1 and 100.
SYSTem:CONFigure:EDEVice:PMAR:TABLe:LOSS:DATA "myDevice", .01, .02, .03

See example program

SYSTem:CONFigure:EDEVice:PMAR:TABLe:LOSS:FREQuency <name>, <value>[,value]

(Read-Write) Sets and returns frequencies for the power loss data.

Parameters

- <name>  String - Name of the power meter.
- <value>[,value]  Power Loss frequencies in Hz.

Examples  SYST:CONF:EDEV:PMAR:TABL:LOSS:FREQ "myDevice", 1e7, 1e8, 1e9
  system:configure:edevice:pmar:table:loss:frequency "myDevice", 5e7, 5e8, 5e9
  See example program

Query Syntax  SYSTem:CONFigure:EDEVice:PMAR:TABLe:LOSS:FREQuency? <name>

Return Type  Numeric - one number per table segment.

Default  Not Applicable

SYSTem:CONFigure:EDEVice:PMAR:TABLe:LOSS:STATe <name>, <value>
(Read-Write) Sets and returns whether to use the power loss table.

**Parameters**

- `<name>`  String - Name of the power meter.
- `<value>`  Boolean. State of the power loss table. Choose from:
  - OFF or 0 - Power loss table not used.
  - ON or 1 - Power loss table used.

**Examples**

```
```

See example program

**Query Syntax**

SYSTem:CONFigure:EDEVice:PMAR:TABLe:LOSS:STATe? <name>

**Return Type**

Boolean

**Default**

OFF

---

**SYSTem:CONFigure:EDEVice:PMAR:TABLe:RFACtor <name>,<value>**

(Read-Write) Sets and returns the reference cal factor for the power sensor.

Note: If the sensor connected to the power meter contains cal factors in EPROM (such as the Agilent E-series power sensors), those will be the cal factors used. The reference cal factor value associated with this command, and any cal factors entered into the PNA for that sensor channel, will not be used.

**Parameters**

- `<name>`  String - Name of the power meter.
- `<value>`  Reference cal factor in percent. Choose any number between 1 and 150.

**Examples**

```
SYST:CONF:EDEV:PMAR:TABL:RFAC "myDevice", 1
system:configure:edevice:pmar:table:rfactor "myDevice", 1
```

See example program

**Query Syntax**

SYSTem:CONFigure:EDEVice:PMAR:TABLe:RFACtor? <name>

**Return Type**

Numeric
Last Modified:

14-Oct-2010  Added note to IOConfig
24-Mar-2010  Fixed configure and added Remove
15-Jan-2010  Added "AG" to driver example
8-Dec-2009  Changed IOEnable default state and other edits
28-Mar-2009  MX New topic (9.0)
Trigger Commands

Controls the starts or ends of data acquisition.

<table>
<thead>
<tr>
<th>TRIGger:</th>
</tr>
</thead>
<tbody>
<tr>
<td>AUX</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>CHANnel:AUX</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

| CHANnel:ROUTE |
|  | INPut |
|  | READY |

| DELay |

| PREFerence |
|  | AIGLobal |

| READY:POLarity |

| [SEQUence] |
|  | LEVEL |
|  | SCOPe |
|  | SOURce |

| SLOPe |
Click on a blue keyword to view the command details.

Red commands are superseded.

See Also

- Example program Triggering the PNA
- See other SCPI Triggering commands
- Learn about External / Aux Triggering
- Synchronizing the PNA and Controller
- SCPI Command Tree

**TRIGger:AUXiliary:COUNt?**

(Read-only) Returns the number of AUX trigger input / output connector pairs in the instrument.

- PNA-X = 2
- E836xB and PNA-L models = 1

**Parameters**

**Examples**

```
TRIG:AUX:COUN?
```

```
trigger:auxiliary:count?
```

**Return Type** Numeric

**Default** Not Applicable

**TRIGger:CHANnel<ch>:AUXiliary<n>:DELay <num>**
**(Read-Write)** Specifies the delay that should be applied by the PNA after the Aux trigger input is received and before the acquisition is made.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connectors used to send or receive signals.

  - PNA-X - choose from 1 *(AUX TRIG 1 IN)* or 2 *(AUX TRIG 2 IN)*
  
  - All other models: choose 1.

  If unspecified, value is set to 1.

- `<num>` Delay value in seconds. Choose a value between 0 and 3.0 seconds.

**Examples**

```
TRIG:CHAN:AUX:DEL .5
```

```
trigger:channel2:aux2:delay 1.5
```

**Query Syntax**

`TRIGger:CHANnel<ch>:AUXiliary<n>:DELay?`

**Return Type** Numeric

**Default** 0

---

**TRIGger:CHANnel<ch>:AUXiliary<n>:DURation <num>**

**(Read-Write)** Specifies the width of the output pulse, which is the time that the Aux trigger output will be asserted.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.

  - PNA-X - choose from 1 *(AUX TRIG 1)* or 2 *(AUX TRIG 2)*
  
  - All other models: choose 1. *(BNC Trig I/O)*

  If unspecified, value is set to 1.

- `<num>` Duration value in seconds. Choose a value between 1us (1E-6) and 1

**Examples**

```
TRIG:CHAN:AUX:DUR .1
```

```
trigger:channel2:aux2:duration .01
```

**Query Syntax**

`TRIGger:CHANnel<ch>:AUXiliary<n>:DURation?`
**TRIGger:CHANnel<ch>:AUXiliary<n>[:ENABle] <bool>**

(Read-Write) Turns ON / OFF the trigger output.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.
  - PNA-X - choose from 1 (AUX TRIG 1) or 2 (AUX TRIG 2)
  - All other models: choose 1. (BNC Trig I/O)
  If unspecified, value is set to 1.
- `<bool>` ON (or 1) - turns trigger output ON.
  OFF (or 0) - turns trigger output OFF.

**Examples**

```
TRIG:CHAN:AUX 1
trigger:channel2:aux2:enable off
```

**Query Syntax**

TRIGger:CHANnel<ch>:AUXiliary<n>[:ENABle]?

**Return Type** Boolean

**Default** OFF

**TRIGger:CHANnel<ch>:AUXiliary<n>:HANDshake <bool>**
(Read-Write) Turns handshake ON / OFF.

To enable handshake, the main trigger enable must also be set using `TRIG:CHAN:AUX:ENAB`.

When ON, PNA waits indefinitely for the input line to be asserted before continuing with the acquisition. When OFF, the PNA acquires data without waiting.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.
  - PNA-X - choose from 1 (AUX TRIG 1) or 2 (AUX TRIG 2)
  - All other models: choose 1. (BNC Trig I/O)

  If unspecified, value is set to 1.
- `<bool>` ON (or 1) - turns handshaking ON.
  
  OFF (or 0) - turns handshaking OFF.

**Examples**

```
TRIG:CHAN:AUX:HAND 1
trigger:channel2:aux2:handshake off
```

**Query Syntax**

`TRIGger:CHANnel<ch>:AUXiliary<n>:HANDshake?`

**Return Type**

Boolean

**Default**

OFF

```
TRIGger:CHANnel<ch>:AUXiliary<n>:INTerval <char>
```
(Read-Write) Specifies how often a trigger output signal is sent.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.

- PNA-X - choose from 1 (AUX TRIG 1) or 2 (AUX TRIG 2)
- All other models: choose 1. (BNC Trig I/O)

If unspecified, value is set to 1.

- `<char>` Choose from:
  - **POINT** Trigger signal is sent every data point. (effectively the same as **Point sweep**)
  - **SWEep** Trigger signal is sent once every sweep.

**Examples**

```
TRIG:CHAN:AUX:INT POI
```

```
trigger:channel2:aux2:interval sweep
```

**Query Syntax**

TRIGger:CHANnel<ch>:AUXiliary<n>:INTerval?

**Return Type** Character

**Default** SWEep

---

**TRIGger:CHANnel<ch>:AUXiliary<n>:IPOLarity  <char>**

(Read-Write) Specifies the polarity of the trigger IN signal to which the PNA will respond.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.

- PNA-X - choose from 1 (AUX TRIG 1) or 2 (AUX TRIG 2)
- All other models: choose 1. (BNC Trig I/O)

If unspecified, value is set to 1.

- `<char>` Choose from:
  - **POSitive** PNA responds to leading edge or HIGH level
- **NEGative**  PNA responds to trailing edge or LOW level.

Set Edge or Level triggering using `TRIG:CHAN:AUX:TYPE`

**Examples**

<table>
<thead>
<tr>
<th>TRIG:CHAN:AUX:IPOL POS</th>
</tr>
</thead>
<tbody>
<tr>
<td>trigger:channel2:aux2:ipolarity negative</td>
</tr>
</tbody>
</table>

**Query Syntax**

TRIGger:CHANnel<ch>:AUXiliary<n>:IPOLarity?

**Return Type**

Character

**Default**

NEGative

---

**TRIGger:CHANnel<ch>:AUXiliary<n>:OPOLarity <char>**

(Read-Write)  Specifies the polarity of the Aux Output signal being supplied by the PNA.

**Parameters**

- `<ch>`  Any existing channel number. If unspecified, value is set to 1.
- `<n>`  Rear panel connector used to send or receive signals.

- PNA-X - choose from 1  *(AUX TRIG 1)*  or 2  *(AUX TRIG 2)*
- All other models: choose 1. *(BNC Trig I/O)*

If unspecified, value is set to 1.

- `<char>`  Choose from:

  - **POSitive**  PNA sends positive going pulse.
  - **NEGative**  PNA sends negative going pulse.

**Examples**

<table>
<thead>
<tr>
<th>TRIG:CHAN:AUX:OPOL NEG</th>
</tr>
</thead>
<tbody>
<tr>
<td>trigger:channel2:aux2:opolarity positive</td>
</tr>
</tbody>
</table>

**Query Syntax**

TRIGger:CHANnel<ch>:AUXiliary<n>:OPOLarity?

**Return Type**

Character

**Default**

NEGative

---

**TRIGger:CHANnel<ch>:AUXiliary<n>:POSition <char>**
(Read-Write) Specifies whether the aux trigger out signal is sent BEFore or AFTer the acquisition.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.
  - PNA-X - choose from 1 *(AUX TRIG 1)* or 2 *(AUX TRIG 2)*
  - All other models: choose 1. *(BNC Trig I/O)*

If unspecified, value is set to 1.

- `<char>` Choose from:
  - **BEFore** Use if the external device needs to be triggered before the data is acquired, such as a power meter.
  - **AFTer** Use if the external device needs to be triggered just after data has been acquired, such as an external source. This could be more efficient since it allows the external device to get ready for the next acquisition at the same time as the PNA.

**Examples**

```
TRIG:CHAN:AUX:POS BEF
```

```
trigger:channel2:aux2:position after
```

**Query Syntax**

TRIGger:CHANnel<ch>:AUXiliary<n>:POSition?

**Return Type**

Character

**Default**

AFTer

```
TRIGger:CHANnel<ch>:AUXiliary<n>:TYPE <char>
```
(Read-Write) Specifies the type of Aux input detection that the PNA will employ.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<n>` Rear panel connector used to send or receive signals.
  - PNA-X - choose from 1 (AUX TRIG 1) or 2 (AUX TRIG 2)
  - All other models: choose 1. (BNC Trig I/O)

If unspecified, value is set to 1.

- `<char>` Choose from:
  - **EDGE** PNA responds to the leading edge of a signal
  - **LEVEL** PNA responds to the level (HIGH or LOW) of a signal

**Examples**

```
TRIG:CHAN:AUX:TYPE EDGE
trigger:channel2:aux2:type level
```

**Query Syntax**

```
TRIGger:CHANnel<ch>:AUXiliary<n>:TYPE?
```

**Return Type**
Character

**Default**
EDGE

---

**TRIGger:CHANnel<ch>:ROUTE:INPut <char>**

(Read-Write) Specifies the connector to use for the external trigger input. (Similar to **CONTROL:SIGNal <conn>**)

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<char>` Choose from:
  - **MAIN** - meas trig ready
  - **MATH** - material handler
  - **AUXT** - auxio (PNA-L and E836xB ONLY)

**Examples**

```
TRIG:CHAN:ROUT:INPut MAIN
trigger:channel2:route:input main
```

**Query Syntax**

```
TRIGger:CHANnel<ch>:ROUTE:INPut?
```

2900
**Return Type** Character

**Default** PNA model dependent

---

**TRIGger:CHANnel<ch>:ROUTE:READy <char>**

*(Read-Write)* Specifies the connector to use for the external trigger OUT ready line.

**Parameters**

- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<char>` Choose from:
  - **MAIN** - Meas trig ready
  - **MATH** - Material handler pin 21
  - **AUXT** - AUX I/O pin 18 (PNA-L and E836xB ONLY)

**Examples**

```
TRIG:CHAN:ROUT:READ main
trigger:channel2:route:ready auxt
```

**Query Syntax**

`TRIGger:CHANnel<ch>:ROUTE:READy?`

**Return Type** Character

**Default** PNA model dependent

---

**TRIGger:DELay <num>**

*(Read-Write)* Sets and reads the trigger delay for ALL channels (globally). This delay is only applied while **TRIG:SOURce** = EXTernal and **TRIG:SCOP** = ALL. After an external trigger is applied, the start of the sweep is held off for an amount of time equal to the delay setting plus any inherent latency.

To apply a trigger delay for the specified channel ONLY, use **SENS:SWE:TRIG:DELay**

**Parameters**

- `<num>` Delay value in seconds. Choose from 0 to 107.

**Examples**

```
TRIG:DEL .0003
```

Sets the trigger delay to 300 microseconds. The sweep will not start until approximately 300 microseconds after an external trigger is applied.

**Query Syntax**

`TRIGger:DELay?`

**Return Type** Numeric
**TRIGger:PREFerence:AIGLobal <bool>**

(Read-Write) Sets the Trigger OUT behavior to either Global or Channel. [Learn more about this setting.](#)

This command will cause the PNA to Preset.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

To send this command using the PNA front panel, open the **GPIB Command Processor Console**, then type either of the following examples at the command prompt. Then type the Query Syntax and press enter to be sure the PNA took the command.

**Parameters**

<bool>  Choose from:

- **ON** (or 1) - Trigger properties apply to ALL channels (Global).
  - Allows use of **CONT:SIGNal** command to configure the external trigger properties.
  - "Per Point" trigger property is not settable. Use the channel's **Point trigger** setting.
  - Default setting for E836xB and PNA-L models.

- **OFF** (or 0) - External Trigger properties apply to each channel independently.
  - Must use **TRIG:CHAN:AUX** commands to configure the external trigger properties. **CONT:SIGNal** will NOT work.
  - "Per Point" trigger output property is set using the channel's **Point trigger** setting **AND** **TRIG:CHAN:AUX:INTerval**.
  - Default setting for PNA-X models.

**Examples**

```
TRIG:PREF:AIGL 1
```

```
trigger:preference:aiglobal 0
```

**Query Syntax**

TRIGger:PREFerence:AIGLobal?

**Return Type**  Boolean

---

**Default**  0
**Default**  
1 - E836xB and PNA-L models  
0 - PNA-X models

**TRIGger:READy:POLarity <char>**  
(Read-Write) Specifies the polarity of Ready for Trigger output.

All existing Ready for Trigger outputs for PNA-X and PNA-L models are configured simultaneously with this command. [See Capabilities Summary](#).

The Ready for Trigger polarity can NOT be configured for E836x models.

**Parameters**
- `<ch>` Any existing channel number. If unspecified, value is set to 1.
- `<char>`
  - **LOW** - Outputs a TTL low when the PNA is ready for trigger.
  - **HIGH** - Outputs a TTL high when the PNA is ready for trigger.

**Examples**
- `TRIG:READ:POL HIGH`
- `trigger:ready:polarity low`

**Query Syntax**
- `TRIGger:READy:POLarity?`

**Return Type**
- Character

**Default** Low

---

**TRIGger[:SEQuence]:LEVel <char> - Superseded**

This command is replaced with [CONTrol:SIGNal](#).

(Read-Write) Triggers either on a High or Low level trigger signal. This setting only has an effect when TRIG:SOURce EXTERNAL is selected.

**Parameters**
- `<char>` Choose from:
  - **HIGH** - analyzer triggers on TTL High
  - **LOW** - analyzer triggers on TTL Low

**Examples**
- `TRIG:LEV HIGH`
- `trigger:sequence:level low`

**Query Syntax**
- `TRIGger[:SEQuence]:LEVel?`
**TRIGger[:SEQuence]:SCOPe <char>**

(Read-Write) Specifies whether a trigger signal is sent to all channels or only the current channel.

See [Triggering the PNA using SCPI](#).

**Parameters**

<char> Choose from:

- **ALL** - trigger signal is sent to all channels. Also sets SENS:SWEep:TRIG:POINT OFF on ALL channels.
- **CURRENT** - trigger signal is sent to only one channel at a time. With each trigger signal, the channel is incremented to the next triggerable channel.

**Examples**

TRIG:SCOP ALL
TRIGGER:SEQUENCE:SCOPE CURRENT

**Query Syntax**

TRIGger[:SEQuence]:SCOPe?

**Return Type** Character

**Default** ALL

---

**TRIGger[:SEQuence]:SLOPe <char>**

(Read-Write) Specifies the polarity expected by the external trigger input circuitry. Also specify TRIG:TYPE (Level |Edge).

See [Triggering the PNA using SCPI](#).

**Parameters**

<char> Choose from:

- **POSitive** (rising Edge) or High Level
- **NEGative** (falling Edge) or Low Level

**Examples**

TRIG:SLOP NEG
TRIGGER:SEQUENCE:SLOPE POSITIVE

**Query Syntax**

TRIGger[:SEQuence]:SLOPe?

**Return Type** Character
**TRIGger[:SEQuence]:SOURce <char>**

(Read-Write) Sets the source of the sweep trigger signal. This command is a super-set of
INITiate:CONTinuous which can NOT set the source to External. To configure external triggering,
see CONTrol:SIGNal.

See Triggering the PNA using SCPI.

**Parameters**

<char> Choose from:

- **EXTernal** - external (rear panel) source.
- **IMMediate** - internal source sends continuous trigger signals
- **MANual** - sends one trigger signal when manually triggered from the front panel or INIT:IMM is sent.

**Examples**

| TRIG:SOUR EXT trigger:sequence:source immediate |

**Query Syntax**

TRIGger[:SEQuence]:SOURce?

**Return Type**

Character

**Default** IMMEDIATE

---

**TRIGger:TYPE <char>**

(Read-Write) Specifies the type of EXTERNAL trigger input detection used to listen for signals on
the Ext I/O Trig and Meas Trig IN connectors. Edge triggers are most commonly used.

**Parameters**

<ch> Any existing channel number. If unspecified, value is set to 1.

<char> Choose from:

- **EDGE** PNA responds to the rising and falling edge of a signal.
- **LEVel** PNA responds to a level (HIGH or LOW).

Use TRIG:SLOPe to specify Rising or falling - High or Low.
### Examples

<table>
<thead>
<tr>
<th>TRIG:TYPE EDGE</th>
</tr>
</thead>
</table>

| trigger:type level |

### Query Syntax

TRIGger:TYPE?

### Return Type

Character

### Default

EDGE

---

Last modified:

- 28-Sep-2009  Fixed CHAN:AUXiliary commands
- 11-Feb-2009  Added TRIG:SLOPe
- 14-Mar-2008  Added READy:POL command
- 22-Feb-2008  Clarified AIGL command
- 24-Apr-2007  Clarified trigger source and scope
- 15-Feb-2007  MX Updated for AUX triggering
Catalog Measurements using SCPI

This Visual Basic Program does the following:

- Catalogs the currently defined measurements, windows, and traces
- Selects a measurement for further definition
- Adds a Title to the window

To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

```vbs
Dim Meas as String
Dim Win as String
Dim Trace as String

' Read the current measurements in Channel 1
GPIB.Write "CALCulate1:PARameter:CATalog?"
Meas = GPIB.Read
MsgBox ("Ch1 Measurements: " & Meas)

' Read the current windows
GPIB.Write "DISPlay:CATalog?"
Win = GPIB.Read
MsgBox ("Windows: " & Win)

' Read current traces in window 1
GPIB.Write "DISPlay:WINDow1:CATalog?"
Trace = GPIB.Read
MsgBox ("Traces in Window1: " & Win)
```
Channels, Windows, and Measurements using SCPI

SOURce and most SENSSe commands act on the channel that is specified in the command. Channel 1 is default if not specified.

Most DISPPlay commands act on the window and trace specified in the command. Window1 and Trace1 are default if not specified.

CALCulate commands act on the selected measurement in the specified channel. Select the measurement for each channel using CALCulate<channel number>:PARameter:SELect <meas name>. You can select one measurement in each channel.

The following Visual Basic program does the following:

- Presets the analyzer
- Create 2 windows
- Create 2 Measurements
- Feed the measurements to windows / traces
- Change frequency ranges for channels
- Select both measurements
- Turn marker 1 ON for each measurement

To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

GPIB.Write "SYSTem:PRESet"

'Create Measurements
GPIB.Write "CALCulate1:PARameter:DEFine:EXT 'Meas1',S11"
GPIB.Write "CALCulate2:PARameter:DEFine:EXT 'Meas2',S21"

'Turn on windows - creates if new
GPIB.Write "DISPlay:WINDow1:STATE ON"
GPIB.Write "DISPlay:WINDow2:STATE ON"

'Associate ("FEED") the measurement name('Meas1') to WINDow(1), and give the new TRACe a number(1).
GPIB.Write "DISPlay:WINDow1:TRACe1:FEED 'Meas1'"
GPIB.Write "DISPlay:WINDow2:TRACe2:FEED 'Meas2'"

'Change each channel's frequency range
GPIB.Write "SENSe1:FREQuency:SPAN 1e9"
GPIB.Write "SENSe2:FREQuency:SPAN 2e9"
Select both measurements
GPIB.Write "CALCulate1:PARameter:SELect 'Meas1'"
GPIB.Write "CALCulate2:PARameter:SELect 'Meas2'"

Turn marker 1 ON for each measurement
GPIB.Write "CALCulate1:MARKer:STATE ON"
GPIB.Write "CALCulate2:MARKer:STATE ON"
PNA as Controller and Talker / Listener

This Visual Basic Program uses VISA to do the following:

- Control the PNA using a VISA LAN Client interface on the PNA.
- Control another instrument using the PNA as GPIB controller.
- Queries both the analyzer and other instrument to identify themselves with *IDN?

Note: This program can be modified to work from a remote PC to control both instruments. In that case, set up the PNA to be a talker/listener.

To run this program, you need to do the following:

- Add module visa32.bas to the VB project. It is located on the analyzer at C:/Program Files/HP/VXIPNP/WINNT/Include/VISA32.bas
- Configure the PNA for VISA / SICL
- Set up the PNA to be GPIB system controller.
  1. On the System menu, point to Configure. Click SICL / GPIB
  2. Click System Controller
- Connect another instrument to the analyzer through a GPIB cable with Primary address of 13 on GPIB0 interface

See Other SCPI Example Programs

Sub main()

' This application run from onboard the PNA
' can control both the PNA and another GPIB instrument.
'
' To run this program the module visa32.bas must be added
' to the project.

'VISA function status return code
Dim status As Long
' Session to Default Resource Manager
Dim defRM As Long
' Session to instrument
Dim viPNA As Long
' Session to other GPIB instrument
Dim viInstrument As Long
' String to hold results
Dim strRes As String * 200
On Error GoTo ErrorHandler

status = viOpenDefaultRM(defRM)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler

'Open the session to the PNA
status = viOpen(defRM, "GPIB1::16::INSTR", 0, 0, viPNA)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler

'Ask for the PNA's ID.
status = viVPrintf(viPNA, "*IDN?" + Chr$(10), 0)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler

'Read the ID as a string.
status = viVScanf(viPNA, "%t", strRes)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler
'Display the results
MsgBox "PNA is: " + strRes

'Open the session to the other instrument
status = viOpen(defRM, "GPIB0::13::INSTR", 0, 0, viInstrument)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler

'Ask for the instrument's ID.
status = viVPrintf(viInstrument, "*IDN?" + Chr$(10), 0)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler

'Read the ID as a string.
status = viVScanf(viPNA, "%t", strRes)
If (status < VI_SUCCESS) Then GoTo VisaErrorHandler
'Display the results
MsgBox "Other instrument is: " + strRes
'Close the resource manager session (which closes everything)
Call viClose(defRM)
End

ErrorHandler:
'Display the error message
MsgBox "*** Error : " + Error$, MB_ICONEXCLAMATION
End

VisaErrorHandler:
Dim strVisaErr As String * 200
Call viStatusDesc(defRM, status, strVisaErr)
MsgBox "*** Error : " + strVisaErr
End
End Sub
Create a Balanced Measurement using SCPI

This example program does the following:

- creates several Balanced measurements in separate windows
- generates markers
- calculates statistics
- sets limit lines and queries results
- queries a measurement to determine if we have a balanced parameter and what type it is.

**Note:** By their nature, balanced measurements are extremely sensitive to phase differences between the two RF paths that make up the balanced port, especially at higher frequencies. A good calibration (not performed in this example) is critical to achieving good balanced measurement results.

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Balanced.vbs. Learn how to setup and run the macro.

### See Other SCPI Example Programs

```vbscript
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' A comment
scpi.Parse("SYST:FPRESET")

' This example uses DUT topology Bal-Bal -
' a DUT with a balanced input and balanced output.

' Port mapping for our DUT:
' logical port 1 = physical ports 1 and 4
' logical port 2 = physical ports 2 and 3
' The default is:
' logical port 1 = physical ports 1 and 2
' logical port 2 = physical ports 3 and 4

' logical 1 logical 2
```
' Turn on Four windows
scpi.Parse("DISP:WIND1:STATe ON")
scpi.Parse("DISP:WIND2:STATe ON")
scpi.Parse("DISP:WIND3:STATe ON")
scpi.Parse("DISP:WIND4:STATe ON")
' Create a trace called "sdd21", and for that trace turn on the balanced transformation and set the balanced transformation to BBAL SDD21.
scpi.Parse("CALC:PAR:DEF:EXT ","sdd21"",S11")
scpi.Parse("CALC:PAR:SEL ","sdd21""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")
' Feed the sdd21 trace to window 1, trace 1
scpi.Parse("DISP:WIND1:TRAC1:FEED ","sdd21""")
' Similarly create 3 more balanced transmission/conversion parameters
' Create Scd21
scpi.Parse("CALC:PAR:DEF:EXT ","scd21"",S11")
scpi.Parse("CALC:PAR:SEL ","scd21""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")
scpi.Parse("DISP:WIND1:TRAC2:FEED ","scd21""")
' Create Sdc21
scpi.Parse("CALC:PAR:DEF:EXT ","sdc21"",S11")
scpi.Parse("CALC:PAR:SEL ","sdc21""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")
scpi.Parse("DISP:WIND1:TRAC3:FEED ","sdc21""")
' Create Scc21
scpi.Parse("CALC:PAR:DEF:EXT ","scc21"",S11")
scpi.Parse("CALC:PAR:SEL ","scc21""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")
scpi.Parse("DISP:WIND1:TRAC4:FEED ","scc21""")
' Now create logical port 1 reflection parameters, and place them in window 2
scpi.Parse("CALC:PAR:DEF:EXT ""sdd11"",S11")
scpi.Parse("CALC:PAR:SEL ""sdd11""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
' Feed the sdd11 trace to window 2, trace 1
scpi.Parse("DISP:WIND2:TRAC1:FEED ""sdd11"""")
' Similarly create 3 more balanced reflection/conversion parameters
scpi.Parse("CALC:PAR:DEF:EXT ""scd11"",S11")
scpi.Parse("CALC:PAR:SEL ""scd11""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND2:TRAC2:FEED ""scd11"""")
scpi.Parse("CALC:PAR:DEF:EXT ""sdc11"",S11")
scpi.Parse("CALC:PAR:SEL ""sdc11""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND2:TRAC3:FEED ""sdc11"""")
scpi.Parse("CALC:PAR:DEF:EXT ""scc11"",S11")
scpi.Parse("CALC:PAR:SEL ""scc11""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND2:TRAC4:FEED ""scc11"""")
' Now create reverse transmission parameters, and place them in window 3
scpi.Parse("CALC:PAR:DEF:EXT ""sdd12"",S11")
scpi.Parse("CALC:PAR:SEL ""sdd12""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
' Feed the sdd11 trace to window 3, trace 1
scpi.Parse("DISP:WIND3:TRAC1:FEED ""sdd12"""")
' Similarly create 3 more balanced reverse transmission/conversion parameters
scpi.Parse("CALC:PAR:DEF:EXT ""scd12"",S11")
scpi.Parse("CALC:PAR:SEL ""scd12""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SCD12")
scpi.Parse("DISP:WIND3:TRAC2:FEED ""scd12"""")
`scpi.Parse("CALC:PAR:DEF:EXT ""sdc12"",S11")
scpi.Parse("CALC:PAR:SEL ""sdc12""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("CALC:PAR:DEF:EXT ""scc12"",S11")
scpi.Parse("CALC:PAR:SEL ""scc12""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND3:TRAC4:FEED ""scc12""")

' Now create reverse reflection parameters, and place them in window 4

scpi.Parse("CALC:PAR:DEF:EXT ""sdd22"",S11")
scpi.Parse("CALC:PAR:SEL ""sdd22""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")

' Feed the sdd11 trace to window 3, trace 1
scpi.Parse("DISP:WIND4:TRAC1:FEED ""sdd22""")

' Similarly create 3 more balanced reverse reflection parameters

scpi.Parse("CALC:PAR:DEF:EXT ""scd22"",S11")
scpi.Parse("CALC:PAR:SEL ""scd22""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND4:TRAC2:FEED ""scd22""")

scpi.Parse("CALC:PAR:DEF:EXT ""sdc22"",S11")
scpi.Parse("CALC:PAR:SEL ""sdc22""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND4:TRAC3:FEED ""sdc22""")

scpi.Parse("CALC:PAR:DEF:EXT ""scc22"",S11")
scpi.Parse("CALC:PAR:SEL ""scc22""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATE ON")
scpi.Parse("DISP:WIND4:TRAC4:FEED ""scc22""")

scpi.Parse("CALC:FSIM:BAL:DEVICE BBALANCED")
scpi.Parse("CALC:FSIM:BAL:TOPOLOGY BBAL:PPORTS 1,4,2,3")

' Set up stimulus
scpi.Parse("SENS:SWE:POINts 801")
scpi.Parse("SENS:FREQ:STARt 10e6")
scpi.Parse("SENS:FREQ:STOP 1e9")
 ' Here we demonstrate how to determine if we have
 ' a balanced parameter and what type it is.
 ' Read back one parameter to verify its type
scpi.Parse("CALC:PAR:SEL ""sdd21""")
' Is this a balanced parameter?
isbal = scpi.Parse("CALC:FSIM:BAL:PAR?")
' Which topology/device is set?
device = scpi.Parse("CALC:FSIM:BAL:DEV?")
device = Left( device, Len(device)-1 ) ' strip off newline
' Which parameter are we measuring within that topology?
balparam = Left( balparam, Len(balparam)-1 ) ' strip off newline
If isbal Then
WScript.Echo "Balanced Parameter: " & balparam & " in topology: " & device & "."
Else
WScript.Echo "Parameter not balanced."
End If
Create a Measurement using SCPI

This VBScript program creates a new S21 measurement and displays it on the PNA screen.
The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.
This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as NewMeas.vbs. Learn how to setup and run the macro.

See Other SCPI Example Programs

Dim app
Dim scpi
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' A comment
'Preset the analyzer
scpi.Execute ("SYST:FPReset")
' Create and turn on window 1
scpi.Execute ("DISPlay:WINDow1:STATE ON")
'Define a measurement name, parameter
scpi.Execute ("CALCulate:PARameter:DEFine:EXT 'MyMeas',S21")
'Associate ("FEED") the measurement name ('MyMeas') to WINDow (1), and give the new TRACe a number (1).
scpi.Execute ("DISPlay:WINDow1:TRACe1:FEED 'MyMeas'")
Create a Narrowband Point-in-Pulse Measurement using the PNA-X

The following SCPI example demonstrates how to create a Narrowband Point-in-Pulse measurement using the Pulsed Application DLL on the PNA-X.

It first gets valid configuration settings and then uses those settings to configure the PNA and internal pulsed generators.

To run this program, you need:

- PNA-X
- Pulsed Application (Option H08)

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as PulseProfile.vbs. Learn how to setup and run the macro.

Note: Because of the long length of some commands in this example, word wrapping may occur when copying. These lines require modification after pasting.

See Also

- Install and register the pulsed .dll on your PC.
- ConfigEnhancedNB2 method for sending and returning parameters to the .dll.
- ConfigEnhancedNBIIFAtten method for setting the receiver IF gain.
- SCPI IF Configuration commands used in the program.
- Other Pulse SCPI examples

```csharp
//This example shows you how to perform point in pulse measurement based on
//PNA-X in narrowband mode using SCPI commands.

public partial class Form1 : Form
{
    private object pna;
    private object scpi;
    private Type srvtype;
    private AgilentPNAPulsed.applicationClass pulseApp;

    public Form1()
    {
        InitializeComponent();
    }
}```
private string sendScpiCommand(string scpitext)
{
    object[] parameter = new object[1];
    parameter[0] = scpitext;
    return (string)srvtype.InvokeMember("parse", BindingFlags.InvokeMethod, null, scpi, parameter);
}

private void ConnectToPNA()
{
    srvtype = Type.GetTypeFromProgID("AgilentPNA835x.Application", true);
pna = Activator.CreateInstance(srvtype);
    scpi = srvtype.InvokeMember("ScpiStringParser", BindingFlags.GetProperty, null, pna, null);
}

private void NBButton_Click(object sender, EventArgs e)
{
    double dPRF = 10000, dBW = 500; //PRF=10kHz
double dPhysicalIF = 0, dNCO = 0, dClockFreq = 0;
    System.Array aStage1TapArray = null, aStage2TapArray = null, aStage3TapArray = null;
    bool bFixedPRF = true;
double dGateDelay = 0.000002, dGateWidth = 0.000005; //Gate width=50ns
double dSWGateDelay = 0, dSWGateWidth = 0;
    int iSWGateRamp = 0;
double dModPulseWidth = 0.000001; //10 us
double dModPulseDelay = 0; //0us
    short myAtten = 0;
pulseApp = new AgilentPNAPulsed.applicationClass();
    ConnectToPNA();
    //Preset PNA-X
    sendScpiCommand("*RST");
    //Measure S21
    sendScpiCommand("DISP:WIND:TRAC1:DEL");
    sendScpiCommand("CALCulate:PARameter:DEFine:EXT /'MyMeas/',S21");
    sendScpiCommand("DISP:WIND:TRAC1:FEED /'MyMeas/'");
Set power leveling mode to Openloop
sendScpiCommand("sour:pow1:alc:mode open");

//Send desired pulsed parameters to the pulsed configuration DLL.
//The DLL will return a new set of pulse parameters to send to the
PNA-X.
pulseApp.ConfigEnhancedNB2(ref dPRF, ref dBW, ref dPhysicalIF, ref
dNCO, ref dClockFreq, ref aStage1TapArray,
    ref aStage2TapArray, ref aStage3TapArray,
bFixedPRF, dGateDelay, dGateWidth,
    ref dSWGateDelay, ref dSWGateWidth, ref
iSWGateRamp);

double pulsePeriod = 1 / dPRF;
//Pulse #1 as modulation source
sendScpiCommand("sens:puls:per " + pulsePeriod.ToString()); // 100us
//Set Puls1 width
sendScpiCommand("sens:puls1:width " + dModPulseWidth.ToString());
//10us
//Set Puls1 delay
sendScpiCommand("sens:puls1:delay " + dModPulseDelay.ToString());
//10us
//Turn on Puls1
sendScpiCommand("SENS:PULS1:STAT 1");
//Set modulation source to Puls1
sendScpiCommand("sens:path:conf:elem /"PulseModDrive","Pulse1"/";
//Enable pulse modulator 1
sendScpiCommand("sens:path:conf:elem
/"Src1Out1PulseModEnable","Enable"/";
//Pulse #2 controls receiver gate
sendScpiCommand("sens:puls2:width " + dGateWidth.ToString()); //50ns
sendScpiCommand("sens:puls2:delay " + dGateDelay.ToString()); //0
sendScpiCommand("SENS:PULS2:STAT 1");
sendScpiCommand("sens:path:conf:elem /"IFGateA","Pulse2"/";
sendScpiCommand("sens:path:conf:elem /"IFGateB","Pulse2"/";
sendScpiCommand("sens:path:conf:elem /"IFGateR1","Pulse2"/";
sendScpiCommand("sens:path:conf:elem /"IFGateR2","Pulse2"/";
//Set IFBW
sendScpiCommand("SENS:BWID " + dBW.ToString());
//Configure IF path
sendScpiCommand("sens:path:conf:elem /"IFSigPathAll/","NBF/"");
sendScpiCommand("SENS:IF:FILT:AUTO 0");
sendScpiCommand("SENS:IF:FREQ:AUTO 0");
sendScpiCommand("SENS:IF:FREQ " + dPhysicalIF.ToString());
// Set filter stages based on pulse parameters
sendScpiCommand("SENS:IF:FILT:STAGe1:FREQ " + dNCO.ToString());
// Convert Stage1TapArray to string
string buf1 = new string(' ', 1000);
for (int i = 0; i < aStage1TapArray.GetLength(0); i++)
{
    buf1 = buf1 + aStage1TapArray.GetValue(i).ToString() + "", ";
}
buf1 = buf1.Trim();
buf1 = buf1.Substring(0, buf1.Length - 1);
// Convert Stage1TapArray to string
string buf2 = new string(' ', 1000);
for (int j = 0; j < aStage2TapArray.GetLength(0); j++)
{
    buf2 = buf2 + aStage2TapArray.GetValue(j).ToString() + "", ";
}
buf2 = buf2.Trim();
buf2 = buf2.Substring(0, buf2.Length - 1);

// Set IF Filter Stage1 and Stage2 Coeficent
sendScpiCommand("SENS:IF:FILT:STAG1:COEF " + buf1);
sendScpiCommand("SENS:IF:FILT:STAG2:COEF " + buf2);
if (dSWGateWidth == 0) // No valid SW gate
{
    sendScpiCommand("SENS:IF:FILT:STAG3:TYPE 'RECT'");
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'C','" + aStage3TapArray.GetValue(0).ToString());
    sendScpiCommand("SENS:PULS0:STAT 0");
}
else
{
    sendScpiCommand("SENS:IF:FILT:STAGe3:TYPE 'PWIN'");
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'C','" + aStage3TapArray.GetValue(0).ToString());
}
sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'P'," +
pulsePeriod.ToString());
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'D'," +
dSWGateDelay.ToString());
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'W'," +
dSWGateWidth.ToString());
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'R'," +
iSWGateRamp.ToString());
        double pulse0Width = 1 / dClockFreq;
    sendScpiCommand("sens:puls0:width " + pulse0Width.ToString());
    sendScpiCommand("sens:puls0:delay 0");
    sendScpiCommand("SENS:PULS0:STAT 1");
}
pulseApp.ConfigEnhancedNBIFAtten(dPRF, dGateWidth, ref myAtten);
    sendScpiCommand("sens:path:conf:elem /"NBFATNA/",/" +
myAtten.ToString() + "/"");
    sendScpiCommand("sens:path:conf:elem /"NBFATNB/",/" +
myAtten.ToString() + "/"");
    sendScpiCommand("sens:path:conf:elem /"NBFATNR1/",/" +
myAtten.ToString() + "/"");
    sendScpiCommand("sens:path:conf:elem /"NBFATNR2/",/" +
myAtten.ToString() + "/"");
        //Set start and stop frequency
    sendScpiCommand("SENS:FREQ:STAR 1000000000");
    sendScpiCommand("SENS:FREQ:STOP 2000000000");
        //Single Sweep
    sendScpiCommand("SENS:SWE:MODE SING");
    sendScpiCommand("DISP:WIND:TRAC:Y:AUTO");
}
Create a Narrowband Pulse Profile Measurement using the PNA-X

The following SCPI example demonstrates how to create a Narrowband Pulse Profile measurement using the Pulsed Application DLL on the PNA-X.

It first gets valid configuration settings and then uses those settings to configure the PNA and internal pulsed generators.

To run this program, you need:

- PNA-X
- Pulsed Application (Option H08)

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as PulseProfile.vbs. Learn how to setup and run the macro.

Note: Because of the long length of some commands in this example, word wrapping may occur when copying. These lines require modification after pasting.

See Also

- Install and register the pulsed .dll on your PC.
- ConfigEnhancedNB2 method for sending and returning parameters to the .dll.
- ConfigEnhancedNBIFAtten method for setting the receiver IF gain.
- SCPI IF Configuration commands used in the program.
- Other Pulse SCPI examples

```vbnet
public partial class Form1 : Form
{
    private object pna;
    private object scpi;
    private Type srvtype;
    private AgilentPNAPulsed.applicationClass pulseApp;
    public Form1()
    {
        InitializeComponent();
    }
    private string sendScpiCommand(string scpitext)
    {
        object[] parameter = new object[1];
    }
```
parameter[0] = scpitext;
    return (string)srvtype.InvokeMember("parse",
        BindingFlags.InvokeMethod, null, scpi, parameter);
};

private void ConnectToPNA()
{
    srvtype = Type.GetTypeFromProgID("AgilentPNA835x.Application", true);
    pna = Activator.CreateInstance(srvtype);
    scpi = srvtype.InvokeMember("ScpiStringParser",
        BindingFlags.GetProperty, null, pna, null);
}

private void NBGPIBbutton_Click(object sender, EventArgs e)
{
    double dPRF = 10000, dBW = 500; //PRF=10kHz
    double dPhysicalIF = 0, dNCO = 0, dClockFreq = 0;
    System.Array aStage1TapArray = null, aStage2TapArray = null,
    aStage3TapArray = null;
    bool bFixedPRF = true;
    double dGateDelay = 0.0, dGateWidth = 0.00000005; //Gate width=50ns
    double dSWGateDelay = 0, dSWGateWidth = 0;
    int iSWGateRamp = 0;
    double dModPulseWidth = 0.00001; //10 us
    double dModPulseDelay = 0.00001; //10us
    short myAtten = 0;
    pulseApp = new AgilentPNAPulsed.applicationClass();
    ConnectToPNA();
    //Preset PNA-X
    sendScpiCommand("*RST");
    //Measure S21
    sendScpiCommand("DISP:WIND:TRAC1:DEL");
    sendScpiCommand("CALCulate:PARameter:DEFine:EXT /'MyMeas/',S21");
    sendScpiCommand("DISP:WIND:TRAC1:FEED /'MyMeas/'");
    //Set power leveling mode to Openloop
    sendScpiCommand("sour:pow1:alc:mode open");
    //Send desired pulsed parameters to the pulsed configuration DLL.
    //The DLL will return a new set of pulse parameters to send to the
    PNA-X.
pulseApp.ConfigEnhancedNB2(ref dPRF, ref dBW, ref dPhysicalIF, ref dNCO, ref dClockFreq, ref aStage1TapArray, ref aStage2TapArray, ref aStage3TapArray, bFixedPRF, dGateDelay, dGateWidth, ref dSWGateDelay, ref dSWGateWidth, ref iSWGateRamp);

double pulsePeriod = 1 / dPRF;
//Pulse #1 as modulation source
sendScpiCommand("sens:puls:per " + pulsePeriod.ToString()); // 100us
//Set Pulsel width
sendScpiCommand("sens:puls1:width " + dModPulseWidth.ToString());
//10us
//Set Pulsel delay
sendScpiCommand("sens:puls1:delay " + dModPulseDelay.ToString());
//10us
//Turn on Pulsel
sendScpiCommand("SENS:PULS1:STAT 1");
//Set modulation source to Pulsel
sendScpiCommand("sens:path:conf:elem /"PulseModDrive/",/"Pulse1/"");
//Enable pulse modulator 1
sendScpiCommand("sens:path:conf:elem /"Src1Out1PulseModEnable/",/"Enable/"");
//Pulse #2 controls receiver gate
sendScpiCommand("sens:puls2:width " + dGateWidth.ToString()); //50ns
sendScpiCommand("sens:puls2:delay " + dGateDelay.ToString()); //0
sendScpiCommand("SENS:PULS2:STAT 1");
sendScpiCommand("sens:path:conf:elem /"IFGateA/",/"Pulse2/""");
sendScpiCommand("sens:path:conf:elem /"IFGateB/",/"Pulse2/""");
sendScpiCommand("sens:path:conf:elem /"IFGateR1/",/"Pulse2/""");
sendScpiCommand("sens:path:conf:elem /"IFGateR2/",/"Pulse2/""");
//Set IFBW
sendScpiCommand("SENS:BWID "+dBW.ToString());
//Configure IF path
sendScpiCommand("sens:path:conf:elem /"IFSigPathAll/",/"NBF/"");
sendScpiCommand("SENS:IF:FILT:AUTO 0");
sendScpiCommand("SENS:IF:FREQ:AUTO 0");
sendScpiCommand("SENS:IF:FREQ "+dPhysicalIF.ToString());
// Set filter stages based on pulse parameters
sendScpiCommand("SENS:IF:FILT:STAGe1:FREQ " + dNCO.ToString());

// Convert Stage1TapArray to string
string buf1 = new string(' ', 1000);
for (int i = 0; i < aStage1TapArray.GetLength(0); i++)
{
    buf1 = buf1 + aStage1TapArray.GetValue(i).ToString() + ",";
}
buf1 = buf1.Trim();
buf1 = buf1.Substring(0, buf1.Length - 1);

// Convert Stage2TapArray to string
string buf2 = new string(' ', 1000);
for (int j = 0; j < aStage2TapArray.GetLength(0); j++)
{
    buf2 = buf2 + aStage2TapArray.GetValue(j).ToString() + ",";
}
buf2 = buf2.Trim();
buf2 = buf2.Substring(0, buf2.Length - 1);

// Set IF Filter Stage1 and Stage2 Coeficent
sendScpiCommand("SENS:IF:FILT:STAG1:COEF " + buf1);
sendScpiCommand("SENS:IF:FILT:STAG2:COEF " + buf2);

if (dSWGateWidth == 0) // No valid SW gate
{
    sendScpiCommand("SENS:IF:FILT:STAG3:TYPE 'RECT'");
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'C'," + aStage3TapArray.GetValue(0).ToString());
    sendScpiCommand("SENS:PULS0:STAT 0");
}
else
{
    sendScpiCommand("SENS:IF:FILT:STAGe3:TYPE 'PWIN'");
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'C'," + aStage3TapArray.GetValue(0).ToString());
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'D'," + dSWGateDelay.ToString());
sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'W'," + dSWGateWidth.ToString());
    sendScpiCommand("SENS:IF:FILT:STAGe3:PAR 'R'," + iSWGateRamp.ToString());
    double pulse0Width=1/dClockFreq;
    sendScpiCommand("sens:puls0:width " + pulse0Width.ToString());
    sendScpiCommand("sens:puls0:delay 0");
    sendScpiCommand("SENS:PULS0:STAT 1");
}

pulseApp.ConfigEnhancedNBIFAtten(dPRF, dGateWidth, ref myAtten);
    sendScpiCommand("sens:path:conf:elem NBFATNA/" + myAtten.ToString() + "/");
    sendScpiCommand("sens:path:conf:elem NBFATNB/" + myAtten.ToString() + "/");
    sendScpiCommand("sens:path:conf:elem NBFATNR1/" + myAtten.ToString() + "/");
    sendScpiCommand("sens:path:conf:elem NBFATNR2/" + myAtten.ToString() + "/");
    //Run pulse profile using below several lines
    //Set CW Mode
    sendScpiCommand("SENS:SWE:TYPE CW");
    double startTime, stopTime, stepTime;
    int myProfilePoints;
    startTime = 0.00001;//10us
    stopTime = 0.00005;//50us
    stepTime = 0.00000005;//50ns
    myProfilePoints = (int)((stopTime - startTime) / stepTime) + 1;
    sendScpiCommand("SENS:SWE:POIN " + myProfilePoints.ToString());
    // Test gates
    sendScpiCommand("sens:puls1:delay " + startTime.ToString());
    sendScpiCommand("SENS:PULS2:DINC " + stepTime.ToString());
    //Single Sweep
    sendScpiCommand("SENS:SWE:MODE SING");
    sendScpiCommand("DISP:WIND:TRAC:Y:AUTO");
}
Last Modified:

2-Jul-2008   New topic.
Configure a PMAR Device

This VB Script program configures a new Power Meter as Receiver device and creates a trace using the PMAR. Learn more about Power Meter as a Receiver

These programs can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as PMAR.vbs. Learn how to setup and run the macro.

See all External Device Configuration commands

See Other SCPI Example Programs

```vbs
' This section gets the PNA application
' starts the scpi parser, and presets the PNA
dim app
Set app = CreateObject("AgilentPNA835x.Application")
set scpi = app.ScpiStringParser
scpi.parse "*rst"
scpi.parse "Syst:conf:edev:add 'newpmar1'
scpi.parse "Syst:conf:edev:dtype 'newpmar1', 'Power Meter'
scpi.parse "Syst:conf:edev:ioconfig 'newpmar1', 'gpib0::14::instr'
scpi.parse "Syst:conf:edev:pmar:sens 'newpmar1', 1"
scpi.parse "Syst:conf:edev:pmar:read:ntolerance 'newpmar1', 0.1"
scpi.parse "Syst:conf:edev:pmar:sens 'newpmar1', 1"
scpi.parse "Syst:conf:edev:pmar:fmin 'newpmar1', 100000000"n
scpi.parse "Syst:conf:edev:pmar:fmax 'newpmar1', 10000000000"
scpi.parse "Syst:conf:edev:pmar:flim 'newpmar1', 0"
scpi.parse "Syst:conf:edev:pmar:tabl:rfac 'newpmar1', 100"
scpi.parse "Syst:conf:edev:pmar:tabl:cfac:freq 'newpmar1', 1e9, 2e9, 3e9"
scpi.parse "Syst:conf:edev:pmar:tabl:loss:freq 'newpmar1', 1e9, 2e9, 3e9"
'Activate and enable the PMAR external device
scpi.parse "Syst:conf:edev:ioen 'newpmar1', 1"
scpi.parse "Syst:conf:edev:stat 'newpmar1', 1"
'Create a PMAR trace with power meter connected to port 3
```
'Use Calc:Par:Def:Ext - NOT CALC:PAR:DEF!!

scpi.parse "CALC:PAR:DEF:EXT 'myPMARTrace', 'newpmar1,3'"
Create a Swept IMDX Measurement

This program configures several Swept IMDx parameters using power sweep. In this configuration, tone power is swept from -20 dBm to -5 dBm while the Input, LO, and Output frequencies are fixed as follows:

- Input center freq= 2.50 GHz
- Tone Delta freq = 10 MHz (f1 = 2.495 GHz and f2 = 2.505 GHz).
- LO freq = 2.00 GHz
- Output freq = 4.50 GHz

This program also allows you to optionally load a .mxr file to perform mixer setup.

To run this program without error, an external source named 'PSG' must be connected to drive the LO.

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as IMDX.vbs. Learn how to setup and run the macro.

See all Swept IMD commands.
See all Mixer Setup commands.

---

See Other SCPI Example Programs

---

```vbs
Dim app
Dim scpi
Dim err

'Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

'Preset the system
scpi.parse "SYST:FPR"
scpi.parse "DISP:WIND1:STAT ON"

'Create an IMDX measurement in Channel 1 and display it as trace 1 in window 1
scpi.parse "CALC1:CUST:DEF 'ch1IMDX', 'Swept IMD Converters', 'PwrMain'
scpi.parse "DISP:WIND1:TRAC1:FEED 'ch1IMDX'"

'Put the channel in trigger hold and setup all the mixer parameters
```
scpi.parse "SENS1:SWE:MODE HOLD"
scpi.parse "SENS1:IMD:SWE:TYPE POW"
scpi.parse "SENS1:MIX:INP:FREQ:MODE FIXED"
scpi.parse "SENS1:MIX:LO:FREQ:MODE FIXED"
scpi.parse "SENS1:MIX:OUTP:FREQ:MODE FIXED"
scpi.parse "SENS1:MIX:INP:FREQ:FIX 2500000000"
scpi.parse "SENS1:MIX:LO:FREQ:FIX 2000000000"
scpi.parse "SENS1:MIX:OUTP:FREQ:SID HIGH"
scpi.parse "SENS1:MIX:LO:NAM 'PSG'"
scpi.parse "SENS1:MIX:LO:POW 10"

'Do an Apply to make sure all the settings are properly transferred to the channel
scpi.parse "SENS1:MIX:CALC OUTP"
scpi.parse "SENS1:MIX:APPL"

'Optionally, put the channel in hold and load an existing .mxr file with all the mixer settings
'scpi.parse "SENS1:SWE:MODE HOLD"
'scpi.parse "SENS1:MIX:LOAD 'C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/IMD/Ch1.mxr'"

'Make additional IMD settings
scpi.parse "SENS1:IMD:TPOW:COUP:STAT ON"
scpi.parse "SENS1:IMD:TPOW:F1:STAR -20"
scpi.parse "SENS1:IMD:TPOW:F1:STOP -5"
scpi.parse "SENS1:IMD:FREQ:DFR:CW 10000000"
scpi.parse "SENS1:SWE:POIN 201"
scpi.parse "SENS1:imd:ifbw:main 1000"
scpi.parse "SENS1:imd:ifbw:imt 500"
scpi.parse "SOUR1:POW2:AMPL -5"

'Create additional measurements in the channel
scpi.parse "CALC1:CUST:DEF 'ch1IMDX2', 'Swept IMD Converters', 'IM3'"
scpi.parse "DISP:WIND1:TRAC2:FEED 'ch1IMDX2'"
scpi.parse "CALC1:CUST:DEF 'ch1IMDX3', 'Swept IMD Converters', 'OIP3'"
scpi.parse "DISP:WIND1:TRAC3:FEED 'ch1IMDX3'"
scpi.parse "CALC1:CUST:DEF 'ch1IMDX4', 'Swept IMD Converters', 'IIP3'"
scpi.parse "DISP:WIND1:TRAC4:FEED 'ch1IMDX4'"
scpi.parse "CALC1:CUST:DEF 'ch1IMDX5', 'Swept IMD Converters', 'ToneGain'"
scpi.parse "DISP:WIND1:TRAC5:FEED 'ch1IMDX5'"

'Take a single sweep to apply all stimulus changes
scpi.parse "*cls;*ese 1"
scpi.parse "sens1:swe:mode SING;*OPC?"

'Check for errors
err=scpi.parse ("SYST:ERR?")
MsgBox(err)
Create a Wideband Pulsed Measurement using the PNA-X

This Visual Basic example shows you how to configure the PNA-X internal pulse generators and modulators to make wideband pulsed measurements in pulse profile mode using the PNA-X.

Visit the PNA website where you can download a free Wideband Pulsed Application that performs this measurement on the PNA-X.

See all SCPI Pulsed examples

Private Sub Form_Load()
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

'Preset the analyzer
scpi.Execute ("*RST")

'Set BW to 5 MHz
scpi.Execute ("SENS:BWID 5MHZ")

'Set sweep type to CW mode
scpi.Execute ("SENS:SWE:TYPE CW")

'Delete S11 trace
scpi.Execute ("DISP:WIND:TRAC1:DEL")

'Create S21 trace
scpi.Execute ("CALCulate:PARameter:DEFine:EXT 'MyMeas',S21")
scpi.Execute ("DISP:WIND:TRAC1:FEED 'MyMeas'")

'Set modulation source to Pulse1
scpi.Execute ("sens:path:conf:elem 'PulseModDrive','Pulse1'")

'Set power leveling mode to Openloop
scpi.Execute ("sour:pow1:alc:mode open")

'Enable pulse modulator 1
scpi.Execute ("sens:path:conf:elem 'Src1Out1PulseModEnable','Enable'")

'Set clock of internal pulse generator to internal
scpi.Execute ("sens:path:conf:elem 'PulseTrigInput','Internal'")

'Turn on Pulse0
scpi.Execute ("SENS:PULS0:STAT 1")
'Turn on Pulse1
scpi.Execute ("SENS:PULS1:STAT 1")
' Set pulse period to 1 ms
scpi.Execute ("sens:puls:per .001")
' Set Pulse1 width to 10 us
scpi.Execute ("sens:puls1:width 0.00001")
' Set Pulse1 delay to 8 us
scpi.Execute ("sens:puls1:delay 0.000008")
' Set Pulse0 width to 1 us
scpi.Execute ("sens:puls0:width 0.000001")
' Set Pulse0 delay to 400 ns
scpi.Execute ("sens:puls0:delay 0.0000004")
' Set trigger scope to Channel
scpi.Execute ("TRIG:SCOP CURRENT")

End Sub
Create an FOM Measurement

All three VBScript examples in this topic create a FOM measurement with the following attributes:

- Sweep the Source (input) from 1 GHz to 2 GHz
- Sweep the Receivers (output) from 2 GHz to 3 GHz
- You provide an LO at 1 GHz

Learn more about Frequency Offset Mode

These programs can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as FOM.vbs. Learn how to setup and run the macro.

See Other SCPI Example Programs

The following example will run on any PNA model with FOM (opt 080). However, these commands have no provisions for internal second source. It uses Sens:Offset commands introduced before 'enhanced FOM' was released for the A.07.10 release.

```
' This section gets the PNA application
' starts the scpi parser, and presets the PNA
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.Execute("SYST:FPRESET")
' Create and turn on window 1
scpi.Execute("DISPlay:WINDow1:STATE ON")
' Define a measurement name, parameter
scpi.Execute("CALCulate:PARameter:DEFine:EXT 'MyMeas',S21")
' Associate ("FEED") the measurement name ("MyMeas") to WINDow (1)
' and give the new TRACe a number (1).
scpi.Execute("DISPlay:WINDow1:TRACe1:FEED 'MyMeas'")

scpi.Execute("SENS:FREQ:START 1e9")
scpi.Execute("SENS:FREQ:STOP 2e9")
' set the receivers to be 2e9 -> 3e9
scpi.Execute("SENS:OFFS:OFFS 1e9")
scpi.Execute("SENS:OFFS ON")
```
The following example can be run ONLY on a PNA with revision A.07.10 or later and has FOM (opt 080). It uses new Sens:FOM commands.

```vba
' This section gets the PNA application
' starts the scpi parser, and presets the PNA
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.Execute("SYST:FPRESET")
' Create and turn on window 1
scpi.Execute("DISPlay:WINDow1:STATE ON")
' Define a measurement name, parameter
scpi.Execute("CALCulate:PARameter:DEFine 'MyMeas',S21")
' Associate ("FEED") the measurement name ('MyMeas') to WINDow (1), and give the new TRACe a number (1).
scpi.Execute("DISPlay:WINDow1:TRACe1:FEED 'MyMeas'")
scpi.Execute("SENS:FREQ:START 1e9")
scpi.Execute("SENS:FREQ:STOP 2e9")
' set the receivers to be 2e9 -> 3e9
scpi.Execute("SENS:FOM:RANG3:FREQ:OFFS 1e9")
scpi.Execute("SENS:OFFS ON")
```

The following example can be run ONLY on a PNA with a second internal source, has revision A.07.10 or later, and has FOM (opt 080). It uses the internal 2nd source for the fixed LO frequency.

```vba
' This section gets the PNA application
' starts the scpi parser, and presets the PNA
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.Execute("SYST:FPRESET")
' Create and turn on window 1
scpi.Execute("DISPlay:WINDow1:STATE ON")
' Define a measurement name, parameter
scpi.Execute("CALCulate:PARameter:DEFine 'MyMeas',S21")
' Associate ("FEED") the measurement name ('MyMeas') to WINDow (1), and give the new TRACe a number (1).
scpi.Execute("DISPlay:WINDow1:TRACe1:FEED 'MyMeas'")
```
scpi.Execute("SENS:FREQ:START 1e9")
scpi.Execute("SENS:FREQ:STOP 2e9")
' set the receivers to be 2e9 -> 3e9
scpi.Execute("SENS:FOM:RANG3:FREQ:OFFS 1e9")
' setup the 2nd source frequencies
scpi.Execute("SENS:FOM:RANG4:COUP 0")
scpi.Execute("SENS:FOM:RANG4:FREQ:START 1e9")
scpi.Execute("SENS:FOM:RANG4:FREQ:STOP 1e9")
' turn off coupling
scpi.Execute("SOUR:POW:COUP 0")
' set LO power to 10dBm
scpi.Execute("SOUR:POW3 10")
'turn ON port 3, our LO signal
scpi.Execute("SOUR:POW3:MODE ON")
scpi.Execute("SENS:FOM:STAT ON")
Create an iTMSA Measurement

This example program does the following:

- Create an iTMSA Balanced Sdd21 measurement
- Set sweep type = power
- Set phase offset on balanced port 1=180 degrees

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Balanced.vbs. Learn how to setup and run the macro.

See Other SCPI Example Programs

```vbs
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' Reset the system
scpi.Parse("SYST:FPRESET")

' This example uses DUT topology Bal-Bal -
' a DUT with a True-mode balanced input and balanced output.

' Port mapping for our DUT:
' logical port 1 = physical ports 1 and 3
' logical port 2 = physical ports 2 and 4

' logical 1          logical 2
' 1 ------|   DUT   |------ 2 +
' 3 ------|_________|------ 4 -

' Turn on a window
scpi.Parse("DISP:WIND1:STATE ON")

' Create a trace called "sdd21", and for that trace turn on the balanced
```
transformation and set the balanced transformation to BBAL SDD21.

scpi.Parse("CALC:PAR:DEF:EXT ""sdd21"",S11")

' Feed the sdd21 trace to window 1, trace 1
scpi.Parse("DISP:WIND1:TRAC1:FEED ""sdd21""")
scpi.Parse("CALC:PAR:SEL ""sdd21""")

' Set the topology of measurement
scpi.Parse("CALC:FSIM:BAL:DEVice BBALanced")
scpi.Parse("CALC:FSIM:BAL:TOPology:BBAL:PPORts 1,3,2,4")

' Set up stimulus
scpi.Parse("SENS:SWE:POINts 801")
scpi.Parse("SENS:FREQ:STARt 10e6")
scpi.Parse("SENS:FREQ:STOP 1e9")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")

' Recall a 4-port Cal Set or perform a 4-port Cal here

' Set the sweep type to power sweep
scpi.Parse("SENS:SWE:TYPE POWer")

' Set iTMSA parameters
**Create an SMC Fixed Output Measurement**

This VB Script example creates a calibrated SMC fixed output measurement using an external, controlled LO. Then a single sweep is taken and data is retrieved. The external LO is NOT required when using the internal second PNA source for the LO.

Requirements:

- The external LO should be configured to match the SENS:MIX:LO:NAME command below.

Fixed output measurements require that an external LO source be swept and synchronized with the PNA source. FCA performs this synchronization using the external source configuration settings. See Configure an External Source using SCPI.

The fastest, and recommended, method of controlling the LO source is Hardware List (BNC) triggering mode. However, in this mode, FCA channels will not respond to manual triggers. Therefore, the example uses the following mechanism to trigger a sweep:

```plaintext
Write "SENS:SWE:MODE HOLD"   'place channel 1 in HOLD mode
Write "INIT:CONT ON"         'place PNA in internal trigger mode
Write "SENS:SWE:MODE SINGle  'place channel 1 in HOLD mode
Write "*OPC?"                'wait until the sweep is complete
Read
```

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You can run a VBScript (*.vbs) program from the PNA using Macros. To run this program, copy the following code into a text editor and save it as a *.vbs file.

```vbs
option explicit
' Setup infrastructure to use the SCPI over COM
dim app
set app = createobject("Agilentpna835x.application")
dim p
set p = app.scpistringparser
dim returnStr
sub Write (command)
    if len(returnStr) <> 0 then
        err.Raise 55,"Write","Query Unterminated"
    end if
    returnStr = p.parse(command)
end sub
sub WriteIgnoreError(command)
    returnStr = p.Execute(command)
p.Parse("SYST:ERR?")   ' clear error queue
end sub
```
function Read
    if len(returnStr) = 0 then
        err.Raise 55, "Read", "Bad read"
    end if
    Read = returnStr
    returnStr = ""
end function

Write "SYST:PRES"
' When programming in remote mode, hold mode is recommended
Write "SENS:SWE:MODE HOLD"
' Delete the standard measurement
Write "CALC:PAR:DEL:ALL"
' Create an SC21 measurement
Write "CALC:CUST:DEF 'MySC21', 'Scalar Mixer/Converter', 'SC21'"
Write "DISP:WIND:TRACE:FEED 'MySC21'"
Write "CALC:PAR:SEL 'MySC21'"
' Set number of points to 11
Write "SENS:SWE:POIN 11"
' Setup the mixer parameters for a swept LO, fixed output measurement
Write "SENS:MIX:INP:FREQ:START 200e6"
Write "SENS:MIX:INP:FREQ:STOP 700e6"
Write "SENS:MIX:LO:FREQ:MODE Swept"
Write "SENS:MIX:OUTPUT:FREQ:FIX 3.4e9"
Write "SENS:MIX:OUTP:FREQ:SID HIGH"
Write "SENS:MIX:INP:POW -17"
Write "SENS:MIX:LO:POW 10"
' Specify the LO name, for controlled LO.
' This name is setup in the External Source Config Dialog
Write "SENS:MIX:LO:NAME '8360'"
' The CALC method calculates the LO frequency from the other parameters,
' It also applies the mixer parameters to the channel.
Write "SENS:MIX:LO:CALC LO_1"
' Create an S11 in the same channel
Write "CALC:CUST:DEF 'MyS11', 'Scalar Mixer/Converter', 'S11'"
Write "DISP:WIND:TRACE2:FEED 'MyS11'"
Write "CALC:PAR:SEL 'MyS11'"
' Create an IPwr in the same channel
Write "CALC:CUST:DEF 'MyIPwr', 'Scalar Mixer/Converter', 'IPwr'"
Write "DISP:WIND:TRACE3:FEED 'MyIPwr'"
Write "CALC:PAR:SEL 'MyIPwr'"

' Create an OPwr in the same channel
Write "CALC:CUST:DEF 'MyOPwr', 'Scalar Mixer/Converter', 'OPwr'"
Write "CALC:PAR:SEL 'MyOPwr'"
Write "DISP:WIND:TRACE4:FEED 'MyOPwr'"

' Perform a single sweep, synchronously. When *OPC returns, the sweep is done
Write "SENS:SWE:MODE SINGle"
Write "*OPC?"
Read

' Retrieve the SC21 data
Write "CALC:PAR:SEL 'MySC21'"
Write "CALC:DATA? SDATA"

  dim data
data = Read()
wscript.echo("SC21=" & data)

' Retrieve the S11 data
Write "CALC:PAR:SEL 'MyS11'"
Write "CALC:DATA? SDATA"
data = Read()
wscript.echo("S11=" & data)
Create and Cal a GCA Measurement

This VBS program does the following:

- creates and configures GCA to perform a SMART Sweep
- performs a calibration using an ECal with 3.5 mm Female on Port A and 3.5 mm Male connectors on Port B

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as GCA.vbs. Learn how to setup and run the macro.

See the Gain Compression commands

```vbscript
option explicit

dim CompLevel , Tolerance , StartFreq , StopFreq , NumFREQs , Scale , LinearPower

dim AcqMode , BackOff , StartPower , StopPower , NumPowers , EnableInterp , CompAlg

dim DwellTime , IFBandwidth , ShowIterations , host , app , parser

'' GCA Settings/Values
''
'' Acquisition Mode:
'' naSmartSweep = 0
'' naSweepPowerAtEachFreq2D = 1
'' naSweepFreqAtEachPower2D = 2
''
'' Compression Algorithm
'' naCompressionFromLinearGain = 0
'' naCompressionFromMaximumGain = 1
'' naBackoffCompression = 2
'' naXYCompression = 3
''
'' EndOfSweepOperation
'' naDefaultPowerSet = 0
'' naSetToStartPower = 1
'' naSetToStopPower = 2
'' naSetRFOff = 3
''
CompLevel = 1 ' 1 dB compression level
Tolerance = 0.05 ' SMART Sweep tolerance
```
StartFreq = 1E9
StopFreq = 9E9
NumFreqs = 201
Scale = 0.1
LinearPower = -20
BackOff = 10 ' Not used for Deviation from linear gain
StartPower = -20
StopPower = 8
NumPowers = 60 ' Not used for SMART Sweep
DwellTime = 0.0005 ' Allow some time for DUT bias/thermal effects
IFBandwidth = 1000 ' Reasonable trace noise at -20 dBm
EnableInterp = False ' Disable interpolation
AcqMode = 0 ' Smart Sweep
CompAlg = 0 ' Deviation from linear gain
ShowIterations = False ' Configure SMART to not show iteration results
dim objargs
set objargs = wscript . Arguments
if (objArgs.Count = 1) then host = objargs (0)

set app = CreateObject("Agilentpna835x.application")
set parser = app . ScpiStringParser
call SetupGCA ( parser,_
    StartFreq,_,
    StopFreq,_,
    NumFreqs,_,
    EnableInterp,_,
    Scale,_,
    CompLevel,_,
    LinearPower,_,
    AcqMode,_,
    BackOff,_,
    StartPower,_,
    StopPower,_,
    NumPowers,_)
CompAlg, _,
DwellTime, _
IFBAndwidth, _
ShowIterations )
call CalGCA ( parser )
call Analysis( parser )

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'' GCA Setup
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
sub SetupGCA ( parser , StartFreq , StopFreq , NumFreqs , EnableInterp , Scale ,
CompLevel , LinearPower , _
AcqMode , BackOff , StartPower , StopPower , NumPowers , CompAlg ,
DwellTime , IFBAndwidth , _
ShowIterations )
parser . Parse "*RST"
parser . Parse "CALC:PAR:DEL:ALL"
parser . Parse "CALC:CUST:DEF "S21","Gain Compression","S21""
parser . Parse "DISP:WIND:TRAC1:FEED "S21"
parser . Parse "CALC:CUST:DEF "CompIn21","Gain Compression","CompIn21"
parser . Parse "DISP:WIND:TRAC2:FEED "CompIn21"
parser . Parse "CALC:CUST:DEF "DeltaGain21","Gain Compression","DeltaGain21"
parser . Parse "DISP:WIND:TRAC3:FEED "DeltaGain21"
parser . Parse "SENS:SWE:MODE HOLD"
parser . Parse "DISP:WIND1:TRAC3:Y:SCAL:PDIV " & Scale
parser . Parse "DISP:WIND1:TRAC3:Y:RLEV " & -CompLevel
select case AcqMode
  case 0 ' SMART Sweep
    parser . Parse "SENS:GCS:AMOD SMAR"
  case 1 ' 2D Power Sweeps
    parser . Parse "SENS:GCS:AMOD PFREQ"
  case 2 ' 2D Freq Sweeps
    parser . Parse "SENS:GCS:AMOD FPow"
end select
select case CompAlg
  case 0 ' Deviation from linear gain

2946
parser . Parse "SENS:GCS:COMP:ALG CFLG"

**case 1** ' Deviation from max gain
parser . Parse "SENS:GCS:COMP:ALG CFMG"

**case 2** ' Back Off
parser . Parse "SENS:GCS:COMP:ALG BACK"

**case 3** ' XY
parser . Parse "SENS:GCS:COMP:ALG XYCOM"

end select

**if** EnableInterp **then**
parser . Parse "SENS:GCS:COMP:INT ON"
**else**
parser . Parse "SENS:GCS:COMP:INT OFF"
**end if**

**if** ShowIterations **then**
parser . Parse "SENS:GCS:SMAR:SIT ON"
**else**
parser . Parse "SENS:GCS:SMAR:SIT OFF"
**end if**

parser . Parse "SENS:GCS:COMP:LEV " & CompLevel
parser . Parse "SENS:GCS:COMP:BACK:LEV " & BackOff
parser . Parse "SENS:GCS:COMP:DELT:X " & BackOff
parser . Parse "SENS:GCS:SMAR:STIM " & DwellTime
parser . Parse "SENS:BAND " & IFBandwidth
parser . Parse "SENS:SWE:DWEL " & DwellTime
parser . Parse "SOUR:POW:STAR " & StartPower
parser . Parse "SOUR:POW:STOP " & StopPower
parser . Parse "SENS:FREQ:STAR " & StartFreq
parser . Parse "SENS:FREQ:STOP " & StopFreq
parser . Parse "SENS:SWE:POIN " & NumFreqs
parser . Parse "SENS:SWE:MODE SING"
dim str
str = parser.Parse("* OPC ?")
end sub

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'' GCA Calibration
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

sub CalGCA ( parser )

dim CalSteps , I
parser . parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 female''"
parser . parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 male''"
parser . parse "SENS:CORR:COLL:GUID:CKIT:PORT1 'N4691-60004 ECal '"
parser . parse "SENS:CORR:COLL:GUID:CKIT:PORT2 'N4691-60004 ECal '"
parser . parse "SENS:CORR:GCSsetup:POW 0"
parser . parse "SENS:CORR:COLL:GUID:INIT "
CalSteps = parser . parse (" SENS:CORR:COLL:GUID:STEP ?")
for I = 1 to CalSteps
    msgBox parser . parse ("SENS:CORR:COLL:GUID:DESC ? " & I )
    parser . parse ("SENS:CORR:COLL:GUID:ACQ STAN"& I )
next
parser . parse "SENS:CORR:COLL:GUID:SAVE "
msgBox "Done"
end sub

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'' GCA Analysis
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

sub Analysis( parser )
'select measurement 1
parser.parse "CALC:PAR:MNUM 1"
pdater.parse "CALC:GCM:ANAL:ENABLE 1" 'turn on the analysis mode
parser.parse "CALC:GCM:ANAL:CWFR 1e9" 'set the analysis cw frequency
'select measurement 2
parser.parse "CALC:PAR:MNUM 2"
pdater.parse "CALC:GCM:ANAL:ENABLE 1"

2948
parser.parse "CALC:GCM:ANAL:CWFR 2e9"

parser.parse "CALC:GCM:ANAL:XAX PSO" 'set the axis to power settings
'select measurement 3

parser.parse "CALC:PAR:MNUM 3"

parser.parse "CALC:GCM:ANAL:ENABLE 1"

parser.parse "CALC:GCM:ANAL:CWFR 3e9"

parser.parse "CALC:GCM:ANAL:ISD 0" ' set the discrete frequency option to false
end sub

Last Modified:

3-Sep-2009  Added Analysis

173-Sep-2009  MX New topic
Create and Cal a GCX Measurement

This VBS program does the following:

- creates and configures GCX
- performs a calibration using an ECal with 3.5 mm Female on Port A and 3.5 mm Male connectors on Port B

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as GCX.vbs. Learn how to setup and run the macro.

See the Gain Compression and Mixer Commands

```vbs
option explicit

dim CompLevel, Tolerance, StartFreq, StopFreq, LOFreq, NumFreqs, Scale, LinearPower

dim AcqMode, BackOff, StartPower, StopPower, NumPowers, EnableInterp, CompAlg

dim DwellTime, IFBandwidth, ShowIterations, host, app, parser

CompLevel         = 1      ' 1 dB compression level
Tolerance         = 0.05   ' SMART Sweep tolerance
StartFreq         = 2.5E9
StopFreq          = 2.6E9
LOFreq = 1.7E9
NumFreqs          = 21
Scale             = 0.1
LinearPower       = -10
BackOff           = 10     ' Not used for Deviation from linear gain
StartPower        = -20
StopPower         = 8
NumPowers         = 60     ' Not used for SMART Sweep
DwellTime         = 0.0005 ' Allow some time for DUT bias/thermal effects
IFBandwidth       = 1000   ' Reasonable trace noise at -20 dBm
EnableInterp      = False  ' Disable interpolation
AcqMode           = 0      ' Smart Sweep
CompAlg           = 0      ' Deviation from linear gain
ShowIterations  = False  ' Configure SMART to not show iteration results

dim objargs
```
set objargs = wscript.Arguments
if (objArgs.Count = 1) then host = objargs(0)

'' Create and Configuration GCX Channel:

set app = CreateObject("Agilentpna835x.application")
set parser = app.ScpiStringParser
call SetupGCAX( parser,_
    StartFreq,_
    StopFreq,_
    LOFreq,_
    NumFreqs,_
    EnableInterp,_
    Scale,_
    CompLevel,_
    LinearPower,_
    AcqMode,_
    BackOff,_
    StartPower,_
    StopPower,_
    NumPowers,_
    CompAlg,_
    DwellTime,_
    IFBAndwidth,_
    ShowIterations )
call CalGCAX( parser )

'' GCX Setup

sub SetupGCAX( parser, StartFreq, StopFreq, LOFreq, NumFreqs, EnableInterp, Scale, CompLevel, LinearPower,_
    AcqMode, BackOff, StartPower, StopPower, NumPowers, CompAlg, DwellTime, IFBAndwidth,_
    ShowIterations )
    parser.Parse "#RST"
    parser.Parse "CALC:PAR:DEL:ALL"
    parser.Parse "CALC:CUST:DEF ">SC21"","Gain Compression Converters",""SC21"" "

parser.Parse "DISP:WIND:TRAC1:FEED "SC21""

parser.Parse "CALC:PAR:SEL "SC21"

parser.Parse "CALC:CUST:DEF "CompIn21","Gain Compression Converters""

parser.Parse "DISP:WIND:TRAC2:FEED "CompIn21"

parser.Parse "CALC:CUST:DEF "DeltaGain21","Gain Compression Converters""

parser.Parse "DISP:WIND:TRAC3:FEED "DeltaGain21"

parser.Parse "SENS:SWE:MODE HOLD"

parser.Parse "DISP:WIND1:TRAC3:Y:SCAL:PDIV " & Scale

parser.Parse "DISP:WIND1:TRAC3:Y:RLEV " & -CompLevel

select case AcqMode
    case 0 ' SMART Sweep
        parser.Parse "SENS:GCS:AMOD SMAR"
    case 1 ' 2D Power Sweeps
        parser.Parse "SENS:GCS:AMOD PFREQ"
    case 2 ' 2D Freq Sweeps
        parser.Parse "SENS:GCS:AMOD FPOW"
end select

select case CompAlg
    case 0 ' Deviation from linear gain
        parser.Parse "SENS:GCS:COMP:ALG CFLG"
    case 1 ' Deviation from max gain
        parser.Parse "SENS:GCS:COMP:ALG CFMG"
    case 2 ' Back Off
        parser.Parse "SENS:GCS:COMP:ALG BACK"
    case 3 ' XY
        parser.Parse "SENS:GCS:COMP:ALG XYCOM"
end select

if EnableInterp then
    parser.Parse "SENS:GCS:COMP:INT ON"
else
    parser.Parse "SENS:GCS:COMP:INT OFF"
end if

if ShowIterations then
parser.Parse "SENS:GCS:SMAR:SIT ON"
else
  parser.Parse "SENS:GCS:SMAR:SIT OFF"
end if

parser.Parse "SENS:GCS:COMP:LEV " & CompLevel
parser.Parse "SENS:GCS:COMP:BACK:LEV " & BackOff
parser.Parse "SENS:GCS:COMP:DELT:X " & BackOff
parser.Parse "SENS:GCS:SMAR:STIM " & DwellTime
parser.Parse "SENS:BAND " & IFBandwidth
parser.Parse "SENS:SWE:DWEL " & DwellTime
parser.Parse "SOUR:POW:STAR " & StartPower
parser.Parse "SOUR:POW:STOP " & StopPower

parser.Parse "SOUR:POW " & LinearPower
parser.Parse "SENS:SWE:POIN " & NumFreqs

'set converter properties
'swept input sweep mode
parser.Parse "SENS:MIX:INP:FREQ:MODE SWEPT"
'fixed lo sweep mode
parser.Parse "SENS:MIX:LO1:NAME ""Port 3""
parser.Parse "SENS:MIX:LO1:FREQ:MODE FIXED"
'swept output sweep mode
parser.Parse "SENS:MIX:OUTP:FREQ:MODE SWEPT"
'input start freq
parser.Parse "SENS:MIX:INP:FREQ:STAR " & StartFreq
'input stop freq
parser.Parse "SENS:MIX:INP:FREQ:STOP " & StopFreq
'lo1 freq
parser.Parse "SENS:MIX:LO1:FREQ:FIX " & LOFreq
'lo1 power
parser.Parse "SENS:MIX:LO1:POW 10"
'calculate output freq
parser.Parse "SENS:MIX:CALC OUTP"
parser.Parse "SENS:SWE:MODE SING"

dim str
str = parser.Parse("*OPC?")

e nd sub

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
Next
Dim calset
'Finish the cal and save the calset
Msgbox("GCX Cal Complete!")
end sub
Create and Cal a Noise Figure Measurement

This example program creates a Noise Figure measurement, then calibrates the measurement. You MUST change the ECal Identification strings (in Blue font).

Optional: Uncomment the following lines (in Blue font) to change these settings:

- Noise Receiver = Noise Receiver to Std (PNA) Receiver
- Cal Method = "Vector" to "Scalar"
- Receiver Characterization Method = "NoiseSource" to "Power Meter"

This VBScript program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as NF.vbs. Learn how to setup and run the macro. See the Noise figure commands.

See Other SCPI Example Programs

```
' This section gets the PNA application
' starts the scpi parser, and presets the PNA
windowNum = 1
channelNum = 1
set pna=CreateObject("AgilentPNA835x.Application")
set scpi = pna.ScpiStringParser
' Create noise figure measurement
scpi.Parse "SYST:FPR"
scpi.Parse "DISP:WIND ON"
scpi.Parse "CALC:CUST:DEF 'noiseFig', 'Noise Figure Cold Source', 'NF'"
scpi.Parse "DISP:WIND:TRAC:FEED 'noiseFig'"
scpi.Parse "CALC:PAR:SEL 'noiseFig'"
' Substitute appropriate Ecal identification strings here
tunerEcal = "N4691-60004 ECal 02821"
pullEcal = "N4691-60004 ECal 02297"
' configure channel
ConfigureChannel
ConfigureNoiseSettings
' perform calibration
SetupNoiseSource
```
SetupCalAttributes_Insertable

FinishCalibration

' ----- Support subroutines ------

' Configure noise channel
sub ConfigureChannel
  scpi.Parse "SENS:FREQ:START 750MHz"
  scpi.Parse "SENS:FREQ:STOP 5.0GHz"
  scpi.Parse "SENS:SWEEP:POINTS 401"
  scpi.Parse "SENS:BWID 1.0E3"
end sub

' Configure noise-specific channel settings
sub ConfigureNoiseSettings
  scpi.Parse "SENS:NOIS:REC NOIsE 'Use noise receivers
'  scpi.Parse "SENS:NOIS:REC NORM 'Use std PNA receiver
  scpi.Parse "SENS:NOIS:AVER:STAT ON" ' turn averaging ON
  scpi.Parse "SENS:NOIS:AVER 40" ' noise averaging
  scpi.Parse "SENS:NOIS:BWID 8MHz" ' noise bandwidth
  scpi.Parse "SENS:NOIS:GAIN 30" ' gain of noise receiver
  scpi.Parse "SENS:NOIS:TEMP:AMB 301" ' ambient temperature, in Kelvin
  scpi.Parse "SENS:NOIS:IMP:COUN 5" ' number of tuner impedance states
  scpi.Parse "SENS:NOIS:TUN:INP 'B'" ' orientation of tuner input port
  scpi.Parse "SENS:NOIS:TUN:OUTP 'A'" ' orientation of tuner output port
  scpi.Parse "SENS:CORR:TCOL:USER:VAL 300" ' noise source cold temperature
end sub

sub SetupCalAttributes_Insertable
  scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 female'
  scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 male'

  scpi.Parse "SENS:NOIS:SOUR:CONN 'APC 3.5 male' ' noise source connector type
  scpi.Parse "SENS:NOIS:SOUR:CKIT ' ' & pullEcal & "' ' noise source calkit
scpi.Parse "SENS:NOISE:CAL:METHOD 'Vector'" ' cal method
' scpi.Parse "SENS:NOISE:CAL:METHOD 'Scalar'"
scpi.Parse "SENS:NOISE:CAL:RMEThod 'NoiseSource' 'Receiver Characterization method
' scpi.Parse "SENS:NOISE:CAL:RMETHod 'PowerMeter'
scpi.Parse "SENS:CORR:COLL:GUID:INIT"
end sub

sub SetupNoiseSource
' specify the ENR file for the noise source
enrfile = "C:/Program Files/Agilent/Network Analyzer/Noise/346C_MY44420454.enr"
scpi.Parse "SENS:NOIS:ENR:FILENAME '' & enrfile & ''"
' set noise source cold temperature
scpi.Parse "SENS:CORR:TCOLd:USER:VAL 301.1"
end sub

sub FinishCalibration
' Build the connection list and acquire the calibration
steps = scpi.Parse("SENS:CORR:COLL:GUID:STEPS?")
for i = 1 to steps
   msgbox str
   scpi.Parse "SENS:CORR:COLL:GUID:ACQ STAN" & i
next
scpi.Parse "SENS:CORR:COLL:GUID:SAVE 0"
wscript.echo "Calibration complete"
end sub

Last Modified:

9-Jun-2011   Updated with Rcvr Char Method A.09.41.
Create and Cal a VMC Measurement

The following example program sets up a 1-stage mixer, then performs a full VMC calibration. By removing the comments (' ') at the start of the BLUE code, it can also do the following:

- Load a mixer setup file
- Use an ECal Module
- Perform manual ECal orientation
- Load a Mixer Characterization
- Perform a Mixer Characterization ONLY

See Also

Setup Converter commands
VMC Cal commands

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as VMC.vbs. Learn how to setup and run the macro.

```
Dim app
Dim scpi
'
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.Parse "SYSTem:PRESet"
'
' Create a Vector Mixer Measurement
' First, delete all measurements on the channel
scpi.Parse "CALC:PAR:DEL:ALL"
'
' Create a forward scalar mixer measurement and configure it in channel 1.
' The first parameter is a unique identifying string to allow subsequent
' commands to be directed at this specific measurement.
scpi.Parse "CALC:CUST:DEF 'My VC21', 'Vector Mixer/Converter', 'VC21'"
'
' Setup the new measurement as the 2nd trace in the active window
scpi.Parse "DISP:WIND:TRAC2:FEED 'My VC21'"
'
' Make the new trace the active measurement
scpi.Parse "CALC:PAR:SEL 'My VC21'"
```
'The parameters of the mixer measurement can now be configured.
'This can be done by either using the SENS:MIX commands
'for each of the parameters or by loading a mixer setup file.
'Uncomment the following line to load a mixer setup file. The path name
'for the mixer file may be loaded from other mapped drives.
'scpa.Parse "SENS:MIXer:Load 'C:/Program Files/Agilent/Network
Analyzer/Documents/Mixer/MyMixer.mxr'"

' Setup Stimulus
' Points and IFBW are channel settings
scpi.Parse "SENS:SWEep:POINts 21"
scpi.Parse "SENS:BANDwidth 1e3"
' The rest are mixer settings
scpi.Parse "SENS:MIX:LO:FREQ:MODE SWEPt"
scpi.Parse "SENS:MIX:INPut:FREQ:STAR 3.6e9"
scpi.Parse "SENS:MIX:INPut:FREQ:STOP 3.9e9"
scpi.Parse "SENS:MIX:LO:FREQ:MODE FIXED"
scpi.Parse "SENS:MIX:LO:FREQ:FIX 1e9"
scpi.Parse "SENS:MIX:LO:POW 10"
scpi.Parse "SENS:MIX:OUTP:FREQ:SID LOW"
scpi.Parse "SENS:MIX:CALC Output"
scpi.Parse "SENS:MIX:LO:NAME 'Port 3'"
scpi.Parse "SENS:MIX:APPLY"

' Perform Cal

' Define the DUT connectors for at ports 1 and 2 of the PNA
scpi.Parse "sens:corr:coll:guid:conn:port1 'APC 3.5 female'"
scpi.Parse "sens:corr:coll:guid:conn:port2 'APC 3.5 male'"
scpi.Parse "sens:corr:coll:guid:conn:port3 'Not used'"
scpi.Parse "sens:corr:coll:guid:conn:port4 'Not used'"

' Specify Mechanical cal kits
scpi.Parse "sens:corr:coll:guid:ckit:port1 '85033D/E'"
scpi.Parse "sens:corr:coll:guid:ckit:port1 '85033D/E'"

' Specify an ECal module the same way
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 ECal"
' Non-factory characterizations are specified as follows:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'
' By default, VMC requires the measurement of a Calibration Mixer.
' To determine the conversion loss of the calmixer, the cal wizard
' will add a step to perform a 1 port cal at the output of the mixer.
' Specify to Perform the mixer characterization.
' To avoid performing this step, provide the wizard with a
' mixer characterization file. Uncomment the following two lines to
' specify the characterization file. This S2P file will be read.
Files\Agilent\Network Analyzer\Documents\MyMixer.s2p"
' OR to perform a mixer characterization ONLY
' uncomment the following two lines.
' The outcome of the calibration is an S2P file.
sypi.Parse "SENS:CORR:COLL:GUID:VMC:OPER 'CHAR"
Files\Agilent\Network Analyzer\Documents\MyMixer.s2p"
'
' ECal orientation
' By default, auto orientation of the ecal module is performed
' Uncomment the following lines to manually orient the ecal
'sypi.Parse "SENS:CORR:PREF:ECAL:ORI OFF"
' for 2-port portion, ecal port A connected to PNA port 1
' for mixer char, ecal port A connected to cal mixer output
' the main calibration loop
' a description for the connection instructions is read
' and then the standard is acquired

dim steps, strPrompt
scpi.Parse "sens:corr:coll:guid:init"
steps=scpi.Parse ("sens:corr:coll:guid:steps?")
wscript.echo "Number of Steps = " + cstr(steps)
if (steps > 0) then ' otherwise an error condition occurred
for i = 1 to steps
    MsgBox strPrompt, vbOKOnly, step
    scpi.Parse ("sens:corr:coll:guid:acq STAN" + CStr(i))
next
MsgBox ("Cal is done!")
end if
Create and Cal a Swept IMD Measurement

This program does the following:

- Create IMD power and IM3 measurements
- Set sweep mode to Center Frequency Sweep
- Calibrate the IMD channel

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as IMD.vbs.

See Also

Learn how to setup and run the macro.
See the IMD commands.
See the IMD Cal commands

See Other SCPI Example Programs

```vbs
option explicit
'declare variables
dim SweepMode, StartDeltaFreq, StopDeltaFreq, NumFreqs, TonePower, CWFreq
dim app, hostname, parser
'' Sweep type:
'' naIMDToneCWSweep = 0
'' naIMDTonePowerSweep = 1
'' naIMDToneCenterFreqSweep = 2
'' naIMDDeltaFrequencySweep = 3
'' naIMDToneSegmentSweep = 4
'init variables
SweepMode = 3 ' Sweep DeltaF
StartDeltaFreq = 100e3
StopDeltaFreq = 1e9
NumFreqs = 201
TonePower = -7
CWFreq = 5e9
' get host name from commandline
```
dim objargs
set objargs = wscript.arguments
if(objargs.Count = 1) then hostname = objargs(0)
set app = CreateObject("Agilentpna835x.application", hostname)
set parser = app.ScpiStringParser

call SetupIMD

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''}

parser.Parse "*RST"
parser.Parse "CALC:PAR:DEL:ALL"
parser.Parse "CALC:CUST:DEF 'PwrMain','Swept IMD', 'PwrMain'"  ' create PwrMain measurement
parser.Parse "DISP:WIND:TRAC1:FEED 'PwrMain'"
parser.Parse "CALC:PAR:SEL 'PwrMain'"
parser.Parse "CALC:CUST:DEF 'IM3', 'Swept IMD', 'IM3'"  ' create IM3 measurement
parser.Parse "DISP:WIND:TRAC2:FEED 'IM3'"
parser.Parse "SENS:SWE:MODE HOLD"

' set sweep mode
select case SweepMode
  case 0  ' CW sweep
    parser.Parse "SENS:IMD:SWE:TYPE CW"
  case 1  ' Power Sweep
    parser.Parse "SENS:IMD:SWE:TYPE POW"
  case 2  ' sweep Fc
    parser.Parse "SENS:IMD:SWE:TYPE FCEN"
  case 3  ' sweep DeltaF
    parser.Parse "SENS:IMD:SWE:TYPE DFR"
  case 4  ' segment sweep
    parser.Parse "SENS:IMD:SWE:TYPE SEGM"
end select

parser.Parse "SENS:IMD:FREQ:FCEN " & CWFreq  ' Frequency Center
sub CalIMD

' Configure IMD GuidedCal for the connector types and ECal module that will be used
' Substitute appropriate connector type and ECal identification strings here

parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 female'"
parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 male'"

' IMD custom settings
' Set the Power Level at the power sensor to be used in calibration
parser.Parse "SENS:CORR:IMD:POW 0"

' Specify the connector type of the power sensor. If there is an adapter between
' the input port and the power sensor, specify the connector type here, and set
' the appropriate cal kit type for the connector so that extra calibration can be
' performed. To skip the calibration for the adapter, set the connection type to "Ignored"
' i.e.: SENS:CORR:IMD:SENS:CONN 'Ignored'
parser.Parse "SENS:CORR:IMD:SENS:CONN 'APC 3.5 female'"  ' set power sensor connector type
parser.Parse "SENS:CORR:IMD:SENS:CKIT 'N4693-60001 User 2 ECal 00012'"  ' set power sensor cal kit type
' Set the Max product to calibrate, valid values are 3, 5, 7, and 9
parser.Parse "SENS:CORR:IMD:MPR 3"

' Set the calibration Frequencies, can choose between calibrate only at center Frequencies (CENT)
' or calibrate at all frequencies (ALL).
parser.Parse "SENS:CORR:IMD:CAL:FREQ ALL"

' Include 2nd order product in calibration
parser.Parse "SENS:CORR:IMD:SORD:INCL 1"

parser.Parse "SENS:CORR:COLL:GUID:INIT"
dim CalSteps, I
for I = 1 to CalSteps
next
parser.Parse "SENS:CORR:COLL:GUID:SAVE"
msgBox "IMD Cal Done"
end sub
Create and Cal an NFX Measurement

This program does the following:

- Setup a Noise Figure SC21 Measurement
- Calibrate Noise Figure channel
- Optional - Configure for an Embedded LO

To run this program, make the following edits, highlighted in yellow:

- Set **hostname** to your PNA computer name
- Set **tunerEcal** and **pullEcal** to your ECal model and info
- Set **ENR** to correct file name and location
- Set **connector types** for ECal, power sensor, and noise source

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as NFX.vbs. **Learn how to setup and run the macro.**

See SCPI commands

- Calc:Custom command
- Noise Figure commands
- Mixer commands
- Embedded LO commands
- Guided Cal commands
- Noise Cal commands

See Other SCPI Example Programs

```vbs
option explicit
dim app
dim hostname
dim parser
hostname = "MyPNA"
set app = CreateObject("Agilentpna835x.application", hostname)
set parser = app.ScpiStringParser
' ecal and noise tunner
```
dim tunerECal

dim sParamECal

tunerECal = "N4691-60003 ECal 00591"

sParamECal = "N4693-60001 User 2 ECal 00012"

' ENR file

dim ENRFile

ENRFile = "C:/Program Files/Agilent/Network Analyzer/Noise/346C_44420601.enr"

call SetupNFX

'optional if not doing embedded LO

call SetupEmbeddedLO

call CalNFX

sub SetupNFX

'Create NF and SC21 traces

parser.Parse "*RST"

parser.Parse "CALC:PAR:DEL:ALL"

parser.Parse "CALC:CUST:DEF 'NF', 'Noise Figure Converters', 'NF' "

parser.Parse "DISP:WIND:TRAC1:FEED 'NF'"

parser.Parse "CALC:CUST:DEF 'SC21', 'Noise Figure Converters', 'SC21' "

parser.Parse "DISP:WIND:TRAC2:FEED 'SC21'"

parser.Parse "SENS:SWE:MODE SING"

' set channel properties

' set sweep type to linear sweep

parser.Parse "SENS:SWE:TYPE LIN"

' set number of points

parser.Parse "SENS:SWE:POIN 101"

' set IF bandwidth

parser.Parse "SENS:BWID 1e3"

' set nfx properties

' turn averaging on

parser.Parse "SENS:NOIS:AVER:STAT ON"

' noise averaging factor

parser.Parse "SENS:NOIS:AVER 40"

' noise tuner ecal module
parsed.Parse "SENS:NOIS:TUN:ID " & tunerECal & " "
' noise tuner input
parsed.Parse "SENS:NOIS:TUN:INP 'B' "
' noise tuner output
parsed.Parse "SENS:NOIS:TUN:OUTP 'A' "
' noise bandwidth
parsed.Parse "SENS:NOIS:BWID 8e6"
' low gain of noise receiver
parsed.Parse "SENS:NOIS:GAIN 0"
' sweep single
parsed.Parse "SENS:SWE:MODE SING"

'set converter properties
' swept input sweep mode
parsed.Parse "SENS:MIX:INP:FREQ:MODE SWEPT"
' fixed lo sweep mode
parsed.Parse "SENS:MIX:LO1:FREQ:MODE FIXED"
' swept output sweep mode
parsed.Parse "SENS:MIX:OUTP:FREQ:MODE SWEPT"
' input start freq
parsed.Parse "SENS:MIX:INP:FREQ:STAR 8e8"
' input stop freq
parsed.Parse "SENS:MIX:INP:FREQ:STOP 3e8"
' lo1 freq
parsed.Parse "SENS:MIX:LO1:FREQ:FIX 1.5825e10"
' lo1 power
parsed.Parse "SENS:MIX:LO1:POW -10"
' calculate output freq
parsed.Parse "SENS:MIX:CALC OUTP"
' sweep single
parsed.Parse "SENS:SWE:MODE SING"
end sub

sub CalNFX
' dut connector
parsed.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 female'"
parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 female''
' port calkits

' power sensor connector
parser.Parse "SENS:CORR:COLL:GUID:PSEN1:CONN 'APC 3.5 female''
' power sensor adapter cal kit
' power calibration power level

' noise source connector
parser.Parse "SENS:NOIS:SOUR:CONN 'APC 3.5 male''
' noise source adapter cal kit
parser.Parse "SENS:NOIS:SOUR:CKIT '' & sParamECal & ''"
' cal method
parser.Parse "SENS:NOIS:CAL:METH 'Vector''
' ENR file
parser.Parse "SENS:NOIS:ENR:FIL '' & ENRFile & ''"
' disable LO power cal
parser.Parse "SENS:CORR:COLL:NOIS:LO:PCAL 0"
' set force both adapter cals de-embed to false
parser.Parse "SENS:CORR:COLL:NOIS:ENR:ADAP:DEEM 0"
parser.Parse "SENS:CORR:COLL:NOIS:PSEN:ADAP:DEEM 0"
' initialize guided cal
parser.Parse "SENS:CORR:COLL:GUID:INIT"

' step through calsteps
dim steps
steps = parser.Parse("SENS:CORR:COLL:GUID:STEP?"")
dim i, str
for i = 1 to steps
msgbox str
parser.Parse "SENS:CORR:COLL:GUID:ACQ STAN" & i
next

parser.Parse "SENS:CORR:COLL:GUID:SAVE 0"
parser.Parse "SENS:SWE:MODE CONT"
end sub

sub SetupEmbeddedLO
' embedded LO properties
' normalize point
parser.Parse "SENS:MIX:ELO:NORM:POIN 101"
' set tuning mode to broadband and precise
parser.Parse "SENS:MIX:ELO:TUN:MODE BRO"
' tuning ifbw
parser.Parse "SENS:MIX:ELO:TUN:IFBW 3e4"
' max tuning iterations
parser.Parse "SENS:MIX:ELO:TUN:ITER 5"
' tuning tolerance
parser.Parse "SENS:MIX:ELO:TUN:TOL 1"
' tuning interval
parser.Parse "SENS:MIX:ELO:TUN:INT 1"
' turn on ELO
parser.Parse "SENS:MIX:ELO:STAT 1"
' sweep single
parser.Parse "SENS:SWE:MODE SING"
end sub
Create and Cal an SMC Measurement

This VB Script example creates and calibrates a scalar mixer measurement.

To run this example without modification you need the following:

- An ECal module that covers the frequency range of the measurement.
- A power meter must be available to the PNA. This can be accomplished either by attaching the meter to the PNA via a GPIB cable, or by using SCPI over LAN.

By removing the comments ( ` ) at the start of the BLUE code, it can also do the following:

- Load a mixer setup file
- Use ECal characterizations
- Specify Mechanical Cal Kits
- Perform manual ECal orientation.
- Specify the thru measurement method.
- Omit the isolation part of the 2-port cal.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example. However, some modification is necessary to make the program run on a traditional GPIB Interface. For example, during the power meter portion of this calibration, scpi.Parse will not process a command until the power meter routine has completed. Traditional GPIB would require a serial polling technique to ensure the routine has completed before proceeding.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as SMC.vbs. Learn how to setup and run the macro.

```vb
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' Create a Scalar Mixer Forward Measurement
' First, delete all measurements on the channel
scpi.Parse "CALC:PAR:DEL:ALL"

' Create a forward scalar mixer measurement and configure it in channel 1. The first parameter is a unique identifying string (specified by the user) to allow subsequent commands to be directed at this specific measurement.
```
scpi.Parse "CALC:CUST:DEF 'My SC21', 'Scalar Mixer/Converter', 'SC21'"

'Setup the new measurement as the 2nd trace in the active window

scpi.Parse "DISP:WIND:TRAC2:FEED 'My SC21'"

'Make the new trace the active measurement

scpi.Parse "CALC:PAR:SEL 'My SC21'"

'The parameters of the mixer measurement can now be configured.
'This can be done by either using the SENS:MIX commands
'for each of the parameters or by loading a mixer setup file.
'Uncomment the following line to load a mixer setup file. The path name
'for the mixer file may be loaded from other mapped drives.
'scpi.Parse "SENS:MIXer:Load 'C:/Program Files/Agilent/Network Analyzer/Documents/Mixer/MyMixer.mxr'"

' Setup Stimulus
' Points and IFBW are channel settings
scpi.Parse "SENS:SWEep:POINts 21"
s CPI.Parse "SENS:BANDwidth 1e3"

'The rest are mixer settings
scpi.Parse "SENS:MIX:INPut:FREQ:MODE SWEp"t"
s CPI.Parse "SENS:MIX:INPut:FREQ:STAR 3.6e9"
s CPI.Parse "SENS:MIX:INPut:FREQ:STOP 3.9e9"
s CPI.Parse "SENS:MIX:LO:FREQ:MODE FIXED"
s CPI.Parse "SENS:MIX:LO:FREQ:FIX 1e9"
s CPI.Parse "SENS:MIX:LO:POW 10"
s CPI.Parse "SENS:MIX:OUTP:FREQ:SID LOW"
s CPI.Parse "SENS:MIX:CALC Output"
s CPI.Parse "SENS:MIX:LO:NAME 'Port 3'"
s CPI.Parse "SENS:MIX:APPLY"

'---------------------------Perform A Scalar Mixer Calibration---------------------------

'Specify the connector types and the cal kits for each of the ports.
s CPI.Parse "SENS:CORR:COLL:GUID:CONN:PORT1:SEL "APC 3.5 male""
s CPI.Parse "SENS:CORR:COLL:GUID:CONN:PORT2:SEL "APC 3.5 female""
Non-factory characterizations are specified as follows:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 User 1 ECal"
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'
' Specify Mechanical cal kits
'scpi.Parse "sens:corr:coll:guid:ckit:port1 '85033D/E"
'scpi.Parse "sens:corr:coll:guid:ckit:port1 '85033D/E"
'
'Optional settings
'Specify the thru measurement method.
'scpi.Parse "SENS:CORR:COLL:GUID:PATH:TMET 1,2,""UNDEFINED THRU"
'Omit the isolation part of the 2-port cal (default behavior).
'scpi.Parse "SENS:CORR:COLL:GUID:ISOL NONE"
' Uncomment the following lines to manually orient
' the ecal port A connected to PNA port 1
'scpi.Parse "SENS:CORR:PREF:ECAL:ORI OFF"
'
'Initialize an SMC guided calibration.
scpi.Parse "SENS:CORR:COLL:GUID:INIT"
'Tell the wizard to generate and report the number of steps in this cal.
Dim steps
Dim desc
'Determine the number of steps required to complete the calibration.
steps = scpi.Parse ("SENS:CORR:COLL:GUID:STEP?")
For i = 1 To steps
'Display the prompt for each step
MsgBox (desc)
'Perform the measurement for each step
scpi.Parse "SENS:CORR:COLL:GUID:ACQ STAN" & CStr(i)
'Finish the cal and save the calset

scpi.Parse("SENS:CORR:COLL:GUID:SAVE ON")

Msgbox("SMC cal saved to CH1_CALREG")

Last Modified:

23-May-2011  Added mixer setup commands
5-Apr-2011    Edited ECal commands
8-Mar-2011    Updated with new SMC commands
4-Jun-2009    Added 'Done' command
Calibrate Multiple SMC Channels

This example allows you to calibrate multiple SMC channels while connecting the power meter and required standards or ECaI module only once.

In the example program:

- Modify `chans = 2` to indicate the number of channels to calibrate.
- You can also change the connector type and cal kit for each port.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as MultChanCal.vbs. Learn how to setup and run the macro.

```vbscript
Dim app
Dim scpi
Dim chans
Dim i
Dim steps
Dim desc

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
app.Preset

' Set number of channels to create
chans = 2
ReDim calset(chans - 1)

For i = 1 To chans
    chanStr = CStr(i+1)  ' calibrate on channels 2 and 3
    Dim parm, measName, sens, calc
    parm = "S" & CStr(i) & CStr(i)
    measName = "My" & parm
    sens = "SENS" & chanStr
    calc = "CALC" & chanStr
    scpi.Parse calc & ":CUST:DEF '" & measName & '"', 'Scalar Mixer/Converter', '" & parm & '"'

    'Setup the new measurement as the 2nd trace in the active window
```
scpi.Parse "DISP:WIND:TRAC" & chanStr & ":FEED '" & measName & "'
'Make the new trace the active measurement
scpi.Parse calc & ":PAR:SEL '" & measName & "'

'--------------Perform A FCA Mixer Calibration--------------
'Set ports and cal kits for 2 port calibration portion
scpi.Parse sens & ":CORR:COLL:GUID:CONN:PORT1:SEL "APC 3.5 male"
scpi.Parse sens & ":CORR:COLL:GUID:CONN:PORT2:SEL "APC 3.5 female"
'ECal modules are specified with the same command

'Specify the thru measurement method.
scpi.Parse sens & ":CORR:COLL:GUID:PATH:TMET 1,2,"DEFINED THRU"

'Omit the isolation part of the 2-port cal
scpi.Parse sens & ":CORR:COLL:GUID:ISOL NONE"

'Initialize an SMC guided calibration.
scpi.Parse sens & ":CORR:COLL:GUID:INIT"
'Determine the number of steps required to complete the calibration.
steps = scpi.Parse (sens & ":CORR:COLL:GUID:STEP?"
Next
For j = 1 To CInt(steps)
'Display the prompt for each step
MsgBox (desc)
'Measure the same standard for each channel
For i = 1 To chans
    chanStr = CStr(i+1) ' channel number as string
    scpi.Parse "SENS" & chanStr & ":CORR:COLL:GUID:ACQ STAN" & CStr(j)
    opc_comp = scpi.Parse("*OPC?"
Next
Next
'Finish the cal and save the calsets
For i = 1 To chans
    calset(i - 1) = scpi.Parse("SENS" & CStr(i+1) & ":CORR:COLL:GUID:SAVE ON")
Next
MsgBox ("SMC Cals Complete!")
Create New Cal Kit using SCPI

When creating new cal kits programmatically, the order in which cal kit commands are sent can be important. For example to create a kit with opens, shorts, loads, and thurs. Be sure to use the following sequence for each newly defined standard.

1. Programmmatically select the standard number
2. Programatically select the standard type.
3. Program the cal standard's values.
4. Repeat steps 1, 2, 3 for additional new standards being defined.

10  !
20  !
30  ! This example program demonstrates how to create
40  ! new PNA calibration kits.
50  !
60  ! 1) Select a kit not previously defined
70  ! 2) Define open, short, load, and thru cal standards
80  !   Note: Each of the newly defined standards is assigned
90  !   a default connector name. These default connector names
100 ! will be replaced in subsequent steps.
110 ! 3) Use the delete connector command to remove default
120 !   connector names.
130 ! 4) Add connectors. Specify:
140 !   Start and Stop Freq
150 !   Z - Impedance
160 !   sex - MALE, FEMALE, NONE
170 !   media - COAX, WAVE
180 !   cutoff - Frequency for waveguide
190 ! 5) Assign the appropriate connector to each standard
200 ! 6) Modify the class assignments for the standards defined
210 ! 7) Verify the kit values
220 !
230 ! Additional Note: After setting each new cal kit value, it is
240 ! recommended that the program periodically perform queries to
250 ! verify the new values.
260 !
270 ! This will prevent program synchronization issues that can affect
280 ! final values stored within new cal kits.
290 !
300 !------------------------------------------------------------
310 !
320 ! Set up I/O path
330 ASSIGN @Na TO 716
340 DIM Calkname$[80],Conn$[80]
350 INTEGER Calkitnum
CLEAR SCREEN

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

Designate the kit selection to be used for performing cal's

OUTPUT @Na;"sens:corr:ckit:count?"
ENTER @Na;Calkitnum
Calkitnum=Calkitnum+1
OUTPUT @Na;"sens:corr:coll:ckit "&VAL$(Calkitnum)

Name this kit with your own name
OUTPUT @Na;"sens:corr:coll:ckit:name "Special 2.4 mm Model 85056"

DISP "Defining kit std 1..."
 OUTPUT @Na;"sens:corr:coll:ckit:stan 1"
 OUTPUT @Na;"sens:corr:coll:ckit:stan:type SHORT"
Get_std
OUTPUT @Na;"sens:corr:coll:ckit:stan:char coax"
OUTPUT @Na;"sens:corr:coll:ckit:stan:label "My Short"
Get_label

DISP "Defining kit std 2..."
 OUTPUT @Na;"sens:corr:coll:ckit:stan 2"
 OUTPUT @Na;"sens:corr:coll:ckit:stan:type OPEN"
Get_std
OUTPUT @Na;"sens:corr:coll:ckit:stan:char coax"
OUTPUT @Na;"sens:corr:coll:ckit:stan:label "My Open"
Get_label

DISP "Defining kit std 3..."
 OUTPUT @Na;"sens:corr:coll:ckit:stan 3"
 OUTPUT @Na;"sens:corr:coll:ckit:stan:type LOAD"
Get_std
OUTPUT @Na;"sens:corr:coll:ckit:stan:char coax"
OUTPUT @Na;"sens:corr:coll:ckit:stan:label "My Fixed Load"
Get_label

DISP "Defining kit std 4..."
 OUTPUT @Na;"sens:corr:coll:ckit:stan 4"
 OUTPUT @Na;"sens:corr:coll:ckit:stan:type THRU"
Get_std
OUTPUT @Na;"sens:corr:coll:ckit:stan:char coax"
OUTPUT @Na;"sens:corr:coll:ckit:stan:label "My Thru"
Get_label
850 !
860 DISP "Defining kit std 5..."
870 ! Now set up standard #5
880 OUTPUT @Na:"sens:corr:coll:ckit:stan 5"
890 OUTPUT @Na:"sens:corr:coll:ckit:stan:type SLOAD"
900 Get_std
910 OUTPUT @Na:"sens:corr:coll:ckit:stan:char coax"
920 OUTPUT @Na:"sens:corr:coll:ckit:stan:label ""Sliding Load"
930 Get_label
940 !
950 DISP "Defining kit std 6..."
960 ! Now set up standard #6
970 !
980 OUTPUT @Na:"sens:corr:coll:ckit:stan 6"
990 OUTPUT @Na:"sens:corr:coll:ckit:stan:type SHORT"
1000 Get_std
1010 OUTPUT @Na:"sens:corr:coll:ckit:stan:char coax"
1020 OUTPUT @Na:"sens:corr:coll:ckit:stan:label ""Short"
1030 Get_label
1040 !
1050 DISP "Defining kit std 7..."
1060 ! Now set up standard #7
1070 OUTPUT @Na:"sens:corr:coll:ckit:stan 7"
1080 OUTPUT @Na:"sens:corr:coll:ckit:stan:type SHORT"
1090 Get_std
1100 OUTPUT @Na:"sens:corr:coll:ckit:stan:char coax"
1110 OUTPUT @Na:"sens:corr:coll:ckit:stan:label ""Short"
1120 Get_label
1130 !
1140 DISP "Defining kit std 8..."
1150 ! Now set up standard #8
1160 !
1170 OUTPUT @Na:"sens:corr:coll:ckit:stan 8"
1180 OUTPUT @Na:"sens:corr:coll:ckit:stan:type ARBI"
1190 Get_std
1200 OUTPUT @Na:"sens:corr:coll:ckit:stan:char coax"
1210 OUTPUT @Na:"sens:corr:coll:ckit:stan:TZR 15"
1220 OUTPUT @Na:"sens:corr:coll:ckit:stan:TZI -9"
1230 OUTPUT @Na:"sens:corr:coll:ckit:stan:label ""Z Load"
1240 Get_label
1250 !
1260 !
1270 !
1280 !
1290 ! First remove any old connector names
1300 OUTPUT @Na:"sens:corr:coll:ckit:conn:del"
1310 ! Verify that no connectors are currently installed
1320 OUTPUT @Na:"sens:corr:coll:ckit:conn:cat?"
1330 ENTER @Na;Conn$
1340 PRINT "Verify empty list: ";Conn$
Define your new connectors

OUTPUT @Na ;":sens:corr:coll:kit:conn:add ""PSC 2.4",0HZ,999GHZ,50.0,MALE,COAX,0.0"

OUTPUT @Na ;":sens:corr:coll:kit:conn:add ""PSC 2.4",0HZ,999GHZ,50.0,FEMALE,COAX,0.0"

Verify that the new connectors are installed

OUTPUT @Na ;":sens:corr:coll:kit:conn:cat?"
ENTER @Na ;Conn$
PRINT "Verify new connectors: ";Conn$
DISP ""

Disp "Defining conn std 1..."

OUTPUT @Na ;":sens:corr:coll:kit:stan 1"
Verify_std
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",FEMALE,1"
Print_connector

Disp "Defining conn std 2..."

OUTPUT @Na ;":sens:corr:coll:kit:stan 2"
Verify_std
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",FEMALE,1"
Print_connector

Disp "Defining conn std 3..."

OUTPUT @Na ;":sens:corr:coll:kit:stan 3"
Verify_std
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",FEMALE,1"
Print_connector

Disp "Defining conn std 4..."

OUTPUT @Na ;":sens:corr:coll:kit:stan 4"
Verify_std
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",FEMALE,1"
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",MALE,2"
Print_connector

Disp "Defining conn std 5..."

OUTPUT @Na ;":sens:corr:coll:kit:stan 5"
OUTPUT @Na ;":sens:corr:coll:kit:stan:label ""Sliding Load"
Verify_std
OUTPUT @Na ;":sens:corr:coll:kit:conn:snam ""PSC 2.4"",MALE,1"
Print_connector
1820 !
1830 DISP "Defining conn std 6..."
1840 ! Now set up standard #6
1850 !
1860 OUTPUT @Na;":sens:corr:coll:ckit:stan 6"
1870 Verify_std
1880 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC 2.4"",MALE,1"
1890 Print_connector
1900 !
1910 DISP "Defining conn std 7..."
1920 ! Now set up standard #7
1930 OUTPUT @Na;":sens:corr:coll:ckit:stan 7"
1940 Verify_std
1950 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC 2.4"",MALE,1"
1960 Print_connector
1970 !
1980 DISP "Defining conn std 8..."
1990 ! Now set up standard #8
2000 OUTPUT @Na;":sens:corr:coll:ckit:stan 8"
2010 Verify_std
2020 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC 2.4"",MALE,1"
2030 Print_connector
2040 !
2050 DISP "Class assignments..."
2060 !
2070 ! Designate the "order" associated with measuring the standards
2080 !
2090 ! Set Port 1, 1st standard measured to be standard #2
2100 OUTPUT @Na;":sens:corr:coll:ckit:order1 2"
2110 ! Set Port 1, 2nd standard measured to be standard #1
2120 OUTPUT @Na;":sens:corr:coll:ckit:order2 1,6,7"
2130 ! Set Port 1, 3nd standard measured to be standard #3 and #5
2140 OUTPUT @Na;":sens:corr:coll:ckit:order3 3,5"
2150 ! Set Port 1, 4th standard measured to be standard #4
2160 OUTPUT @Na;":sens:corr:coll:ckit:order4 4"
2170 !
2180 ! Set Port 2, 1st standard measured to be standard #2
2190 OUTPUT @Na;":sens:corr:coll:ckit:order5 2"
2200 ! Set Port 2, 2nd standard measured to be standard #1
2210 OUTPUT @Na;":sens:corr:coll:ckit:order6 1,6,7"
2220 ! Set Port 2, 3nd standard measured to be standard #3 and #6
2230 OUTPUT @Na;":sens:corr:coll:ckit:order7 3,5"
2240 ! Set Port 2, 4th standard measured to be standard #4
2250 OUTPUT @Na;":sens:corr:coll:ckit:order8 4"
2260 !
2270 ! Set Port 1, 1st standard
2280 OUTPUT @Na;":sens:corr:coll:ckit:olabel1 ""MyOpen1"
2290 ! Set Port 1, 2nd standard
2300 OUTPUT @Na;":sens:corr:coll:ckit:olabel2 ""MyShorts1""
2310 ! Set Port 1, 3rd standard
2320 OUTPUT @Na;":sens:corr:coll:ckit:olabel3 "MyLoads1"
2330 ! Set Port 1, 4th standard measured to be standard #4
2340 OUTPUT @Na;":sens:corr:coll:ckit:olabel4 "MyThru1"
2350 !
2360 ! Set Port 2, 1st standard
2370 OUTPUT @Na;":sens:corr:coll:ckit:olabel5 "MyOpen2"
2380 ! Set Port 2, 2nd standard
2390 OUTPUT @Na;":sens:corr:coll:ckit:olabel6 "MyShorts2"
2400 ! Set Port 2, 3rd standard
2410 OUTPUT @Na;":sens:corr:coll:ckit:olabel7 "MyLoads2"
2420 ! Set Port 2, 4th standard
2430 OUTPUT @Na;":sens:corr:coll:ckit:olabel8 "MyThrus2"
2440 !
2450 BEEP
2460 DISP "Done!"
2470 END
2480 SUB Get_label
2490 OUTPUT 716;":sens:corr:coll:ckit:stan:label?"
2500 ENTER 716;Label$
2510 PRINT Label$
2520 SUBEND
2530 !
2540 SUB Get_std
2550 OUTPUT 716;":sens:corr:coll:ckit:stan:type?"
2560 ENTER 716;Type$
2570 PRINT Type$
2580 SUBEND
2590 !
2600 SUB Print_connector
2610 DIM Nam$[40]
2620 OUTPUT 716;":sens:corr:coll:ckit:conn:sname?"
2630 ENTER 716;Nam$
2640 PRINT Nam$
2650 SUBEND
2660 !
2670 SUB Verify_std
2680 OUTPUT 716;":sens:corr:coll:ckit:stan:label?"
2690 ENTER 716;Label$
2700 SUBEND
2710 !
Create a Custom Power Meter Driver

This topic requires that you have a working knowledge of Visual Basic.

This topic will help you create your own power meter driver for use with Source Power Calibration on the PNA. If you are using an Agilent Power Meter to perform a Source Power Calibration, you do NOT need to create your own driver.

Your Power Meter driver will be created from a template written in Visual Basic using VISA over the GPIB bus.

*Note:* This procedure applies to Visual Basic 6.0. Applicability to Visual Basic .NET has not yet been investigated.

- **Prepare Template Files**
- **Modify Template Files**
- **Compile, Copy, and Register, Your New Driver**
- **Test Your new Driver**

---

**Other SCPI Example Programs**

**Prepare Template Files**

1. Copy all the files from the PNA hard drive C:/Program Files/Agilent/Network Analyzer/Automation/Power Meter Driver Template folder, to a folder on your development PC.

2. In Visual Basic click **File**, then **Open Project…**, find **MyPowerMeter.vbp** (a file you copied from the PNA). Click **Open**. This is a VB ActiveX EXE template, which you will fill in to become your driver.

3. Click **Project**, then **MyPowerMeter Properties**. Click the **General** tab.

4. Overwrite the Project Name with a name of your own choosing. This will be the name of your driver’s type library (also the default name of your exe).

*Note* If the name of your exe does not match the VB Project Name with which it was compiled, registration of the exe on the PNA will not succeed.

5. Set the Project Description. After building your driver if you wish to test it using VB, this is the string that will show up in the VB References list of your test project, and also in the lower pane of the VB Object Browser.

6. Set the Thread Pool size to 1 thread.

7. Click **OK** to close the project properties dialog.

8. From the VB **Project** menu, click **References…** Ensure that **Agilent PNA Power Meter 1.0 Type Library** and **VISA Library** are checked. Click **OK**.

*Note:* Agilent's implementation of VISA is installed as part of the Agilent I/O Libraries on the PNA. For help on
VISA, go to the Windows Start button on your PNA, select Programs, Agilent IO Libraries, VISA Help.

Modify Template Files
From Visual Basic View menu click Project Explorer. Expand the Modules and Class Modules folders. Ensure there is one module (WinAPI) and one class module (PowerMeter).

Let's look at the WinAPI module first.

1. In the Project Explorer window, click WinAPI.
2. From the View menu click Code.

There is only one line of code you should need to modify in this module: the value of the string constant named sIDSEARCH. The comments preceding the declaration of that string describe how to change it. The rest of this module contains functions which will use the Microsoft Windows API to insure proper registration of your driver on the PNA. If you know of other Windows API functions you feel might be helpful to call from within your PowerMeter class module (to help in formatting data, for example), this module would be the place to declare them.

Now let's look at the class module.

1. In the Project Explorer window, click PowerMeter.
2. From the View menu click Properties Window. The Instancing property must be set to MultiUse. This allows other applications to create objects from this class, such that one instance of your driver EXE can supply more than one such object at a time.
3. From the View menu click Code.

Do NOT modify the Interfaces to IPowerMeter subroutines and functions. PNA source power cal expects to find these interfaces as they are currently defined.

The only members that you need to supply code to are those containing “Your code here” comments.

In addition, comments have been provided at the beginning of each member to describe the information that member needs to be read from or written to the power meter.

To get an idea of how communicate with the power meter using the VISA functions viWrite and viRead, examine the code which has been implemented for you in IPowerMeter_Connect, IPowerMeter_QueryMeter, and IPowerMeter_WriteMeter.

Compile, Copy, and Register Your New Driver
When your driver is ready to run, you will first need to compile it into an EXE.
From the File menu select Make exe.
After compiling, the following will instruct VB to use the same ID (GUID) every time you re-compile your project.

1. From the Project menu, click PowerMeter Properties.
2. On the Component tab, select Binary Compatibility and click ...
3. Browse to and select your project EXE. Click Open.
4. Click OK to close Project Properties.

5. Save your project.

6. Copy your driver EXE file to a folder on your PNA (do NOT use C:/Program Files/Agilent/Network Analyzer/Automation/Power Meter Driver Template folder).

7. Run the EXE file. A message box will pop up reporting whether or not registration was successful. If not successful, it will make a suggestion on what to fix.

When your driver is properly registered, PNA Source Power Cal should be able to associate it with the ID string of your power meter.

---

**Test Your Power Meter Driver**

We have also provided a Visual Basic project to test your new Power Meter driver. This project individually calls every IPowerMeter method and property in your driver to verify that it performs correctly. Before running the test your PC and PNA must be configured to communicate using DCOM.

1. Connect your PC and the PNA to LAN.

2. Add your PC logon to the PNA. Both logons and password must match to communicate using DCOM. See [Additional PNA users](#).

3. Configure your driver using DCOM Config on the PNA. This will give you permission to launch and access the driver. See [Configure for COM-DCOM Programming](#).

**Modify the Test Project**

1. In Visual Basic click File, then Open Project..., find MyPowerMeterTest.vbp (a file you copied from the PNA). Click Open.

2. From the Project menu, click References... From the list, find and check your new Power Meter Driver. (It should have been registered on your PC when you successfully made your driver EXE.) Click OK.

3. From the View menu click Code.

4. Modify the CreateObject line as follows:
   - Replace MyPowerMeter with the Project Name that you chose for your driver
   - Replace MyPNA with the Computer Name of your PNA.
   - For example:

   ```vba
   Set PowerMeterObj = CreateObject("AcmeBrand.PowerMeter", "AGILENT-PNA123")
   ```
   (This assumes that you kept PowerMeter as class module name in your driver.)

**Run the Test Project**

Ensure your power meter is connected to the PNA with a GPIB cable.

Put the PNA in system controller mode:
1. From the PNA **System** menu point to **Configure** then click **SICL/GPIB**.

2. In the GPIB box click **System Controller**.

Run the test project. If there are no errors, the driver is created successfully. If there are errors, try to figure out what went wrong and fix it. Then re-compile, re-copy the .exe to the PNA, and re-run the test. You should not need to re-register the driver or re-modify the test program.
ECAL Confidence Check using SCPI

This Visual Basic program performs a complete ECAL confidence check.

To run this program, you need:

- An established GPIB interface connection
- Agilent's VISA or National Instrument's VISA installed on your PC
- The module visa32.bas added to your VB project.
- A form with two buttons: cmdRun and cmdQuit
- A calibrated S11 1-port or N-port measurement active on Channel 1
- Window 1 is visible

**Note:** A confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

### See Other SCPI Example Programs

```vba
' Session to VISA Default Resource Manager
Private defRM As Long
' Session to PNA
Private viPNA As Long
' VISA function status return code
Private status As Long

Private Sub Form_Load()
    defRM = 0
End Sub

Private Sub cmdRun_Click()
    ' String to receive data from the PNA
    Dim strReply As String * 200

    ' Open the VISA default resource manager
    status = viOpenDefaultRM(defRM)
    If (status < VI_SUCCESS) Then HandleVISAError

    ' Open a VISA session (viPNA) to the PNA at GPIB address 16.
    status = viOpen(defRM, "GPIB0::16::INSTR", 0, 0, viPNA)
    If (status < VI_SUCCESS) Then HandleVISAError

    ' Need to set the VISA timeout value to give all our GPIB Reads
    ' sufficient time to complete before a timeout error occurs.
    ' For this example, let's try setting the limit to
    ' 10000 milliseconds (10 seconds).
```
status = viSetAttribute(viPNA, VI_ATTR_TMO_VALUE, 10000)
If (status < VI_SUCCESS) Then HandleVISAError

' Get the catalog of all the measurements currently on Channel 1.
status = myGPIBWrite(viPNA, "CALC1:PAR:CAT?")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError

' If an S11 measurement named "MY_S11" doesn't already exist,
' then create it.
If InStr(strReply, "MY_S11") = 0 Then
  status = myGPIBWrite(viPNA, "CALC1:PAR:DEF:EXT MY_S11,S11")
  If (status < VI_SUCCESS) Then HandleVISAError
End If
strReply = ""

' Get the catalog of all the trace numbers currently active
' in Window 1.
status = myGPIBWrite(viPNA, "DISP:WIND1:CAT?")
If (status < VI_SUCCESS) Then HandleVISAError

status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError

' If a trace number 4 already exists in Window 1, then this
' will remove it.
If InStr(strReply, "4") > 0 Then
  status = myGPIBWrite(viPNA, "DISP:WIND1:TRAC4:DEL")
  If (status < VI_SUCCESS) Then HandleVISAError
End If

' Set trace number 4 to MY_S11.
status = myGPIBWrite(viPNA, "DISP:WIND1:TRAC4:FEED MY_S11")
If (status < VI_SUCCESS) Then HandleVISAError

' Set up trace view so we are viewing only the data trace.
status = myGPIBWrite(viPNA, "DISP:WIND1:TRAC4 ON")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBWrite(viPNA, "DISP:WIND1:TRAC4:MEM OFF")
If (status < VI_SUCCESS) Then HandleVISAError

' Select MY_S11 as the measurement to be used for the
' Confidence Check.
status = myGPIBWrite(viPNA, "SENS1:CORR:CCH:PAR MY_S11")
If (status < VI_SUCCESS) Then HandleVISAError

' Acquire the S11 confidence check data from ECal Module A
' into the memory buffer (asking for an opc reply when it's done).
status = myGPIBWrite(viPNA, "SENS1:CORR:CCH:ACQ ECAL1;*OPC?")
If (status < VI_SUCCESS) Then HandleVISAError
The PNA sends an OPC reply ("+1") when the confidence data acquisition into memory is complete, so this Read is waiting on the reply until it is received.

status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError

Turn on trace math so the trace shows data divided by memory.

You can be confident the S11 calibration is reasonably good if the displayed trace varies no more than a few tenths of a dB from 0 dB across the entire span.

status = myGPIBWrite(viPNA, "CALC1:PAR:SEL MY_S11")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBWrite(viPNA, "CALC1:MATH:FUNC DIV")
If (status < VI_SUCCESS) Then HandleVISAError
End Sub

Private Sub cmdQuit_Click()
' Turn off trace math
status = myGPIBWrite(viPNA, "CALC1:MATH:FUNC NORM")
If (status < VI_SUCCESS) Then HandleVISAError

' Conclude the confidence check to set the ECAL module back to it's idle state.
status = myGPIBWrite(viPNA, "SENS1:CORR:CCH:DONE")
If (status < VI_SUCCESS) Then HandleVISAError

' Close the resource manager session (which also closes the session to the PNA).
If defRM <> 0 Then Call viClose(defRM)

' End the program
End
End Sub

Private Function myGPIBWrite(ByVal viHandle As Long, ByVal strOut As String) As Long
' The "+ Chr$(10)" appends an ASCII linefeed character to the output, for terminating the write transaction.
myGPIBWrite = viVPrintf(viHandle, strOut + Chr$(10), 0)
End Function

Private Function myGPIBRead(ByVal viHandle As Long, strIn As String) As Long
myGPIBRead = viVScanf(viHandle, "+\t", strIn)
End Function

Sub HandleVISAError()
Dim strVisaErr As String * 200
Call viStatusDesc(defRM, status, strVisaErr)
MsgBox "*** Error : "+ strVisaErr, vbExclamation
End
End Sub
Establish a VISA Session

This Visual Basic program demonstrates how to send a SCPI command using VISA and the Agilent IO libraries. To run this program, you need:

- Your PC and PNA both connected to a LAN (for communicating with each other).
- The SICL and VISA components of Agilent’s I/O Libraries software installed on your PC. Both are included when you install the software, unless you already have another vendor’s VISA installed. Then specify Full SICL and VISA installation to overwrite the other vendor’s VISA.
- The module visa32.bas added to your VB project. After you install VISA, the module will be located at C:/VXI/PNP/WinNT (or equivalent)/INCLUDE/Visa32.bas
- A form with two buttons: cmdRun and cmdQuit.
- Your PC configured to be a VISA LAN Client, and the SICL Server capability enabled on the PNA. See Configure for VISA and SICL

See Other SCPI Example Programs

Note: This example is a piece of a larger VISA program that performs a source power calibration.

```
' Session to VISA Default Resource Manager
Private defRM As Long
' Session to PNA
Private viPNA As Long
' VISA function status return code
Private status As Long

Private Sub Form_Load()
  defRM = 0
End Sub

Private Sub cmdRun_Click()
  ' String to receive data from the PNA.
  ' Dimensioned large enough to receive scalar comma-delimited values
  ' for 21 frequency points (20 ASCII characters per point)
  Dim strReply As String * 420
  
  ' Open the VISA default resource manager
  status = viOpenDefaultRM(defRM)
  If (status < VI_SUCCESS) Then HandleVISAError
  
  ' Open a VISA session (viPNA) to the SICL LAN server
  ' at “address 16” on the PNA pointed to by the “GPIB0”
  ' VISA LAN Client on this PC.
  ' CHANGE GPIB0 TO WHATEVER YOU PNA IS SET TO
  status = viOpen(defRM, "GPIB0::16::INSTR", 0, 0, viPNA)
  If (status < VI_SUCCESS) Then HandleVISAError
  
  ' Need to set the VISA timeout value to give all our calls to
```
'myGPIBRead sufficient time to complete before a timeout error occurs.
For this example, let's try setting the limit to 30000 milliseconds (30 seconds).
status = viSetAttribute(viPNA, VI_ATTR_TMO_VALUE, 30000)
If (status < VI_SUCCESS) Then HandleVISAError

Preset the PNA
status = myGPIBWrite(viPNA, "SYST:PRES")
If (status < VI_SUCCESS) Then HandleVISAError

Print the data using a message box
MsgBox strReply
End Sub

Private Sub cmdQuit_Click()
' Close the resource manager session (which also closes the session to the PNA).
If defRM <> 0 Then Call viClose(defRM)

' End the program
End
End Sub

Private Function myGPIBWrite(ByVal viHandle As Long, ByVal strOut As String) As Long
' The "+ Chr$(10)" appends an ASCII linefeed character to the output, for terminating the write transaction.
myGPIBWrite = viVPrintf(viHandle, strOut + Chr$(10), 0)
End Function

Private Function myGPIBRead(ByVal viHandle As Long, strIn As String) As Long
myGPIBRead = viVScanf(viHandle, "%t", strIn)
End Function

Sub HandleVISAError()
Dim strVisaErr As String * 200
Call viStatusDesc(defRM, status, strVisaErr)
MsgBox "*** Error : " + strVisaErr, vbExclamation
End
End Sub
This program demonstrates the use of several External Test Set Control commands. The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do not need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as ExtTS.vbs. Learn how to setup and run the macro.

```vbs
' Demonstrate some SCPI commands for external testsets.
Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Set scpi = pna.ScpiStringParser
' The K64 testset is only usable on a 4-port PNA
If (pna.NumberOfPorts <> 4) Then
    MsgBox("This program only runs on 4-port analyzers.")
Else
    ' If Help is active, show the measurement window and help
    scpi.Execute("DISP:ARR TILE")
    ' Return the list of supported test sets
    list = scpi.Execute("SENS:MULT:CATalog?")
    MsgBox(list)
    '************* K64 *****************
    ' The K64 is connected using the Testset I/O
    ' connector. There is no handshake information.
    ' Therefore, a testset need not be connected.
    ' Load a configuration file.
    scpi.Execute("SENS:MULT1:TYPE 'Z5623AK64'")
    scpi.Execute("SENS:MULT1:ADDR 0")
    ' return stuff about the test set
    ' Returns number of input ports
    Inports = scpi.Execute("SENS:MULT1:INCount?")
    MsgBox("Input Ports: " & CStr(Inports))
    ' Returns number of output ports
    ports = scpi.Execute("SENS:MULT1:COUNt?")
    MsgBox("Output Ports: " & CStr(ports))
    ' Returns valid output ports for each input port
```

2994
For portNum = 1 To Inports

    ports = scpi.Execute("SENS:MULT1:PORT" & CStr(portNum) & ":CAT?"")
    MsgBox("Port " & CStr(portNum) & " catalog: " & (ports))

Next

'Set different port mapping
scpi.Execute("SENS:MULT1:ALLPorts '1 ext R,2 ext R,3 ext R,4 ext R'"")

'Return port mapping
portMap = scpi.Execute("SENS:MULT1:ALLPorts?"")
MsgBox("Ports will be mapped to " & CStr(portMap))

'Enable external testset control and execute port mapping. This automatically enables status bar display as well.
scpi.Execute("SENS:MULT1:STATe 1")
MsgBox("Z5623A K64 Enabled")

End If
Getting and Putting Data using SCPI

This Visual Basic Program does the following:

- Reads data from the analyzer
- Puts the data back into memory
- To see the data on the analyzer after running the program, from the front panel click:
  Trace - Math/Memory - Memory Trace

To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

Note: To change the read and write location of data, removing the comment from the beginning of ONE of the lines, and replace the comment in the beginning of the SDATA and SMEM lines.

```
Private Sub ReadWrite_Click()
  Dim i As Integer
  Dim t As Integer
  Dim q As Integer
  Dim dat As String
  Dim cmd As String
  Dim datum() As Double

  GPIB.Configure
  GPIB.Write "SYSTem:PRESet;*wai"

  'Select the measurement
  GPIB.Write "CALCulate:PARameter:SELect 'CH1_S11_1'"

  'Read the number of data points
  GPIB.Write "SENSe1:SWEep:POIN?"
  numpts = GPIB.Read

  'Turn continuous sweep off
  GPIB.Write "INITiate:CONTinuous OFF"

  'Take a sweep
  GPIB.Write "INITiate:IMMediate;*wai"

  'Ask for the Data

  'PICK ONE OF THESE LOCATIONS TO READ
  GPIB.Write "CALCulate:DATA? FDATA" 'Formatted Meas
  GPIB.Write "CALCulate:DATA? FMEM" 'Formatted Memory
  GPIB.Write "CALCulate:DATA? SDATA" 'Corrected, Complex Meas
  GPIB.Write "CALCulate:DATA? SMEM" 'Corrected, Complex Memory
  GPIB.Write "CALCulate:DATA? SCORR1" 'Error-Term Directivity

  'Number of values returned per data point
```
'q = 1 ' Pick this if reading FDATA or FMEM
q = 2 ' Otherwise pick this

'Parse the data
ReDim datum(q, numpts)
For i = 0 To numpts - 1
    For t = 0 To q - 1
        'Read the Data
        dat = GPIB.Read(20)
        'Parse it into an array
        datum(t, i) = Val(dat)
    Next t
Next i

'PUT THE DATA BACK IN
GPIB.Write "format ascii"

'PICK ONE OF THESE LOCATIONS TO PUT THE DATA
'cmd = "CALCulate:DATA FDATA," 'Formatted Meas
'cmd = "CALCulate:DATA FMEM," 'Formatted Memory
'cmd = "CALCulate:DATA SDATA," 'Corrected, Complex Meas
cmd = "CALCulate:DATA SMEM," 'Corrected, Complex Memory
'cmd = "CALCulate:DATA SCORR1," 'Error-Term Directivity

For i = 0 To numpts - 1
    For t = 0 To q - 1
        If i = numpts - 1 And t = q - 1 Then
            cmd = cmd & Format(datum(t, i))
        Else
            cmd = cmd & Format(datum(t, i)) & "",
        End If
    Next t
Next i

GPIB.Write cmd
End Sub
** GPIB Pass-Through Example  

The SCPI **SYSTem** commands used in this example allow you to send GPIB commands to another GPIB device through the PNA. The other GPIB device cannot be connected to the GPIB bus through the PNA rear panel if the PNA is being controlled by a remote PC using that connector. The other device would typically be connected to the PNA using a **USB/GPIB interface**.

This VB Script example uses the COM **SCPIStringParser** object. However, this is not critical to the use of these commands; they can be sent using the normal syntax of your programming environment. Using the SCPIStringParser over LAN allows you to communicate with GPIB devices without requiring your remote PC to have a GPIB interface card installed.

Although this method of pass-through works for most applications, there are a couple of limitations:

- All data is transferred using ASCII format. Therefore, transferring large blocks of data is very slow.
- Only read and write functions are possible. Service Interrupts are not supported.

*See Other SCPI Example Programs*

```vbscript
option explicit

dim app
set app = CreateObject("AgilentPNA835x.Application")

dim p
set p = app.ScpiStringParser

' Open a new GPIB session on Bus:2 Device:14 Timeout: 100ms
p.Parse "SYST:COMM:GPIB:RDEV:OPEN 2,14,100"

dim handleAsStr

' Retrieve the handle (ID number)
handleAsStr = p.Parse("SYST:COMM:GPIB:RDEV:OPEN?")

' Convert the handle to an integer
dim handleAsInt
handleAsInt = CInt(handleAsStr)

' Send the "*IDN?" query
p.Parse "SYST:COMM:GPIB:RDEV:WRITE " & handleAsInt & ",*IDN?"

' Read its results
dim idn
msgbox idn

' Close the GPIB session
p.Parse "SYST:COMM:GPIB:RDEV:CLOSE " & handleAsInt
```
See Other SCPI Example Programs

/*
 * This example assumes the user's PC has a National Instruments GPIB board. The example is comprised of three basic parts:
 *
 * 1. Initialization
 * 2. Main Body
 * 3. Cleanup
 *
 * The Initialization portion consists of getting a handle to the PNA and then doing a GPIB clear of the PNA.
 *
 * The Main Body consists of the PNA SCPI example.
 *
 * The last step, Cleanup, releases the PNA for front panel control.
 */

#include <stdio.h>
#include <stdlib.h>

#include <windows.h>
#include "decl-32.h"

#define ERRMSGSIZE 1024 // Maximum size of SCPI command string
#define ARRAYSIZE 1024 // Size of read buffer

#define BDINDEX 0 // Board Index of GPIB board
#define PRIMARY_ADDR_OF_PNA 16 // GPIB address of PNA
#define NO_SECONDARY_ADDR 0 // PNA has no Secondary address
#define TIMEOUT T10s // Timeout value = 10 seconds
#define EOTMODE 1 // Enable the END message
#define EOSMODE 0 // Disable the EOS mode

int pna;
char ValueStr[ARRAYSIZE + 1];

void GPIBWrite(char* SCPIcmd);
char *GPIBRead(void);
void GPIBCleanup(int Dev, char* ErrorMsg);

int main()
{
  char *opc;
  char *result;
  char *value;

  /*
   * =========================================
   * INITIALIZATION SECTION
   * =========================================
   */

  /*
   * The application brings the PNA online using ibdev. A device handle, pna, is
   * returned and is used in all subsequent calls to the PNA.
   */
  pna = ibdev(BDINDEX, PRIMARY_ADDR_OF_PNA, NO_SECONDARY_ADDR,
              TIMEOUT, EOTMODE, EOSMODE);
  if (ibsta & ERR)
    {
    printf("Unable to open handle to PNA/nibsta = 0x%x iberr = %d/n",
           ibsta, iberr);
    return 1;
    }

  /*
   * Do a GPIB Clear of the PNA. If the error bit ERR is set in ibsta, call
   * GPIBCleanup with an error message.
   */
  ibclr (pna);
  if (ibsta & ERR)
    {
    GPIBCleanup(pna, "Unable to perform GPIB clear of the PNA");
    return 1;
    }

  /*
   * =========================================
   * MAIN BODY SECTION
   * =========================================
   */

  // Reset the analyzer to instrument preset
  GPIBWrite("SYSTem:FPRESET");

  // Create S11 measurement
  GPIBWrite("CALCulate1:PARameter:DEFine:EXT 'My_S11',S11");
// Turn on Window #1
GPIBWrite("DISPlay:WINDow1:STATE ON");

// Put a trace (Trace #1) into Window #1 and 'feed' it from the measurement
GPIBWrite("DISPlay:WINDow1:TRACe1:FEED 'My_S11'");

// Setup the channel for single sweep trigger
GPIBWrite("INITiate1:CONTinuous OFF;*OPC?");
opc = GPIBRead();
GPIBWrite("SENSe1:SWEep:TRIGger:POINt OFF");

// Set channel parameters
GPIBWrite("SENSe1:SWEep:POINts 11");
GPIBWrite("SENSe1:FREQuency:STARt 1000000000");
GPIBWrite("SENSe1:FREQuency:STOP 2000000000");

// Send a trigger to initiate a single sweep
GPIBWrite("INITiate1:*OPC?");
opc = GPIBRead();

// Must select the measurement before we can read the data
GPIBWrite("CALCulate1:PARameter:SELect 'My_S11'");

// Read the measurement data into the "result" string variable
GPIBWrite("FORMat ASCII");
GPIBWrite("CALCulate1:DATA? FDATA");
result = GPIBRead();

// Print the data to the display console window
printf("S11(dB) - Visual C++ SCPI Example for PNA/n/n");
value = strtok(result, ",");
while (value != NULL)
{
    printf("%s/n", value);
    value = strtok(NULL, ",");
}

/*
* =-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=
* CLEANUP SECTION
* =-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=-=
*/

/* The PNA is returned to front panel control. */
ibonl(pna, 0);
return 0;
}

/*
* Write to the PNA
*/
```c
void GPIBWrite(char* SCPIcmd)
{
    int length;
    char ErrorMsg[ERRMSGSIZE + 1];
    length = strlen(SCPIcmd);

    ibwrt (pna, SCPIcmd, length);
    if (ibsta & ERR)
    {
        strcpy(ErrorMsg, "Unable to write this command to PNA:/n");
        strcat(ErrorMsg, SCPIcmd);
        GPIBCleanup(pna, ErrorMsg);
        exit(1);
    }
}

/*
 * Read from the PNA
 */
char* GPIBRead(void)
{
    ibrd (pna, ValueStr, ARRAYSIZE);
    if (ibsta & ERR)
    {
        GPIBCleanup(pna, "Unable to read from the PNA");
        exit(1);
    } else
    { return ValueStr;
    }
}

/*
 * After each GPIB call, the application checks whether the call succeeded. If an
 * NI-488.2 call fails, the GPIB driver sets the corresponding bit in the global status
 * variable. If the call failed, this procedure prints an error message, takes the PNA
 * offline and exits.
 */
void GPIBCleanup(int Dev, char* ErrorMsg)
{
    printf("Error : %s/nibsta = 0x%x iberr = %d (%s)/n", ErrorMsg, ibsta, iberr, ErrorMnemonic[iberr]);
    if (Dev != -1)
    {
        printf("Cleanup: Returning PNA to front panel control/n");
        ibonl (Dev, 0);
    }
}
```
Load Error Terms during a Cal Sequence

This example requires that you already have a Cal Set named "foo" that contains a 1-port cal on port 1 and a 1-port cal on port 2.

This example starts a Guided Calibration specifying an Unknown Thru. It loads the 1-port Cals from the existing "foo" Cal Set, then recalculates the number of steps required to complete the cal. After loading the 1-port cals, only the Unknown Thru standard is left to acquire.

```
SENS:CORR:COLL:GUID:CONN:PORT1 "APC 3.5 female"
SENS:CORR:COLL:GUID:CONN:PORT2 "APC 3.5 female"
SENS:CORR:COLL:GUID:CKIT:PORT1 "85033D/E"
SENS:CORR:COLL:GUID:CKIT:PORT2 "85033D/E"
SENS:CORR:COLL:GUID:METH UNKN
' auto-create user calsets for SCPI
SENS:CORR:PREF:CSET:SAVU 1
SENS:CORR:COLL:GUID:INIT
' should return the number 7
SENS:CORR:COLL:GUID:STEPS?
' to port 1, from port 1 in calset
SENS:CORR:COLL:GUID:ETER:LOAD "foo",1,1
' to port 2, from port 2 in calset
SENS:CORR:COLL:GUID:ETER:LOAD "foo",2,2
' should now return the number 1
SENS:CORR:COLL:GUID:STEPS?
' measure the unknown thru
SENS:CORR:COLL:GUID:ACQ STAN1
' save the cal to new user calset
SENS:CORR:COLL:GUID:SAVE
```
Modify a Calibration Kit using SCPI

This Visual Basic program:

- Modifies Calibration kit number 3
- Completely defines standard #4 (thru)

To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

'Modifying cal kit number 3
Calkitnum = 3

'Designate the kit selection to be used for performing cal's
GPIB.Write "SENSe:CORRection:COLLect:CKIT:SELect " & Val(Calkitnum)

'Reset to factory default values.
GPIB.Write "SENSe:CORRection:COLLect:CKIT:RESet " & Val(Calkitnum)

'Name this kit with your own name
GPIB.Write "SENSe:CORRection:COLLect:CKIT:NAME 'My Cal Kit'"

'Assign standard numbers to calibration classes
'Set Port 1, class 1 (S11A) to be standard #8
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer1 8"
'Set Port 1, class 2 (S11B) to be standard #7
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer2 7"
'Set Port 1, class 3 (S11C) to be standard #3
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer3 3"
'Set Port 1, class 4 (S21T) to be standard #4
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer4 4"
'Set Port 2, class 1 (S22A) to be standard #8
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer5 8"
'Set Port 2, class 2 (S22B) to be standard #7
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer6 7"
'Set Port 2, class 3 (S22C) to be standard #3
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer7 3"
'Set Port 2, class 4 (S12T) to be standard #4
GPIB.Write "SENSe:CORRection:COLLect:CKIT:ORDer8 4"

'Set up Standard #4 completely
'Select Standard #4; the rest of the commands act on it
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard 4"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:FMIN 300KHz"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:FMAX 9GHz"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:IMPedance 50"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:DELay 1.234 ns"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:LOSS 23e6"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:C0 0"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:C1 1"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:C2 2"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:C3 3"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:L0 10"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:L1 11"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:L2 12"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:L3 13"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:LABel 'My Special Thru'"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:TYPE THRU"
GPIB.Write "SENSe:CORRection:COLLect:CKIT:STANdard:CHARacteristic Coax"
Perform a Guided 2-Port or 4-Port Cal using SCPI

This example performs a Guided 2-Port or 4-port Calibration using ONE set of calibration standards or an ECAL module.

A measurement must first be set up with desired frequency range, power, and so forth, ready to be calibrated. The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as *.vbs.

Learn how to setup and run the macro.

See Guided Cal SCPI commands

---

```vbs
Dim app
Dim scpi
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' To perform 2-port cal, Uncomment TwoPortGuidedCal()
' Then comment FourPortGuidedCal()

'Do 2-port Cal
'TwoPortGuidedCal()

'Do 4-port Cal
FourPortGuidedCal

Sub TwoPortGuidedCal()
' Select the connectors
scpi.Execute("sens:corr:coll:guid:conn:port1 ""APC 3.5 female""
scpi.Execute("sens:corr:coll:guid:conn:port2 ""APC 3.5 male""
scpi.Execute("sens:corr:coll:guid:conn:port3 ""Not used"
scpi.Execute("sens:corr:coll:guid:conn:port4 ""Not used"
MsgBox("Connectors defined for Ports 1 and 2")

' Select the Cal Kit for each port being calibrated.
scpi.Execute("sens:corr:coll:guid:ckit:port1 ""85052D"

' To use an ECAL module instead, comment out the above two lines
' and uncomment the appropriate lines below:
' Your ECAL module must already be connected
' via USB to the PNA.
```
Non-factory characterizations are specified as follows:

When two or more ECal modules with the same model number are connected also specify the serial number as follows:

When Disk Memory ECal user characterizations are used, specify both the User char and the serial number as follows:

MsgBox(“Cal kits defined for Ports 1 and 2”)

' Initiate the calibration and query the number of steps numSteps = GenerateSteps()
' Measure the standards, compute and apply the cal MeasureAndComplete(numSteps)

End Sub

Sub FourPortGuidedCal()

' Select the connectors
scpi.Execute("sens:corr:coll:guid:conn:port1 ""APC 3.5 female""")
scpi.Execute("sens:corr:coll:guid:conn:port2 ""APC 3.5 female""")
MsgBox("Connectors defined for Ports 1 to 4")

' Select the Cal Kit for each port being calibrated.

' To use an ECal module instead, comment out the above four lines
' and uncomment these four lines and use the part number printed on your module (which in our case was N4431-60003), followed by the word 'ECal'. Your ECal module must already be connected via USB to the PNA.

' see above for ECal options
MsgBox("Cal kits defined for Ports 1 to 4")

' Initiate the calibration and query the number of steps numSteps = GenerateSteps()
If your selected cal kit is not a 4-port ECal module which can
mate to all 4 ports at once, then you may want to choose which
thru connections to measure for the cal. You must measure at
least 3 different thru paths for a 4-port cal (for greatest
accuracy you can choose to measure a thru connection for all 6
pairings of the 4 ports). If you omit this command, the default
is to measure from port 1 to port 2, port 1 to port 3, and
port 1 to port 4. For this example we select to measure
from port 1 to port 2, port 2 to port 3, and port 2 to port 4.
scpi.Execute("sens:corr:coll:guid:thru:ports 1,2,2,3,2,4")
'Re-generate the connection steps to account for the thru changes
numSteps = GenerateSteps()
'Measure the standards, compute and apply the cal
MeasureAndComplete(numSteps)
End Sub

Function GenerateSteps()
'Initiate the calibration and query the number of steps
scpi.Execute("sens:corr:coll:guid:init")
GenerateSteps = scpi.Execute("sens:corr:coll:guid:steps?")
End Function

Sub MeasureAndComplete(numSteps)
MsgBox("Number of steps is " + CStr(numSteps))
'Measure the standards
For i = 1 To numSteps
step = "Step " + CStr(i) + " of " + CStr(numSteps)
MsgBox strPrompt, vbOKOnly, step
'Note: if you have set up a slow sweep speed (for example, if
'you're using a narrow IF bandwidth) or you're using ECal, and
'through a cal step is being measured you wish to have your program
'perform other operations (like checking for the click event of a
'Cancel button) and you're NOT using the COM ScpiStringParser,
you can use the optional ASYNchronous argument with the ACQuire
'command as shown in this commented-out line below. The SCPI
'parser then will return immediately while the cal step measurement
'proceeds (i.e., the parser will NOT block-and-wait for the
'measurement step to finish, so you can send additional commands
'in the meantime). So you can do "*ESR?" or "*STB?" queries to
'monitor the status register bytes to see when the OPC bit gets set,
'which indicates the cal measurement step has finished. This OPC
'detection works for all of the PNA’s SCPI parsers except the COM
'ScpiStringParser.
"sens:corr:coll:guid:acq STAN" + CStr(i) + ",ASYN;*OPC"
scpi.Execute("sens:corr:coll:guid:acq STAN" + CStr(i))
Next
'Conclude the calibration
MsgBox ("Cal is done!")
End Sub
Last modified:

5-Apr-2011  edited for ECal options
Perform a Simple Source Power Cal

This example performs a Source Power Cal using ONE USB Power Sensor, already connected to the PNA. A measurement must first be set up with desired frequency range, power, and so forth, ready to be calibrated.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as spc.vbs.

Learn how to setup and run the macro.
See Source Power Cal SCPI commands

See Other SCPI Example Programs

'Performs a source power cal on channel 1 - port 1 using a USB power sensor
'This example assumes ONE USB power sensor is connected to the PNA
Dim app
Dim scpi
Dim sensor
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.parse "SYST:PRES"

' set power accuracy tolerance and iterations
scpi.parse "SOUR1:POW1:CORR:COLL:ITER:NTOL 0.1"
scpi.parse "SOUR1:POW1:CORR:COLL:ITER:COUN 15"

' set power sensor settling tolerance
scpi.parse "SOUR1:POW1:CORR:COLL:AVER:NTOL 0.1"
scpi.parse "SOUR1::POW1:CORR:COLL:AVER:COUN 15"

' set offset value for amp or attenuation
scpi.parse "SOUR1:POW1:CORR:OFFS 0 DB"

' show source power cal dialog
scpi.parse "SOUR1:POW1:CORR:COLL:DISP ON"

' read the usb power sensor ID string
sensor=scpi.parse("SYST:COMM:USB:PMET:CAT?")

'specify that sensor
scpi.parse "SYST:COMM:PSEN usb," + sensor
'do the measurement
scpi.parse "SOUR1:POW1:CORR:COLL:ACQ PMR,"ASENSOR"
'save the source cal and create an R-Channel response calset
scpi.parse "SOUR:POW:CORR:COLL:SAVE RREC"
Perform a Source and Receiver Power Cal using SCPI

Programming the PNA using COM or using SICL/VISA over LAN (as in this example) leaves the PNA free to control GPIB devices as needed.

The first example, using Visual Basic, demonstrates the following:

- Performing a source power calibration of Port 2 for Channel 1.
- Reading the calibration data.

The second example performs a Receiver Power Cal using VBScript.

Learn more about Power Calibrations.

See an example that Uploads a Source Power Cal.

Other SCPI Example Programs

To run this program, you need:

- One of the following power meters connected to the PNA through GPIB: E4416A, E4417A, E4418A/B, E4419A/B, 437B, 438A, EPM-441A, EPM-442A

Note: If your power meter is other than these, you can create your own Power Meter Driver using our template.

- Your PC and PNA both connected to a LAN (for communicating with each other).
- The SICL and VISA components of Agilent’s I/O Libraries software installed on your PC (both are included when you install the software, unless you already have another vendor’s VISA installed. Then specify Full SICL and VISA installation to overwrite the other vendor’s VISA.
- The module visa32.bas added to your VB project.
- A form with one button labeled cmdRun.
- A VISA interface configured on your remote PC to control the PNA. This could be GPIB interface or a VISA LAN Client.
- On the PNA connect a Thru cable from port 1 to port 2.

Note: The SOURce:POWer:CORRection:COLLect:ACQuire command, when used with a power meter, cannot be sent over the GPIB unless the power meter is connected to a different GPIB interface. See the alternative methods described in the command details.

'Session to VISA Default Resource Manager

Private defRM As Long
'Session to PNA
Private viPNA As Long
'VISA function status return code
Private status As Long

Private Sub Form_Load()
defRM = 0
End Sub

Private Sub cmdRun_Click()
' String to receive data from the PNA.
' Dimensioned large enough to receive scalar comma-delimited values
' for 21 frequency points (20 ASCII characters per point)
Dim strReply As String * 420

Dim strStimulus, strCalValue
Dim strResult As String

' Open the VISA default resource manager
status = viOpenDefaultRM(defRM)
If (status < VI_SUCCESS) Then HandleVISAError

' Open a session (viPNA) to the PNA at "address 16" on the VISA
' interface configured as "GPIB1" on this PC. This could be a
' VISA LAN Client pointing to the SICL LAN Server on the PNA, or
' an actual GPIB interface on this PC connected to the PNA GPIB
' (in which case the power meter would need to be connected to a
' different GPIB interface on the PNA, such as the Agilent 82357A
' USB-to-GPIB).
status = viOpen(defRM, "GPIB0::16::INSTR", 0, 0, viPNA)
If (status < VI_SUCCESS) Then HandleVISAError

' Set the number of sweep points to 21 on Channel 1.
status = myGPIBWrite(viPNA, "SENS1:SWE:POIN 21")
If (status < VI_SUCCESS) Then HandleVISAError
'Specify the GPIB address of the power meter
that will be used in performing the calibration.
status = myGPIBWrite(viPNA, "SYST:COMM:GPIB:PMET:ADDR 13")
If (status < VI_SUCCESS) Then HandleVISAError

'Turn use of the loss table OFF (this assumes there is
'virtually no loss in the RF path to the power sensor
'due to a splitter, coupler or adapter).
status = myGPIBWrite(viPNA, "SOUR:POW:CORR:COLL:TABL:LOSS OFF")
If (status < VI_SUCCESS) Then HandleVISAError

'Turn frequency checking OFF (so one power sensor is used for the entire cal
'acquisition sweep regardless of frequency span).
status = myGPIBWrite(viPNA, "SOUR:POW:CORR:COLL:FCH OFF")
If (status < VI_SUCCESS) Then HandleVISAError

'Specify a nominal power accuracy tolerance (NTOLerance) in dB for the
calibration,
'and the maximum number (COUNT) of iterations to adjust power at each point,
'attempting to achieve within tolerance of the desired power. If at any
stimulus
'point the power fails to reach within the set tolerance of the desired power
'after the maximum number of iterations, the power at that point will be set to the
'value determined by the last iteration (the Source Power Cal dialog box will
'indicate the FAIL, but we can still apply the cal if desired when it's
complete).
'Each iteration is based upon a SETTLED power reading (see comments preceding the
'next two commands below).
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:ITER:NTOL 0.1")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:ITER:COUN 3")
If (status < VI_SUCCESS) Then HandleVISAError

'The worst-case window of power uncertainty (for a calibration which meets
tolerance) is the sum of the iteration tolerance and the power meter settling
tolerance (which is described below).
At each stimulus point, the PNA takes power meter readings and determine when they have settled by comparing the magnitude difference between consecutive readings versus a nominal dB tolerance limit (NTOLerance) on that magnitude difference. When consecutive readings are within tolerance of each other, or if they are not within tolerance but we've taken a maximum number of readings (COUNT), the PNA does a weighted average of the readings taken at that stimulus point and that is considered our settled power reading.

```vbnet
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:AVER:NTOL 0.1")
If (status < VI_SUCCESS) Then HandleVISAError

status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:AVER:COUN 5")
If (status < VI_SUCCESS) Then HandleVISAError

' Specify if the cal power level is offset (positive value for a gain, negative value for a loss) from the PNA port power setting on the channel when no source power cal is active. This is to account for components between the PNA test port and cal reference plane. In this example, we will calibrate at the PNA test port, so there is no offset (it is zero).

status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:OFFS 0 DB")
If (status < VI_SUCCESS) Then HandleVISAError

' Show the source power cal dialog during the source power cal acquisition. (this is the default, so this command is only necessary if this setting may have been changed beforehand, perhaps by another program).

status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:DISP ON")
If (status < VI_SUCCESS) Then HandleVISAError

' Clear the PNA's SCPI status registers because we are going to be monitoring the event status register to detect when the OPC bit gets set indicating the cal ACQuire completed.

status = myGPIBWrite(viPNA, "*CLS")
If (status < VI_SUCCESS) Then HandleVISAError

' Setting the I/O timeout value to 6000 milliseconds (6 seconds), because the PNA may take up to that amount of time to respond to some commands/queries while the cal ACQuire is progressing.

status = viSetAttribute(viPNA, VI_ATTR_TMO_VALUE, 6000)
```
If (status < VI_SUCCESS) Then HandleVISAError

' Specify the method (type of device) that will be used to perform the cal.
' Choose from power meter (PMETer), power meter and receiver (PMReceiver)
' or just receiver (RECeiver).
' PMReceiver uses the power meter for the first iteration of each point and
' the PNA's reference receiver for subsequent iterations, so is much faster
' than using power meter only. But the power meter accounts for compression
' when calibrating at the output of an active device, whereas the reference
' receiver cannot unless it is coupled to the cal reference plane (on a PNA
' which allows direct access to the receivers).
' Perform the source power cal acquisition sweep using the sensor attached to
' Channel A of the power meter (asking for an OPC reply when it’s done). This
' assumes that the power sensor is already connected to Port 2 of the PNA.

' We'll put up an hourglass cursor while waiting for the acquire to complete.
Screen.MousePointer = vbHourglass

status = myGPIBWrite(viPNA, "SOUR1:POW:CORR:COLL:ACQ PMET,'ASEN','Port 2',ASYNchronous;$OPC")
If (status < VI_SUCCESS) Then HandleVISAError

' Other valid selections would be the following:
' This mode uses Power Meter and Reference Receiver
'Status = myGPIBWrite(viPNA, "SOUR1:POW:CORR:COLL:ACQ PMR,'BSEN','Port 2',ASYN;$OPC")
' This mode uses PNA receiver only (no power meter)
'Status = myGPIBWrite(viPNA, "SOUR1:POW:CORR:COLL:ACQ REC,'b1','Port 2',ASYN;$OPC")

' Polling in a loop to detect when the OPC bit (bit 0, weight value 1) gets
' set in the Event Status Register indicating the ACQuire finished. In this
' type of loop is where you could do other operations in-between the polling
' (like having your program’s user-interface still respond to user input).
' If instead of Visual Basic you are programming in C or C++, as an
' alternative to having a polling loop like this, you could set up an SRQ
' handling function in your program (for example, see the documentation
' supplied with your vendor’s implementation of VISA on how to register for
' callback when an SRQ event occurs).

Do
status = myGPIBWrite(viPNA, "*ESR?")
    If (status < VI_SUCCESS) Then HandleVISAError
    status = myGPIBRead(viPNA, strReply)
    If (status < VI_SUCCESS) Then HandleVISAError
    esrByte = CByte(strReply)
Loop While (esrByte And 1) = 0

' Change mouse cursor from hourglass back to normal
Screen.MousePointer = vbDefault

' Conclude the calibration. This applies the cal data to PNA channel memory,
' and turns the correction ON for Port 2 on Channel 1,
' but does NOT save the calibration.
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:COLL:SAVE")
    If (status < VI_SUCCESS) Then HandleVISAError

' At this point, if you choose to save the instrument state as a ".CST" file,
' the calibration will be saved with the instrument state in that file.

' Prepare for doing data transfer in ASCII format.
status = myGPIBWrite(viPNA, "FORM:DATA ASCII")
    If (status < VI_SUCCESS) Then HandleVISAError

' Read the stimulus values from Channel 1.
status = myGPIBWrite(viPNA, "SENS1:X?")
    If (status < VI_SUCCESS) Then HandleVISAError
    status = myGPIBRead(viPNA, strReply)
    If (status < VI_SUCCESS) Then HandleVISAError

' Tokenize the reply string into an array containing the values
strStimulus = Split(strReply, ",")

' Read the source power correction data.
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:DATA?")
    If (status < VI_SUCCESS) Then HandleVISAError
    status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError

' Tokenize the reply string into an array containing the values
strCalValue = Split(strReply, ",")

' Print the data using a message box (here, Chr returns the ASCII characters
' for Tab (9) and Linefeed (10)).
strResult = "Stimulus" & Chr(9) & Chr(9) & " Cal Value" & Chr(10)
For i = 0 To UBound(strStimulus)
strResult = strResult & Val(strStimulus(i)) & Chr(9) & Val(strCalValue(i)) & Chr(10)
Next
MsgBox strResult
End Sub

Private Function myGPIBWrite(ByVal viHandle As Long, ByVal strOut As String) As Long

' The "+ Chr$(10)" appends an ASCII linefeed character to the
' output, for terminating the write transaction.
myGPIBWrite = viVPrintf(viHandle, strOut + Chr$(10), 0)
End Function

Private Function myGPIBRead(ByVal viHandle As Long, strIn As String) As Long
myGPIBRead = viVScanf(viHandle, "%t", strIn)
End Function

Sub HandleVISAError()
Dim strVisaErr As String * 200
Call viStatusDesc(defRM, status, strVisaErr)
MsgBox "*** Error : " + strVisaErr, vbExclamation

' Close the resource manager session (which also closes
' the session to the PNA).
If defRM <> 0 Then Call viClose(defRM)
End
Public Sub Wait(ByVal mS_delay As Long)

Dim t0 As Single

    t0 = Timer

Do While Timer - t0 < mS_delay / 1000

    Dim dummy As Integer

    dummy = DoEvents() ' if we cross midnight, back up one day

    If Timer < t0 Then t0 = t0 - 86400

Loop

End Sub

Perform a Receiver Power Cal

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as *.vbs.

Learn how to setup and run the macro.

Dim pna
Dim scpi
Set pna = CreateObject("AgilentPNA835x.Application")
Set scpi = pna.ScpiStringParser

' For simplicity, this example starts from the preset instrument state
scpi.Execute "SYST:PRESet"

' Turn off continuous sweep
scpi.Execute "INITiate:CONTinuous OFF"

' Select the S11 measurement that was created by the instrument preset
scpi.Execute "CALCulate:PARameter:SELect 'CH1_S11_1'"

' Change the measurement parameter to measure the B receiver
scpi.Execute "CALCulate:PARameter:MODify B,1"

' Specify the Calibration Type, then Prompt
to ensure the receiver is connected to port 1.
scpi.Execute "SENSe:CORRection:COLLect:METHOD RPOWer"

MsgBox "Connect port 1 to port 2 so power is supplied to the B receiver, then press enter"

' Acquire the power measurement; returning reply to *OPC? when finished.
response = scpi.Execute( "SENSe:CORRection:COLLect:ACQuire POWer;*OPC?" )

' Compute the error term, store to calset and turn on the calibration.
response = scpi.Execute( "SENSe:CORRection:COLLect:SAVE" )

MsgBox "Done with calibration."
9/22/06  Modified for receiver only feature
Perform a Source Power Cal with TWO Sensors

This example performs a Source Power Cal using TWO power sensors on a Dual-Channel Power Meter connected to the PNA via GPIB. Use of two power sensors is necessary when the frequency range of the measurement is greater than can be attained with a single sensor.

A measurement must first be set up with desired frequency range, power, and so forth, ready to be calibrated.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do not need to control the PNA via GPIB to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as spc.vbs.

Learn how to setup and run the macro.

See Source Power Cal SCPI commands

See Other SCPI Example Programs

'Performs a source power cal on channel 1 – port 1 using TWO USB power sensors.

Dim app
Dim scpi
Dim sensor

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.parse "SYST:PRES"

' set power accuracy tolerance and iterations
scpi.parse "SOUR1:POW1:CORR:COLL:ITER:NTOL 0.1"
scpi.parse "SOUR1:POW1:CORR:COLL:ITER:COUN 15"

' set power sensor settling tolerance
scpi.parse "SOUR1:POW1:CORR:COLL:AVER:NTOL 0.1"
scpi.parse "SOUR1:POW1:CORR:COLL:AVER:COUN 15"

' set offset value for amp or attenuation
scpi.parse "SOUR1:POW1:CORR:OFFS 0 DB"

' show source power cal dialog
scpi.parse "SOUR1:POW1:CORR:COLL:DISP ON"

' set frequency ranges for two power sensors
scpi.parse "SOUR:POW:CORR:COLL:ASEN 100E3, 3E9"
scpi.parse "SOUR:POW:CORR:COLL:BSEN 3E9,6E9"

'enable frequency check
scpi.parse "SOUR1:POW:CORR:COLL:FCHeck ON"
'specify the address of the power meter
scpi.parse "SYST:COMM:PSEN GPIB,'14'"
'do the measurements
MsgBox "Connect Sensor A to Port 1"
scpi.parse "SOUR1:POW1:CORR:COLL:ACQ PMR,'ASENSOR'"
MsgBox "Disconnect Sensor A from Port 1 and connect Sensor B to Port 1"
scpi.parse "SOUR1:POW1:CORR:COLL:ACQ PMR,'BSENSOR'"
'save the source cal and create an R-Channel response calset
scpi.parse "SOUR:POW:CORR:COLL:SAVE RREC"
Perform an ECal User Characterization

This example performs a user-characterization and stores it to both the ECal module memory and PNA disk memory. It also demonstrates the use of the EXPort, CLEar, IMPort and ‘KNAMe:INF?’ commands.

It then performs two 2-port cals: the first using the characterization from module memory, then using the characterization from disk memory.

**Note:** This example requires that channel 1 be already calibrated.

The SCPI commands in this example are sent over a COM interface using the **SCPIStringParser** object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file, such as Notepad, and save it on the PNA hard drive as ECal.vbs.

**Learn how to setup and run the macro.**

**See all ECal User Characterization SCPI commands**

```
Option Explicit

Dim pna
Set pna = CreateObject("AgilentPNA835x.Application")
Dim scpi
Set scpi = pna.ScpiStringParser

' Substitute here the model number and serial number of your own ECal.
' Note that this example corresponds to a 4-port ECal module with
' serial number 00001. If you have a 2-port ECal module, their model
' numbers are '5x5' numbers -- for example, 'N4691-60001'.
Dim ecalModelNum
ecalModelNum = "N4433A"
Dim ecalSerialNum
ecalSerialNum = "00001"

MsgBox "ECal module to be characterized is: " & 

' Set which user characterization number (1-12) the new characterization
' will be stored to in the ECal module when it is done. If you intend to
' store your user characterization just to PNA Disk Memory and NOT the
' ECal module's memory, then omit this command.
```
Dim characterizationNumber
characterizationNumber = 1

' The following commented-out lines of code show how you can access
' the list of connector type names you can set for the ports of an
' ECal when you user-characterize it. However, please note that if
' you are writing the user characterization to the ECAL module's memory,
' as of yet only the Factory Defined set of connector choices will work
' properly (see SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?). If you will be saving
' your characterization to just PNA Disk Memory only, then all connector
' names returned by this query will work,
' user-defined connector names as well as factory-defined.

'Dim connTypeList
'MsgBox connTypeList

' For each port of the ECAL module, specify which connector type
' is at the end of the adapter (or cable or fixture) that is
' connected to that port of the ECAL for the characterization
' (must be one of the connector types that is included in the
' list that "SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?" returns). The
default is "No adapter", which assumes you are characterizing that
' port of the ECAL "as is" (nothing attached to it). So in this
' example, Ports C and D of the ECAL are being characterized to just
' the ECAL's connectors.
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:CONN:PORT1 'APC 3.5 male'" ' ECAL Port A

' As with the connector types, the information set in these next
' few properties also gets stored within the characterization.
' Set the name of the person and/or company that is producing
' this characterization.
' Set user-specified description of the PNA being used.
' Set descriptions of what you have connected to the ECAL module's
' ports for the characterization.
' Port A of the ECal
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:DESC:PORT1 '3.5 mm adapter, SN 00001'"
' Port B of the ECal
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:DESC:PORT2 '3.5 mm adapter, SN 00002'"
' Note that the "SENS:CORR:CKIT:ECAL:CHAR:" INITiate, ACQuire and SAVE
' ("CHAR:SAVE" but not "CHAR:DMEMory:SAVE") commands can all each take a
' significant
' amount of time to execute/complete. If you are looking at this example to
' leverage this functionality into a SCPI via GPIB or SCPI via SICL-LAN
' (VXI-11.2/11.3) application, then you could issue the "*CLS" and "*ESE 1"
' commands
' as shown in the commented-out lines below, and use your I/O libraries' Serial Poll
' function to repeatedly read the status byte until you detect bit 5 (weight of
' in that byte is set. That will happen when the command you are pairing with
' ";*OPC" has completed its operation. But that technique only works for the
' and SICL-LAN interfaces. If you need to use the TCPIP Socket or COM
' ScpiStringParser (as is used in this example) SCPI interfaces where there's
' no "built-in" Serial Poll type of function, to ensure your program operates in a
' synchronized manner it will need to wait on the "*OPC?" reply (and not time out)
' before proceeding to the next line of your program. In that event, we
' recommend
' you execute these commands on a thread of execution separate from your
' program's
' user interface thread.
' Of the "SENS:CORR:CKIT:ECAL:CHAR:" INITiate, ACQuire and SAVE commands, the
' SAVE
' command takes the longest amount of time to complete (unless you've set up your
' measurement channel to have a very slow sweep time, in which case the ACQuire
' command could take longer). For an ECal that is a N469x, N4432A or N4433A, or an
' 8509x or N4431x produced by Agilent in 2005 or later, the SAVE command can take a
' maximum of approximately 4 to 5 minutes to complete (that corresponds to a
' characterization that will result in the ECal's memory becoming completely
For an 8509x or N4431x ECal that was produced in 2004 or earlier, the SAVE command can take a maximum of 9 to 10 minutes to complete (again that corresponds to a characterization that will result in the ECal's memory becoming completely full).

Begin a user characterization on Channel 1.

If you will be storing this characterization to the ECal module's memory, then the boolean argument to this command is optional (but if you choose to include it for that case then you must specify it as 1 or ON). If you will be storing this characterization to PNA disk memory ONLY, then you should specify 0 or OFF for that argument. In this example we will be storing the characterization to both module memory and PNA disk memory, so we can just omit the argument and let it default to 1.

```scpi```
```
```

```vba```
```
Dim numSteps
Dim opcReply
'Dim statusByte
'Measure the steps.
'Note: prior to measuring the steps you must already have a calibration of the necessary number of ports applied to the channel (which in this example is Channel 1).
'Otherwise an error will be reported to the SCPI error queue.
Dim i
For i = 1 To numSteps
  'Display the step's description.
  'Clear the instrument's Status Byte.
  'scpi.Execute "*CLS"
  'Enable for the OPC bit (bit 0, which has weight 1) in the instrument's Event Status Register, so that when that bit's value transitions from 0 to 1 then the Event Status Register bit in the Status Byte (bit 5 of that byte, weight 32) will become set.
  'scpi.Execute "*ESE 1"
  'Issue the ACQuire command
```
'    scpi.Execute "SENS1:CORR:CKIT:ECAL:CHAR:ACQ STAN" & CStr(i) & ";*OPC"
'    Do
'        here is where if you leverage this example into an environment where
'        you are using SCPI via GPIB or SICL-LAN, that in this loop you could do a
'        Serial Poll via that interface to read the status byte into this
'        statusByte variable. Then this If statement would detect when bit 5 is
'        set.
'        If ( (statusByte/32) Mod 2) Then Exit Do
'        And note that normally you would want to have your program do some other
'        processing (for example, check for user input from keyboard/mouse, for
'        a cancellation request) here in this loop.
'    Loop
    MsgBox "ACQuire is complete"
Next
MsgBox "Now the user characterization will be saved to the ECal module and to PNA
disk memory"
'    scpi.Execute "*CLS:*ESE 1"
'    Save the user characterization to the ECal module's memory.
'Do
'    again here you could do a Serial Poll to get statusByte if using GPIB or
SICL-LAN
'    If ( (statusByte/32) Mod 2) Then Exit Do
'    Loop
'    Save the user characterization to PNA Disk Memory.
Dim characterizationName
characterizationName = "test"
Dim pnaDiskMemCalKitName
pnaDiskMemCalKitName = GetCalKitName(characterizationName)
'    Exporting the characterization from PNA disk memory into a file.
'    The file can be used for loading the characterization into PNA disk memory on
another PNA.
'Demonstrating that the characterization can be cleared from PNA disk memory and then re-loaded (IMPORTed) from the file that was created by the "SENS:CORR:CKIT:ECAL:EXP".

```vbnet
Dim moduleMemCalKitName
moduleMemCalKitName = GetCalKitName("User " & CStr(characterizationNumber))
MsgBox "Information about the characterization from ECal module memory = " & 
MsgBox "Information about the characterization from PNA disk memory = " & 
MsgBox "User characterization is complete. Now we will calibrate using it. First we will use it from ECal module memory."
DoTwoPortCal moduleMemCalKitName
MsgBox "Now we will calibrate using the characterization from PNA Disk Memory."
DoTwoPortCal pnaDiskMemCalKitName
MsgBox "Example has completed"
```

Function GetCalKitName(characterizationName)
Dim calKitName
    calKitName = ecalModelNum
    If Len(characterizationName) > 0 Then calKitName = calKitName & " " & characterizationName
    calKitName = calKitName & " ECal " & ecalSerialNum
    GetCalKitName = calKitName
End Function

Sub DoTwoPortCal(calKitName)
    ' Specify the DUT connector for each PNA port to be calibrated (DUT connector = ECal characterization's connector)
    scpi.Parse "SENS1:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 male'"
    scpi.Parse "SENS1:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 male'"
    ' Specify the "cal kit" for each of those ports
    scpi.Parse "SENS1:CORR:COLL:GUID:CKIT:PORT1 '" & calKitName & ""
    scpi.Parse "SENS1:CORR:COLL:GUID:CKIT:PORT2 '" & calKitName & ""
    ' This results in a calibration sequence of a single "connection step"
    scpi.Parse "SENS1:CORR:COLL:GUID:INIT"
```
' Acquire the cal connection step
opcReply = scpi.Parse("SENS1:CORR:COLL:GUID:ACQ STAN1:*OPC?")
' Again here instead of waiting for opcReply you could do a Serial Poll to get statusByte if using GPIB or SICL-LAN
'scpi.Execute "SENS1:CORR:COLL:GUID:ACQ STAN1:*OPC"
'Do
' If ( (statusByte/32) Mod 2) Then Exit Do
'Loop
' Conclude the cal and turn it on
scpi.Parse "SENS1:CORR:COLL:GUID:SAVE"
End Sub
Perform an Unguided Cal on a 4-Port PNA

This topic describes how to perform an unguided calibration on a multiport network analyzer using SCPI. The objective here is to make clear the relationship between the physical port on which a standard is being measured, the actual device in the cal kit, and the SCPI command used to acquire the device.

There are two sets of SCPI commands that acquire calibrations. One set is used for guided cal, the other for unguided. The SCPI commands that provide remote access to unguided cal are in the SENS:CORR:COLL block:

- SENS:CORR:COLL:METHod
- SENS:CORR:COLL:ACQuire
- SENS:CORR:COLL:SAVE

On a four port network analyzer, the remote programmer needs to be aware of the relationship between the physical port and the calibration kit class assignments. The example program (below) illustrates the usage by performing three unique 2 port cals, taking care to acquire the appropriate standards.

Calibration standards classes are ‘categories’ of standard types. To perform a 2 port calibration, the cal wizard requires the user to measure:

3 reflection standards on the forward port:

- Class S11A typically an open
- Class S11B typically a short
- Class S11C typically a load

Likewise, 3 reflection standards are required for the reverse port:

- Class S22A typically an open
- Class S22B typically a short
- Class S22C typically a load

There is also a transmission standard that is measured in both directions:

- Class S21T typically a thru

The following illustrates the relationship between cal kit physical standards and calibration classes.

Here is a list of the physical devices in my calibration kit.

Standard #1 = "3.5 mm male short"
Standard #2 = "3.5 mm male open"
Standard #3 = "3.5 mm male broadband load"
Standard #4 = "Insertable thru standard"
Standard #5 = "3.5 mm male sliding load"
Standard #6 = "3.5 mm male lowband load"
Standard #7 = "3.5 mm female short"
Standard #8 = "female to female characterized thru adapter"
Standard #9 = "0-2 Load"
Standard #10 = "Open"
Standard #11 = "Non-insertable thru"
Standard #12 = "3.5 mm female lowband load"
Standard #13 = "3.5 mm female sliding load"
Standard #14 = "3.5 mm female broadband load"
Standard #15 = "3.5 mm female open"

When you perform a calibration remotely using SCPI, you don’t specify the device number directly. Rather, you specify the class you want to measure. Each device in the calibration kit is assigned to a class. And since more than one device can be assigned to the same class, each class contains an ordered list of devices. The class assignments are user-settable using the Advanced Modify Cal Kit dialog or the SCPI command:

```
SENS:CORR:COLL:CKIT:ORDer <class>, <std>, <std>, <std>, <std>,<std>,<std>,<std>
```

The 85052B kit used in the example program had the following standard list for each class: The list was obtained by issuing the corresponding SCPI query:

```
SENS:CORR:COLL:CKIT:OLIST1? S11A = +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST2? S11B = +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST3? S11C = +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST4? S21T = +4,+8,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST5? S22A = +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST6? S22B = +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST7? S22C = +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST8? S12T = +4,+8,+0,+0,+0,+0,+0
```

When you perform the calibration, you acquire data by issuing the ACQuire command:

```
SENS:CORR:COLL:ACQ <class>[, <substd> ]
```

For example:

```
SENS:CORR:COLL:SFOR 1
SENS:CORR:COLL:ACQ STANA, SST2
```

The SFOR command tells the wizard to make the next acquisition in the forward direction. The ACQuire command specifies that we are measuring the 2nd device in the list for STANA. And since we are measuring SFORward, STANA refers to class #1 or S11A. The list of devices for this class are specified in the OLIST1 query above. The associations are shown in red.

Alternately, you could modify the device order for the S11A class to move device #15 into the first position (SENS:CORR:COLL:CKIT:ORDER1). When the desired device is in the first position, you needn’t specify the order number in the ACQuire command. The default is the first device in the OLIST. This worked well for two port network analyzers where the order for S11A,B,C classes were setup for port 1 and the order for S22A,B,C was set up for port 2. With the kit setup in the proper order, you could eliminate the specification of the substandard
When performing 2 port calibrations on 4 Port Network Analyzers (e.g.: PNA Model N5230A), the wizard applies S11A,B,C standards to the lower numbered port, S22A,B,C standards to the higher numbered port. Since the two classes (S11A,B,C and S22A,B,C) are applied to multiple ports, the programmer must take into account the ports being measured and take greater care when specifying the ACQuire command to ensure that the correct device is being measured.

### Port to class relationship

<table>
<thead>
<tr>
<th>Ports</th>
<th>S11A Port</th>
<th>S22A Port</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,2</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>1,3</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>1,4</td>
<td>1</td>
<td>4</td>
</tr>
<tr>
<td>2,3</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>2,4</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>3,4</td>
<td>3</td>
<td>4</td>
</tr>
</tbody>
</table>

The following example program shows one method of handling two port cals on a multiport network analyzer. The connectors at the measurement plane are assumed to be (1) male, (2) female, (3) male, and (4) male. In the example, three cals are performed: 1-2 (insertable male to female), 2-3 (insertable female to male), and 3-4 (noninsertable using an characterized adapter).

```visualbasic
option explicit
public scpi
public pna
' assume a 4 port PNA with the following connectors:
' the standard measured on these ports will be the opposite gender
' PORT 1 = 3.5 male
' PORT 2 = 3.5 female
' PORT 3 = 3.5 male
' PORT 4 = 3.5 male
' To perform 2 port calibrations between 1-2, 2-3, and 3-4 you need to do the following

call main

sub main
set pna = CreateObject("AgilentPnA835x.Application")
set scpi = pna.ScpiStringParser
pna.Preset
' select a kit to use for this demonstration
' kit #1 for the N5230A is the 85052B 3.5mm kit with sliding load
scpi.execute("SENS:CORR:COLL:CKIT:SELECT 1")
```
wscript.echo "Calibrating ports 1 and 2"
scpi.execute("SYST:PRES;")
scpi.execute("calc:par:sel CH1_S11_1")
scpi.execute("SENS:CORR:TST:STATE 0")
scpi.execute("SENS:CORR:COLL:METHod SPARSOLT")
scpi.execute("SENS:CORR:SFOR 1")
MeasureFemaleStandards 1
scpi.execute("SENS:CORR:SFOR 0")
MeasureMaleStandards 2
MeasureTransmissionStandards 1, 2
scpi.execute("SENS:CORR:COLL:SAVE")

wscript.echo "Calibrating ports 2 and 3"
scpi.execute("SYST:PRES;")
scpi.execute("calc:par:sel CH1_S11_1")
scpi.execute("calc:par:mod S23")
scpi.execute("SENS:CORR:TST:STATE 0")
scpi.execute("SENS:CORR:COLL:METHod SPARSOLT")
scpi.execute("SENS:CORR:SFOR 1")
MeasureMaleStandards 2
scpi.execute("SENS:CORR:SFOR 0")
MeasureFemaleStandards 3
MeasureTransmissionStandards 2, 3
scpi.execute("SENS:CORR:COLL:SAVE")

wscript.echo "Calibrating ports 3 and 4"
scpi.execute("SYST:PRES;")
scpi.execute("calc:par:sel CH1_S11_1")
scpi.execute("calc:par:mod S43")
scpi.execute("SENS:CORR:COLL:METHod SPARSOLT")
scpi.execute("SENS:CORR:SFOR 1")
MeasureFemaleStandards 3
scpi.execute("SENS:CORR:SFOR 0")
MeasureFemaleStandards 4
MeasureAdapter 3, 4
scpi.execute("SENS:CORR:COLL:SAVE")
end sub
sub MeasureMaleStandards ( portNumber )
    dim portstr
    portstr = formatnumber(portNumber,0)
    Promptconnect1 1, 1, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN1;*OPC?"
    Promptconnect1 2, 1, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN2;*OPC?"
    Promptconnect1 3, 3, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN3,SST3;*OPC?"
end sub

sub MeasureFemaleStandards (portNumber)
    dim portstr
    portstr = formatnumber(portNumber,0)
    Promptconnect1 1, 2, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN1,SST2;*OPC?"
    Promptconnect1 2, 2, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN2,SST2;*OPC?"
    Promptconnect1 3, 6, portNumber
    scpi.execute("SENS:CORR:COLL:ACQ STAN3,SST6;*OPC?"
end sub

sub MeasureTransmissionStandards( port1, port2)
    dim p1str
    dim p2str
    p1str = formatnumber( port1, 0)
    p2str = formatnumber( port2, 0)
    Promptconnect2 4, 1, port1, port2
    scpi.execute("SENS:CORR:COLL:ACQ STAN4;*OPC?"
end sub

sub MeasureAdapter( port1, port2)
    dim p1str
    dim p2str
    p1str = formatnumber( port1, 0)
    p2str = formatnumber( port2, 0)
    Promptconnect2 4, 2, port1, port2
    scpi.execute("SENS:CORR:COLL:ACQ STAN4,SST2;*OPC?"
end sub

' return the nth item in the comma separated list
Function GetItemNumber( list, n)
    dim strVector
    strVector = split(list",",-1,1)
    GetItemNumber = strVector(n-1)
end function
' remove the trailing newline from str
function chop( str )
dim tmp
  tmp = str
  ' remove the appended newline
dim pos
  pos = InStrRev(tmp,vblf)
  if (pos >0) then
    tmp = mid(tmp,1,pos-1)
  end if
  chop = tmp
end function

' return the label for the nth standard assigned to the class described by class_index.
' if class_index = 1, class is S11A (STAN1)
' if class_index = 2, class is S11B (STAN2), etc
function GetStandardLabel( class_index, nth)
dim olist
  dim stdnum
  dim resp
  olist = scpi.execute("SENS:CORR:COLL:CKIT:OLIST" + formatnumber(class_index,0)+"?")
  stdnum = GetItemNumber( olist, nth)
  scpi.execute("SENS:CORR:COLL:CKIT:STAN " + formatnumber(stdnum,0))
  resp = scpi.execute("SENS:CORR:COLL:CKIT:STAN:LABel?")
  GetStandardLabel = chop(resp)
end function

sub PromptConnect1( class_index, nth, port)
  wscript.echo "CONNECT " + GetStandardLabel( class_index, nth) + " to port " + formatnumber(port,0)
end sub

sub PromptConnect2( class_index, nth, port1, port2)
  wscript.echo "CONNECT " + GetStandardLabel( class_index, nth) + " between ports " + formatnumber(port1,0) + " and " + formatnumber(port2,0)
end sub

' Print the order of standards per class for this kit
sub PrintKitOlist( kit )
dim i
  dim cmd
  dim resp
  wscript.echo
  dim olistcmd
  olistcmd = "SENS:CORR:COLL:CKIT:OLIST"
  ' list the sub standards for each of the following classes
  ' S11A, S11B, S11C, FWD TRANS, FWD ISOL, S22A, S22B, S22C, REV TRANS, REV ISOL
  for i = 1 to 8
    cmd = olistcmd + formatNumber(i,0) + "?"
    resp = scpi.execute(cmd)
    wscript.echo cmd + "= " + chop(resp)
  next
end sub
sub PrintKitStandardInfo( kit )
    wscript.echo scpi.execute("SENS:CORR:COLL:CKIT:NAME?")
    dim i
    for i = 1 to 30
        dim slabel
        dim snum
        snum = formatNumber(i,0)
        scpi.execute("SENS:CORR:COLL:CKIT:STAN " + snum)
        slabel = scpi.execute("SENS:CORR:COLL:CKIT:STAN:LABel?")
        wscript.echo "Standard #" + snum + " = " + chop(slabel)
    next
end sub

The output from this program is as follows:
Microsoft (R) Windows Script Host Version 5.6
Copyright (C) Microsoft Corporation 1996-2001. All rights reserved.
"85052B 3.5 mm with sliding load"
Standard #1 = "3.5 mm male short"
Standard #2 = "3.5 mm male open"
Standard #3 = "3.5 mm male broadband load"
Standard #4 = "Insertable thru standard"
Standard #5 = "3.5 mm male sliding load"
Standard #6 = "3.5 mm male lowband load"
Standard #7 = "3.5 mm female short"
Standard #8 = "Female to female characterized thru adapter"
Standard #9 = "0-2 Load"
Standard #10 = "Open"
Standard #11 = "Non-insertable thru"
Standard #12 = "3.5 mm female lowband load"
Standard #13 = "3.5 mm female sliding load"
Standard #14 = "3.5 mm female broadband load"
Standard #15 = "3.5 mm female open"
Standard #16 = "Open"
Standard #17 = "Open"
Standard #18 = "Open"
Standard #19 = "Open"
Standard #20 = "Open"
Standard #21 = "Open"
Standard #22 = "Open"
Standard #23 = "Open"
Standard #24 = "Open"
Standard #25 = "Open"
Standard #26 = "Open"
Standard #27 = "Open"
Standard #28 = "Open"
Standard #29 = "Open"
Standard #30 = "Open"

SENS:CORR:COLL:CKIT:OLIST1?= +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST2?= +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST3?= +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST4?= +4,+8,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST5?= +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST6?= +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST7?= +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST8?= +4,+8,+0,+0,+0,+0,+0

Calibrating ports 1 and 2
CONNECT "3.5 mm female open" to port 1
CONNECT "3.5 mm female short" to port 1
CONNECT "3.5 mm female broadband load" to port 1
CONNECT "3.5 mm male open" to port 2
CONNECT "3.5 mm male short" to port 2
CONNECT "3.5 mm male broadband load" to port 2
CONNECT "Insertable thru standard" between ports 1 and 2

Calibrating ports 2 and 3
CONNECT "3.5 mm male open" to port 2
CONNECT "3.5 mm male short" to port 2
CONNECT "3.5 mm male broadband load" to port 2
CONNECT "3.5 mm female open" to port 3
CONNECT "3.5 mm female short" to port 3
CONNECT "3.5 mm female broadband load" to port 3
CONNECT "Insertable thru standard" between ports 2 and 3

Calibrating ports 3 and 4
CONNECT "3.5 mm female open" to port 3
CONNECT "3.5 mm female short" to port 3

3037
CONNECT "3.5 mm female broadband load" to port 3
CONNECT "3.5 mm female open" to port 4
CONNECT "3.5 mm female short" to port 4
CONNECT "3.5 mm female broadband load" to port 4
CONNECT "female to female characterized thru adapter" between ports 3 and 4
Perform Global Delta Match Cal

The following program performs a Global Delta Match Calibration. This is required when performing an Unknown Thru Cal or TRL Cal on PNAs without a reference receiver for each test port. See example of Unknown Thru or TRL Cal.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Delta.vbs. Learn how to setup and run the macro.

```vbs
Sub PerformGlobalDeltaMatchCal()
    Set pna = CreateObject("AgilentPNA835x.Application")
    Set scpi = pna.ScpiStringParser

    ' Initiate a Global Delta Match calibration, choosing connector and cal kit
    scpi.Parse "SENS:CORR:COLL:GUID:DMAT 'APC 3.5 female', '85033D/E'"

    ' Query the number of calibration steps
    retStr = scpi.Parse("SENS:CORR:COLL:GUID:STEP?")
    numSteps = CInt(retStr)

    ' Measure the cal standards
    For i = 1 To numSteps
        retVal = MsgBox(prompt, vbOKCancel)
        If retVal = vbCancel Then Exit Sub
        retStr = scpi.Parse("SENS:CORR:COLL:GUID:ACQ STAN" & CStr(i) & ";*OPC?")
    Next

    ' Compute the error coefficients and save the cal to Global Delta Match CalSet
    scpi.Parse "SENS:CORR:COLL:GUID:SAVE"
    MsgBox "Cal is done!"
End Sub
```
Perform a Guided 1-Port Cal on Port 2

This VBScript program does the following:

1. Clear measurements from the PNA
2. Create a new S22 measurement
3. Set an instrument state
4. Select the connector types
5. Select a cal kit
6. Initiate a Guided calibration
7. Display a prompt to connect each standard
8. Save the calibration to a newly created cal set

Note: This example illustrates an important step when calibrating a reflection measurement in the reverse direction. You MUST create a reverse (S22) measurement and have it be the active (selected) measurement on the channel that is being calibrated. This is not necessary for any calibrating any other measurement parameter.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Guided.vbs. Learn how to setup and run the macro.

```vbs
Dim App
Set App = CreateObject("AgilentPNA835x.Application")
App.Preset

Dim step
Dim Parser
Dim prompt
Dim txtDat
Dim Chan

' Rem Clear old measurements
App.Reset

' Rem Create a new Measurement
Set Parser = App.SCPIStringParser
Parser.Parse "DISPlay:WINDow1:STATE ON"
Parser.Parse "CALCulate:PARameter:DEFine:EXT 'MyMeas',S22"
Parser.Parse "DISPlay:WINDow1:TRACe1:FEED 'MyMeas'"

' Rem Initialize state
Set Chan = App.ActiveChannel
Chan.StartFrequency = 200e6
```
Chan.StopFrequency = 1.5e9
Chan.IFBandwidth = 1000
step = 3

Rem Begin a guided calibration
Parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'Not used'"
Parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'Type N (50) male'"
Parser.Parse "SENS:CORR:COLL:GUID:INIT"

Rem Query the number of steps

Rem Display the number of steps
MsgBox("Number of steps is " + txtDat)

Rem Set the loop counter limit
step = txtDat

Rem Measure the standards
For i = 1 To step
If i = 1 Then
MsgBox(prompt)
Parser.Parse ("sens:corr:coll:guid:acq STAN1")
ElseIf i = 2 then
MsgBox(prompt)
Parser.Parse ("sens:corr:coll:guid:acq STAN2")
ElseIf i = 3 then
MsgBox(prompt)
Parser.Parse ("sens:corr:coll:guid:acq STAN3")
End If
Next

Rem All standards have been measured. Save the result
Parser.Parse "SENS:CORR:COLL:GUID:SAVE"
MsgBox("The calibration has been completed")
Perform a Guided Calibration using SCPI

This VBScript program performs a Guided Calibration using ECal or Mechanical standards. This example includes optional ECal orientation features.

This example has been updated to include:

- Guided Power Cal (Oct 8, 2010)
- The setting of Unknown Thru or Adapter Removal adapter delay. (March 2006).
- The activation of a channel to be calibrated. (Aug. 2006).

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Guided.vbs. Learn how to setup and run the macro.

```vbs
' Performing a Guided 2-port cal (Ports 1 and 2)
TwoPortGuidedCal
Sub TwoPortGuidedCal
    Dim app
    Dim scpi
    Dim connList
    Dim selectedConn1, selectedConn2
    Dim kitList
    Dim selectedKit
    Dim message
    ' Create / Get the PNA application.
    Set app = CreateObject("AgilentPNA835x.Application")
    Set scpi = app.ScpiStringParser
    
    'The following demonstrates that the Active Channel is cal'd
    'Preset the PNA
    scpi.Execute "SYST:UPR"
    'Create a new measurement on Chan 2
    'Now there are two windows, channels and measurements
    'This becomes the Active Measurement
    scpi.Execute ("DISPlay:WINDow2:STATE ON")
    'Define a measurement name, parameter
```
scpi.Execute("CALCulate2:PARameter:DEFine:EXT 'MyMeas',S21")

"FEED" the measurement

scpi.Execute("DISPlay:WINDow2:TRACe1:FEED 'MyMeas'"")

'This is the Active Measurement

'Activate the 'Preset' measurement to cal chan 1
scpi.Execute("CALC1:PAR:SEL 'CH1_S11_1'"")

' Query the list of connectors that the PNA system recognizes
' Format the list with linefeed characters in place of the commas
connList = FormatList(connList)

message = "Enter your DUT connector for Port 1. Choose from this list:
message = message & Chr(10) & Chr(10) & connList

' Select the connector for Port 1
selectedConn1 = InputBox(message)
If selectedConn1 = "" Then Exit Sub

scpi.Execute("sens:corr:coll:guid:conn:port1 '" & selectedConn1 & "'")

message = "Enter your DUT connector for Port 2. Again, choose from this list:
message = message & Chr(10) & Chr(10) & connList

' Select the connector for Port 2
selectedConn2 = InputBox(message)
If selectedConn2 = "" Then Exit Sub


' Note: If your PNA has more than 2 ports, then uncomment
' one or both of these next two lines.
'scpi.Execute("sens:corr:coll:guid:conn:port3 ""Not used"" "
'scpi.Execute("sens:corr:coll:guid:conn:port4 ""Not used"" "

'This next block of commented code demonstrates how to specify an adapter
' and it's electrical delay, in situations where you are performing an
' Unknown Thru or Adapter Removal calibration. In most situations, the
' PNA is able to correctly determine an adapter's electrical length
' at the end of the calibration. However, there are scenarios where
' the PNA cannot correctly calculate the length -- such as when the channel
' has a relatively small number of measurement points (for example, 201 or less)
' and the adapter is significantly long (for example, a cable that is several
' feet).
' In these cases, the ADAP commands (below) enable you to explicitly specify
the adapter you are using.

Send these commands prior to the "sens:corr:coll:guid:init" command.

Create adapter and return the adapter number

```
"','"& selectedConn2 & ":")
```

The adapterNum string contains a '+ character.

Here we convert to integer to remove that.

```
adapterNum = CStr( CInt(adapterNum) )
```

Specify that this adapter has 10 nanoseconds electrical delay (coaxial).

```
scpi.Execute "sens:corr:coll:guid:adap" & adapterNum & ":del 10E-9"
```

Text description of adapter

```
scpi.Execute "sens:corr:coll:guid:adap" & adapterNum & ":desc 'My adapter'"
```

Select to use this adapter specifically between ports 1 and 2

```
scpi.Execute "sens:corr:coll:guid:adap" & adapterNum & ":path 1,2"
```

End of adapter block

Query the list of acceptable cal kits and

'ECal module characterizations for Port 1.

```
```

Format the list with linefeed

characters in place of the commas

```
kitList = FormatList(kitList)
```

```
message = "Enter your cal kit or ECal module characterization for Port 1. "
message = message & "Choose from this list:" 
message = message & Chr(10) & Chr(10) & kitList 
```

Select the Cal Kit or ECal module

characterization to use for Port 1.

```
selectedKit = InputBox(message)
```

If selectedKit = "" Then Exit Sub

```
scpi.Execute "sens:corr:coll:guid:ckit:port1 " & selectedKit & ":"
```

Query the list of acceptable cal kits

and ECal module characterizations for Port 2.

```
```

Format the list with linefeed characters in place of the commas

```
kitList = FormatList(kitList)
```

message = "Enter your cal kit or ECal module characterization for Port 2. "
message = message & "Choose from this list:"
message = message & Chr(10) & Chr(10) & kitList
' Select the Cal Kit or ECal module
' characterization to use for Port 2.
selectedKit = InputBox(message)
If selectedKit = "" Then Exit Sub
scpi.Execute "sens:corr:coll:guid:ckit:port2" & selectedKit & ""
' This determines whether the cal will be a "Guided Power Cal"
' or just a traditional S-parameter cal.
message = "On which port number shall power be measured?  "
message = message & "For a traditional guided cal without power cal, enter 0"
Dim powerPort
powerPort = CInt( InputBox(message) )
If powerPort > 0 Then
scpi.Execute("sens:corr:coll:guid:psen" & CStr(powerPort) & ":stat on")
Dim retVal
retVal = MsgBox("Is the power sensor's connector type or gender different from
the DUT connector for that port? ", vbYesNo)
If retVal = vbYes Then
message = "Enter your power sensor's connector.  Choose from this list:"
message = message & Chr(10) & Chr(10) & connList
' Select the sensor's connector.
selectedConn1 = InputBox(message)
If selectedConn1 = "" Then Exit Sub
scpi.Execute("sens:corr:coll:guid:psen" & CStr(powerPort) & ":conn " & selectedConn1 & ""
' Query the list of acceptable cal kits and ECal module characterizations
' that are applicable for the sensor's connector.
' Format the list with linefeed
' characters in place of the commas
kitList = FormatList(kitList)
message = "Enter your cal kit or ECal module characterization to use for de-embed
of the sensor's connector.  "
message = message & "Choose from this list:"
message = message & Chr(10) & Chr(10) & kitList
' Select the Cal Kit or ECal module characterization to use for de-embed of the
sensor's connector.

selectedKit = InputBox(message)

If selectedKit = "" Then Exit Sub

scpi.Execute "sens:corr:coll:guid:psen" & CStr(powerPort) & ":ckit '" & selectedKit & "'"

Else

scpi.Execute("sens:corr:coll:guid:psen" & CStr(powerPort) & ":conn 'Ignored'\n"

End If ' End of block that considers the sensor's connector

' Ask for the power level to perform the power cal at
' (if this command is omitted, the default is 0 dBm).

Dim powerLevel

powerLevel = InputBox("Enter the power level for the power cal to be performed at")

If powerLevel = "" Then Exit Sub

scpi.Execute "sens:corr:coll:guid:psen" & CStr(powerPort) & ":pow:lev " & powerLevel

Else


End If ' End of block that considers if the cal will include power calibration

' This next block of commented code
' shows optional functions when using ECal.
' Send these "sens:corr:pref" commands prior to the
' Read ECAL information from ECal module #1 on the USB bus
' about the Agilent factory characterization data
'module1Info = scpi.Execute("sens:corr:coll:ckit:inf? ECAL1,CHAR0")
'MsgBox "Description of ECal Module #1:" & Chr(10) & Chr(10) & module1Info

' The following command enables auto orientation of
' the ECal module (The PNA senses which port of the
' module is connected to which port of the PNA).
'scpi.Execute "sens:corr:pref:ecal:ori ON"

' However, if you are measuring at very low power levels where
' the PNA may fail to sense the module's orientation, then turn auto
' orientation OFF and specify how the module is connected.
' "A1,B2" indicates Port A of the module is connected
' to PNA Port 1 and Port B is connected to PNA Port 2).
'sci.Execute "sens:corr:pref:ecal:ori OFF"
' End of optional ECal setup

'Select the thru method of "Default". This instructs the PNA to
determine which thru standard measurement technique to use
based upon the selected connectors and
calibration kit(s) and the PNA model number.
'with new CMET and TMET 'default' is set by not sending the commands
',

'Initiate the calibration and query the number of steps
sci.Execute "sens:corr:coll:guid:init"
umSteps = sci.Execute("sens:corr:coll:guid:steps?"
MsgBox "Number of steps is " + CStr(numSteps)
' Measure the standards
For i = 1 To numSteps
step = "Step " + CStr(i) + " of " + CStr(numSteps)
MsgBox strPrompt, vbOKOnly, step
sci.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next
' Conclude the calibration
sci.Execute "sens:corr:coll:guid:save"
MsgBox "Cal is done!"
End Sub

Function FormatList(list)
Dim tokens
' Strip the leading and trailing quotation
'marks from the list string
list = Mid(list, 2, Len(list) - 3)
'Tokenize the comma-delimited list string
'into an array of the individual substrings
tokens = Split(list, ",")
'Rebuild the list string, placing linefeed
'characters where the commas were,
' using Trim to remove leading and trailing spaces.
list = ""
For i = 0 To UBound(tokens)
tokens(i) = Trim(tokens(i))
list = list & tokens(i) & Chr(9)
If i < UBound(tokens) Then
  i = i + 1
  tokens(i) = Trim(tokens(i))
  list = list & tokens(i) & Chr(10)
End If
Next
FormatList = list
End Function
Perform Guided ECal using SCPI

This VBScript program performs a Guided ECal Calibration. While this example is good to use as a starting point for Guided ECal, the Guided comprehensive cal example has some advanced features that are not in this program.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Guided.vbs. Learn how to setup and run the macro.

' Performing a 2-port cal (Ports 1 and 2)
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' Specify the DUT connectors
' (for each connector of your DUT, one of the ECal module's ports must have
' that same connector, or else you cannot achieve the cal using that module).
scpi.Execute "sens:corr:coll:guid:conn:port1 ""APC 3.5 female""
scpi.Execute "sens:corr:coll:guid:conn:port2 ""APC 3.5 male""

' Note: If your PNA has more than 2 ports, you would need to uncomment
' one or both of these next two lines, to explicitly specify this is
' just a 2-port cal.
'scpi.Execute "sens:corr:coll:guid:conn:port3 ""Not used""
'scpi.Execute "sens:corr:coll:guid:conn:port4 ""Not used"
MsgBox "Connectors defined for Ports 1 and 2"

' Specify ECal modules
scpi.Parse "sens:corr:coll:guid:ckit:port1 'N4691-60004 ECal'

' Non-factory characterizations are specified as follows:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 User 1 ECal'
' When two or more ECal modules with the same model number are connected
' also specify the serial number as follows:
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:

MsgBox "Cal kits defined for Ports 1 and 2"
' Initiate the calibration and query the number of steps
scpi.Execute "sens:corr:coll:guid:init"
MsgBox "Number of steps is " + CStr(numSteps)

' Measure the standards
For i = 1 To numSteps
    step = "Step " + CStr(i) + " of " + CStr(numSteps)
    MsgBox strPrompt, vbOKOnly, step
    scpi.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next

' Conclude the calibration
scpi.Execute "sens:corr:coll:guid:save"
MsgBox "Cal is done!"
Perform Guided Mechanical Cal using SCPI

This VBScript program performs a Guided Calibration using Mechanical standards. While this example is good to use as a starting point for guided mechanical cal, the Guided comprehensive cal example has some advanced features that are not in this program.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Guided.vbs. Learn how to setup and run the macro.

' Performing a 2-port cal (Ports 1 and 2)
Dim app
Dim scpi

' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser

' Specify the DUT connectors
scpi.Execute "sens:corr:coll:guid:conn:port1 ""APC 3.5 female"" "
scri.Execute "sens:corr:coll:guid:conn:port2 ""APC 3.5 male"" "

' Note: If your PNA has more than 2 ports, you would need to uncomment
' one or both of these next two lines, to explicitly specify this is
' just a 2-port cal.
'scri.Execute "sens:corr:coll:guid:conn:port3 ""Not used"" "
'scri.Execute "sens:corr:coll:guid:conn:port4 ""Not used"" "
MsgBox "Connectors defined for Ports 1 and 2"

' Select the Cal Kit for each port being calibrated.
scpi.Execute "sens:corr:coll:guid:ckit:port1 ""85052D"" "
scpi.Execute "sens:corr:coll:guid:ckit:port2 ""85052D"" "
MsgBox "Cal kits defined for Ports 1 and 2"

' Initiate the calibration and query the number of steps
scpi.Execute "sens:corr:coll:guid:init"
MsgBox "Number of steps is " + CStr(numSteps)

' Measure the standards
' The following series of commands shows that standards
' can be measured in any order. These steps acquire
' measurement of standards in reverse order.
' It is easiest to iterate through standards using
' a For-Next Loop.
For i = numSteps To 1
    step = "Step " + CStr(i) + " of " + CStr(numSteps)
    MsgBox strPrompt, vbOKOnly, step

3051
scpi.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next

' Conclude the calibration
scpi.Execute "sens:corr:coll:guid:save"
MsgBox "Cal is done!"

Last Modified:

20-Jan-2007   Added note about any order for steps.
Perform Guided TRL Calibration

This VBScript file performs a 2-Port Guided TRL calibration on 2-port PNA analyzers. (See an example of TRL cal on a 4-port PNA.) This program does the following:

- Clear old measurements from the PNA
- Create a new S22 measurement
- Set an instrument state
- Select the connectors and cal kit
- Initiate a Guided calibration
- Display a prompt as each new standard must be connected
- Save the calibration to a newly created cal set.

**Note:** This program runs without error on all PNA code revisions 7.21 and higher.

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as TRL.vbs. Learn how to setup and run the macro.

```vbs
Dim App
Dim Parser
Dim Chan
Dim txtDat
Dim step
Dim parserTxt
Dim prompt
Set App = CreateObject("AgilentPNA835x.Application")
' Clear old measurements
App.Reset
' Create a new Measurement
Set Parser = App.SCPIStringParser
Parser.Parse "DISPlay:WINDowl:STATE ON"
Parser.Parse "CALCulate:PARameter:DEFine:EXT 'MyMeas',S12"
Parser.Parse "DISPlay:WINDowl:TRACe1:FEED 'MyMeas'"
' Initialize state
Set Chan = App.ActiveChannel
```

3053
Chan.StartFrequency = 18.0e9
Chan.StopFrequency = 20.0e9
Chan.IFBandwidth = 1000
' Begin a guided calibrations
Parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 male'"
Parser.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 female'"
' Select TRL cal method.
Parser.Parse "SENS:CORR:COLL:GUID:PATH:CMET 1,2,'TRL'"
txtDat = Parser.Parse("SENS:CORR:COLL:GUID:PATH:CMET? 1,2")
MsgBox("Method " + txtDat)
Parser.Parse "SENS:CORR:COLL:GUID:INIT"
' Query the number of steps
' Display the number of steps
MsgBox("Number of steps is " + txtDat)
' Set the loop counter limit
step = CInt(txtDat)
' Measure the standards
For i = 1 To step
parserTxt = "sens:corr:coll:guid:desc? " + CStr(i)
prompt = Parser.Parse(parserTxt)
MsgBox(prompt)
parserTxt = "sens:corr:coll:guid:acq STAN" + CStr(i)
Parser.Parse (parserTxt)
Next
' All standards have been measured.  Save the result
Parser.Parse "SENS:CORR:COLL:GUID:SAVE"
MsgBox("The TRL calibration has been completed")
Perform an Unguided 1-Port Cal on Port 2

This VBScript program does the following:

1. Clear measurements from the PNA
2. Create a new S22 measurement
3. Set an instrument state
4. Select a cal kit
5. Initiate an Unguided calibration
6. Display a prompt to connect each standard
7. Save the calibration to a newly created cal set

**Note:** This example illustrates an important step when calibrating a reflection measurement in the reverse direction. You MUST create a reverse (S22) measurement and have it be the active (selected) measurement on the channel that is being calibrated. This is not necessary for any calibrating any other measurement parameter.

**Note:** The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unguided.vbs. Learn how to setup and run the macro.

```vbs
Dim App
Set App = CreateObject("AgilentPNA835x.Application")
App.Preset

Dim Parser
Dim Chan

Rem Clear old measurements
App.Reset

Rem Create a new Measurement
Set Parser = App.SCPIStringParser
Parser.Parse "DISPLAY:WINDow1:STATE ON"
Parser.Parse "CALCulate:PARameter:DEFine:EXT 'MyMeas',S22"
Parser.Parse "DISPLAY:WINDow1:TRACe1:FEED 'MyMeas'"

Rem Initialize state
Set Chan = App.ActiveChannel
Chan.StartFrequency = 200e6
Chan.StopFrequency = 1.5e9
Chan.IFBandwidth = 1000

Rem Begin an unguided calibration
```
Rem Set the calibration method
Parser.Parse "SENSe:CORRection:COLLect:METHod REFL3"

Rem Turn off continuous sweep
Parser.Parse "INITiate:CONTinuous OFF"

Rem Select a cal kit
Parser.Parse "SENSe:CORRection:COLLect:CKIT:SELect 1"

Rem Measure the standards
MsgBox("Connect OPEN to port 2. Then press OK")
Parser.Parse ("sens:corr:coll:acq STAN1")

MsgBox("Connect SHORT to port 2. Then press OK")
Parser.Parse ("sens:corr:coll:acq STAN2")

MsgBox("Connect LOAD to port 2. Then press OK")
Parser.Parse ("sens:corr:coll:acq STAN3")

Rem All standards have been measured. Save the result
Parser.Parse "SENS:CORR:COLL:SAVE"

Rem Turn ON continuous sweep
Parser.Parse "INITiate:CONTinuous ON"
MsgBox("The calibration has been completed")
Perform an Unguided 2-Port Mechanical Cal

This VBScript program performs an Unguided, Full 2-Port, calibration using ONE set of mechanical calibration standards.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unguided.vbs. Learn how to setup and run the macro.

```vbs
Set App = CreateObject("AgilentPNA835x.Application")
Set Scpi = App.SCPIStringParser

'Initialize state
Scpi.Execute ("SYSTem:PRESet")

'Select the Preset measurement
Scpi.Execute ("CALCulate:PARameter:SELect 'CH1_S11_1'")

'Set the calibration method
Scpi.Execute ("SENSe:CORRection:COLLect:METHod SPARSOLT")

'Select a cal kit
Scpi.Execute ("SENSe:CORRection:COLLect:CKIT:SELect 1")

'Set one set of standards
Scpi.Execute ("SENSe:CORRection:TSTandards OFF")

'Set acquisition to FORWARD
Scpi.Execute ("SENSe:CORRection:SFORward ON")

'Measure the standards in forward direction
MsgBox "Connect OPEN to Port 1; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan1")

MsgBox "Connect SHORT to Port 1; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan2")

MsgBox "Connect LOAD to Port 1; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan3")

'Set acquisition to REVERSE
Scpi.Execute ("SENSe:CORRection:SFORward OFF")

'Measure the standards in reverse direction
MsgBox "Connect OPEN to Port 2; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan1")

MsgBox "Connect SHORT to Port 2; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan2")
```

3057
MsgBox "Connect LOAD to Port 2; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan3")

'Measure the thru standard
MsgBox "Connect THRU between Ports 1 and 2; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan4")

'OPTIONAL Measure Isolation
MsgBox "Connect LOADS to Port 1 AND Port 2; then press OK"
Scpi.Execute ("SENSe:CORRection:COLLect:ACQuire stan5")

'All standards have been measured. Save the result
Scpi.Execute ("SENS:CORR:COLL:SAVE")
MsgBox "The calibration has been completed"
Perform an Unguided ECal

This VBScript program performs an Unguided Full 2-Port ECal.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unguided.vbs. Learn how to setup and run the macro.

See Sense:Correction commands.

See other SCPI Examples

Set pna = CreateObject("AgilentPNA835x.Application")
Set scpi = pna.ScpiStringParser
'
Preset the analyzer
scpi.Execute "SYSTem:PRESet"
'
Start frequency of 10 MHz
scpi.Execute "SENSe:FREQuency:STARt 10E6"
'
Stop frequency of 9 GHz
scpi.Execute "SENSe:FREQuency:STOP 9E9"
'
Select the preset S11 measurement
scpi.Execute "CALCulate:PARameter:SELect 'CH1_S11_1'"
'
Read the information about the Agilent factory
characterization data of ECal module #1 on the USB bus
module1Info = scpi.Execute("SENSe:CORRection:COLLect:CKIT:INFormation? ECAL1,CHAR0")
'
Prompt for the ECal module
MsgBox "Description of ECal Module #1:" & Chr(10) & Chr(10) & module1Info & _Chr(10) & Chr(10) & "Make port connections to the ECal module, then press enter"
'
ECal full 1 port and 2 port
Choose a Calibration Type (comment out one of these)
scpi.Execute "SENSe:CORRection:COLLect:METHod refl3"
scpi.Execute "SENSe:CORRection:COLLect:METHod SPARSOLT"
'
Specify to have the PNA automatically determine which port of the ECal module is connected to which port of the PNA.
scpi.Execute "SENSe:CORRection:PREFerence:ECAL:ORIentation ON"
'
Alternatively, if you are measuring at very low power levels where the PNA fails to sense the module's orientation, you may need to turn off the auto orientation and specify how the module is connected (as in these next two commented lines of code -- "A1,B2" would indicate Port A of the module is connected to Port 1 and Port B is connected to Port 2).
scpi.Execute "SENSe:CORRection:PREFerence:ECAL:ORIentation OFF"
scpi.Execute "SENSe:CORRection:PREFerence:ECAL:PMAP ECAL1,'A1,B2'"
'
Acquire and store the calibration terms. *OPC? causes a "+1" to be
' returned when finished. CHAR0 indicates to use the Agilent factory
characterized data within the ECAL module (as opposed to a user characterization).

x = scpi.Execute("SENSe:CORRection:COLLect:ACQuire ECAL1,CHAR0;*OPC?"")
' Note: if you have set up a slow sweep speed (for example, if
you’re using a narrow IF bandwidth), and while this calibration is
being acquired you wish to have your program perform other operations
(like checking for the click event of a Cancel button) and you’re
NOT using the COM ScpiStringParser, you can use the optional
ASYNchronous argument with the ACQuire command as shown here below
instead of sending that command in the way shown above. The SCPI
parser then will return immediately while the cal acquisition
proceeds (i.e., the parser will NOT block-and-wait for the
cal to finish, so you can send additional commands in the meantime).
So you can do "*ESR?" or "*STB?" queries to monitor the status register
bytes to see when the OPC bit gets set, which indicates the cal has
finished. That type of OPC detection works for all of the PNA’s SCPI
parsers except the COM ScpiStringParser.
An alternative to querying the status register, is to setup an SRQ handler
if your IO Libraries supports that.
When an SRQ event occurs, a call back will automatically
"SENSe:CORRection:COLLect:ACQuire ECAL1,CHAR0,ASYNchronous;*OPC"
MsgBox "Done with calibration."
Perform Unknown Thru or TRL Cal

The following program performs either a 2-port SOLT Unknown Thru Cal or a 2-port TRL Cal. The 85052C Cal Kit used in this program contains both types of standards. This program can be run on 2-port or 4-port PNA's. When run on a multiport (4 or more ports) PNA, which does not have a reference receiver per port, a Delta Match Cal is required.

See example of Delta Match Cal.
See the Guided Cal commands

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unknown.vbs. Learn how to setup and run the macro.

```vbs
Sub PerformUnknownThruOrTRLCal()
    Set pna = CreateObject("AgilentPNA835x.Application")
    Set scpi = pna.ScpiStringParser

    ' Specify connectors for Ports 1 and 2
    scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT1 'APC 3.5 female'
    scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT2 'APC 3.5 male'

    ' If your PNA has 3 or 4 ports, uncomment one or both of these next two lines, to explicitly specify this is a 2-port cal.
    ' scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT3 'Not used'
    ' scpi.Parse "SENS:CORR:COLL:GUID:CONN:PORT4 'Not used'

    ' Specify cal kit for Ports 1 and 2
    scpi.Parse "SENS:CORR:COLL:GUID:CKIT:PORT1 '85052C'

    ' Since the 85052C cal kit contains SOLT standards and also TRL standards, these next two lines set cal and thru method
    scpi.Execute "SENS:CORR:COLL:GUID:PATH:CMEThod 1,2,"SOLT"
    scpi.Execute "SENS:CORR:COLL:GUID:PATH:TMEThod 1,2,"UNKN"

    ' To set up the cal as TRL, comment the previous line and uncomment this next line. The TMEThod is set by default
    scpi.Execute "SENS:CORR:COLL:GUID:PATH:CMEThod 1,2,"TRL"

    ' Initiate the calibration
    scpi.Parse "SENS:CORR:COLL:GUID:INIT"

    ' Query the list of ports that need delta match
    portList = Split(retStr, ",")

    ' If portList contains just one element and it's value is 0, then that indicates none of the ports being calibrated require delta match data.
    ' Note: if each testport on the PNA has it's own reference receiver (R channel), then delta match is never needed, so portList will always be just 0.
    lowerBound = LBound(portList)
    If (UBound(portList) <> lowerBound) Or (CInt( portList(lowerBound) ) <> 0) Then
```

---

3061
Delta match data is required for at least one port.
For this example, we assume a Global Delta Match Cal has previously been
performed so the Global Delta Match CalSet exists.
The Global Delta Match CalSet is used when the APPL command is invoked
without a specific calset ID (GUID).
scpi.Parse "SENS:CORR:COLL:GUID:DMAT:APPL"
End If

' Query the number of calibration steps
retStr = scpi.Parse("SENS:CORR:COLL:GUID:STEP?")
numSteps = CInt(retStr)

' Measure the cal standards
For i = 1 To numSteps
retVal = MsgBox(prompt, vbOKCancel)
If retVal = vbCancel Then Exit Sub
retStr = scpi.Parse("SENS:CORR:COLL:GUID:ACQ STAN" & CStr(i) & ";*OPC?")
Next

' Compute the error coefficients and save the cal to CalSet, and turn it on
scpi.Parse "SENS:CORR:COLL:GUID:SAVE"
MsgBox "Cal is done!"
End Sub

Last Modified:

14-May-2007  MX Updated for new CMET and TMET commands
Setup Fast CW and FIFO

This example program does the following:

- Setup an A/R and B/R measurement
- Turn ON point averaging
- Set external edge triggering (commented out)
- Set FIFO and Fast CW
- Write data into FIFO data buffer
- Read FIFO data buffer

**IMPORTANT** - Because the IFBW is set to 600 kHz, data will NOT be sent to the FIFO after each acquisition. [Learn more.]

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as FIFO.vbs. [Learn how to setup and run the macro.]

[See the SCPI FIFO commands.]

---

**See Other SCPI Example Programs**

```vbs
Dim returnStr
Dim app
Dim p
Dim start
Dim complete
Dim init
Dim finished
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set p = app.ScpiStringParser
sub Write (command)
    if len(returnStr) <> 0 then
        err.Raise 55,"Write","Query Unterminated"
    end if
    returnStr = p.parse(command)
end sub
```
end sub

sub WriteIgnoreError(command)
    returnStr = p.Execute(command)
p.Parse("SYST:ERR?") ' clear error queue
end sub

function Read
    if len(returnStr) = 0 then
        err.Raise 55, "Read", "Bad read"
    end if
    Read = returnStr
    returnStr = ""
end function

' Setup and measure A/R and B/R
Write "SYST:FPRESET"
Write "DISP:WIND ON"
Write "CALC:PAR:DEF:EXT 'meas1','A/R1,0'"
Write "DISP:WIND:TRACE:FEED 'meas1'"
Write "CALC:PAR:DEF:EXT 'meas2','B/R1,0'"
Write "DISP:WIND:TRACE2:FEED 'meas2'"

' Set IFBW to 600 khz (400 thousand pts/second)
Write "SENS:BWID:RES 600khz"

' Point Averaging Count = 1
Write "SENS:AVER:MODE POINT"
Write "SENS:AVER ON"
Write "SENS:AVER:COUNT 1"

' Edge triggering - positive edge
'Write "CONT:SIGN BNC1,TIEPOSITIVE"
'Write "TRIG:SOUR EXT"
'Write "SENS:SWE:TRIG:POIN ON"

' Setup FIFO and Fast CW count
Write "SENS:SWE:MODE HOLD"
Write "SYST:FIFO ON"
Write "SYST:FIFO:DATA:CLEAR"
Write "SENS:SWE:TYPE CW"
Write "SENS:SWE:TYPE:FACW 1000000"' set the point count to 1 million
Write "SENS:SWE:MODE SING" ' start an asynchronous acquisition.
init = now()
' Gather data
'wait until end of sweep. Timeout needs to be very large here.
Write "*OPC?" '
opcCount = Read()
Dim points
Write "SYST:FIFO:DATA:COUNT?"
points = Read()
msgbox points
' points == 2000000 ' points = 2million. Took 5 seconds to acquire
For I = 0 to 1 ' 2 iterations (2 parameters * 2 sets of 1 million)
Dim data
Write "SYST:FIFO:DATA? 1000000"
Data = Read()
Next
'turn FIFO and FastCW OFF
Write "SYST:FIFO OFF"
Write "SENS:SWE:TYPE:FACW 0"

finished = now()
msgbox "Init =" & init & vbCrLf & "Done =" & finished
Setup Markers using SCPI

This VBScript program does the following:

- Preset the PNA
- Return active channel number and measurement string
- Create a marker
- Set X-axis value
- Read X, Y-axis values
- Set marker to trace Min
- Read X, Y-axis values

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Markers.vbs. Learn how to setup and run the macro.

See all Marker SCPI commands.

---

Dim na, vi, ret
Set na = CreateObject("AgilentPNA835x.Application")
Set vi = na.ScpiStringParser
'Get Identification String from Analyzer
ret=vi.Parse("*IDN?")
msgbox ret
'Preset PNA
ret=vi.Parse("SYST:PRES; *OPC?")
'Get Active Channel and Measurement
chan = vi.Parse("SYST:ACT:CHAN?")
meas = vi.Parse("SYST:ACT:MEAS?")
'Convert chan to a single number
chan=CStr(CInt(chan))
'Select Active Measurement
vi.Parse "CALC" + chan + ":PAR:SEL " + meas
'Turn Marker 1 on and set X value to 1 GHz
vi.Parse "CALC" + chan + ":MARK1:STAT ON"
vi.Parse "CALC" + chan + ":MARK1:X 1e9"
'Get X and Y marker values
x_val = vi.Parse("CALC" + chan + ":MARK1:X?")
y_val = vi.Parse("CALC" + chan + ":MARK1:Y?")
'Display Marker Values
msgbox "X Value = " + x_val + Chr(10) + "Y Value = " + y_val
'Use Marker 1 as a minimum search
vi.Parse "CALC" + chan + ":MARK1:FUNC:EXEC MIN"
'Get X and Y marker values
x_val = vi.Parse("CALC" + chan + ":MARK1:X?")
y_val = vi.Parse("CALC" + chan + ":MARK1:Y?")
'Display Marker Values
msgbox "X Value = " + x_val + Chr(10) + "Y Value = " + y_val
Setup Noise Figure Port Mapping

This program demonstrates how to change source and receive ports when measuring noise figure. It assumes that option 029 ("Fully Corrected Noise Figure") is installed.

If only option 028 ("Noise figure measurements using standard receivers") is installed, switching ports is simpler, since only one noise receiver selection is available.

This program can be run as a macro in the PNA. To do this, copy the code into a text editor file such as Notepad and save on the PNA hard drive as NF.vbs. Learn how to setup and run the macro.

See the Noise figure commands.

option explicit
dim app, hostname, parser
set app = CreateObject("Agilentpna835x.application")
set parser = app.ScpiStringParser
' Create Noise Figure measurement
parser.Parse "*RST"
parser.Parse "CALC:PAR:DEL:ALL"
parser.Parse "CALC:CUST:DEF 'NF', 'Noise Figure Cold Source', 'NF' "
parser.Parse "DISP:WIND:TRAC1:FEED 'NF'"
' To change from the default input/output port settings of
' source port = PNA1, receive port = PNA2, you must first
' change the noise receiver, then select the desired ports.
dim srcPort, rcvPort
' Set source=PNA port 3 and receiver=PNA port 4
srcPort = 3
rcvPort = 4
' use PNA receiver for noise measurements
parser.Parse "SENS:NOIS:REC NORMAL"
' set port mapping
parser.Parse "SENS:NOIS:PMAP " & srcPort & "," & rcvPort
' To revert back to using the noise receiver, the source
' and receive ports must be set to their default values
' BEFORE switching to the noise receiver. Otherwise, a
' SCPI "Execution error" will occur.
Setup Phase Control

This VBScript program configures and displays Phase Sweep measurements.
The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as RxLev.vbs. Learn how to setup and run the macro.

See Also
Phase Control SCPI commands
About Phase Control

See Other SCPI Example Programs

```
'start
Set pna = CreateObject("AgilentPNA835x.Application")
Set SCPI = PNA.ScpiStringParser
'start
'Assume port 1 is connected to port 3
Set pna = CreateObject("AgilentPNA835x.Application")
Set SCPI = PNA.ScpiStringParser
'Create 3 trace S33, R3/C(amp),R3/C(phase)
SCPI.Parse("SYST:FPR")
SCPI.Parse("DISP:WIND:STATE ON")
SCPI.Parse("CALC:PAR:DEF 'MyMeas1',S33")
SCPI.Parse("DISP:WIND1:TRAC1:FEED 'MyMeas1'")
SCPI.Parse("CALC:PAR:SEL 'MyMeas1'")
SCPI.Parse("CALC:FORM SMIT")
SCPI.Parse("CALC:PAR:DEF 'MyMeas2',R3C,3")
SCPI.Parse("DISP:WIND1:TRAC2:FEED 'MyMeas2'")
SCPI.Parse("CALC:PAR:SEL 'MyMeas2'")
SCPI.Parse("CALC:FORM MLOG")
SCPI.Parse("CALC:PAR:DEF 'MyMeas3',R3C,3")
SCPI.Parse("DISP:WIND1:TRAC3:FEED 'MyMeas3'")
SCPI.Parse("CALC:PAR:SEL 'MyMeas3'")
SCPI.Parse("CALC:FORM PHAS")
SCPI.Parse("SENS:SWE:TYPE PHAS")
'turn on 3 and 1
SCPI.Parse("SOUR:POW1:MODE ON")
SCPI.Parse("SOUR:POW3:MODE ON")
```

'set port3's control parameter to R3/C
SCPI.Parse("SOUR:PHAS3:PAR 'R3/C'")

'Set port3 to PAR mode
SCPI.Parse("SOUR:PHAS3:PAR:MODE PAR")
SCPI.Parse("SOUR:PHAS3:PAR:PORT 1")
SCPI.Parse("SOUR:PHAS3:POFF:FIX 3")
SCPI.Parse("SOUR:PHAS3:STAR 0")
SCPI.Parse("SOUR:PHAS3:STOP 180")
Setup PNOP and PSAT Marker Search

This example program does the following:

- Sets up measurement for either PNOP or PSAT marker search
- Sets parameters for search
- Reads a parameter for each

See PNOP and PSAT SCPI commands.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as SearchMkr.vbs. Learn how to setup and run the macro.

See Other SCPI Example Programs

Dim app
Set app = CreateObject("AgilentPNA835X.Application")
Dim scpi
set scpi = app.ScpiStringParser
scpi.Execute("SYST:FPReset")
' View Power Out vs Power In
' Create and turn on window/channel 1
scpi.Execute("DISPlay:WINDow1:STATE ON")
'Define a measurement name, parameter
scpi.Execute("CALCulate1:PARameter:DEFine:EXT 'MyMeas',B")
'Associate ("FEED") the measurement name ("MyMeas") to WINDow (1)
scpi.Execute("DISPlay:WINDow1:TRACe1:FEED 'MyMeas'")
scpi.Execute("CALCulate1:PARameter:SELection 'MyMeas'")
'perform power sweep
scpi.Execute("SENSe1:SWEep:TYPE POWer")
scpi.Execute("SOURce1:POWer:STARt -5")
scpi.Execute("SOURce1:POWer:STOP 0")
'-------------
'Choose marker search
resp=Msgbox("PNOP (yes) or PSAT (no)" , 4, "PNA Marker Search Demo")
if resp=6 then
    PNOP1()
else
    PSAT1()
End If

'--------------------
'PSAT marker search
Sub PSAT1()
    scpi.Execute("CALCulate1:MARKer:PSATuration:BACKoff 2")
    'Read PSAT Parameter
    dim answer
    answer=scpi.Execute("CALCulate1:MARKer:PSATuration:GAIN?")
    wscript.echo("Gain Sat: ", answer)
End Sub

'--------------------
'PNOP marker search
Sub PNOP1()
    scpi.Execute("CALCulate1:MARKer:PNOP:BACKoff 2")
    scpi.Execute("CALCulate1:MARKer:PNOP:POFFset 1")
    'Read PNOP Parameter
    dim answer
    answer=scpi.Execute("CALCulate1:MARKer:PNOP:GAIN?")
    wscript.echo("PNOP Gain: ", answer)
End Sub
Setup Receiver Leveling using SCPI

This VBScript program configures Receiver Leveling.

- Preset the PNA
- Make all receiver leveling settings

The SCPI commands in this example are sent over a COM interface using the `SCPIStringParser` object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as RxLev.vbs. Learn how to setup and run the macro.

See all Receiver Leveling SCPI commands.

See Other SCPI Example Programs

```vbs
Set pna = CreateObject("AgilentPNA835x.Application")
Set SCPI = pna.ScpiStringParser
'set source port
dim srcP
srcP = "1"
'Preset PNA
SCPI.Parse "SYST:PRES"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:ref 'R1'"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:tol 0.02"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:iter 10"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:fast OFF"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:ifbw 100"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:offs 0"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:safe:max 20"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:safe:max -100"
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec:safe ON"
'Last, enable receiver leveling
SCPI.Parse "sour1:pow" + srcP + "::alc:mode:rec ON"
```

Last Modified:

13-Feb-2009     MX New topic
Setup Sweep Parameters using SCPI

This Visual Basic program sets up sweep parameters on the Channel 1 measurement. To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

```vbnet
GPIB.Write "SYSTem:PRESet"
'
Select the measurement
GPIB.Write "CALCulate:PARameter:SELection 'CH1_S11_1'"
' Set sweep type to linear
GPIB.Write "SENSe1:SWEep:TYPE LIN"

' Set IF Bandwidth to 700 Hz
GPIB.Write "SENSe1:BANDwidth 700"

' Set Center and Span Freq's to 4 GHz
GPIB.Write "SENSe1:FREQuency:CENTer 4ghz"
GPIB.Write "SENSe1:FREQuency:SPAN 4ghz"

' Set number of points to 801
GPIB.Write "SENSe1:SWEep:POINts 801"

' Set sweep generation mode to Analog
GPIB.Write "SENSe1:SWEep:GENeration ANAL"

' Set sweep time to Automatic
GPIB.Write "SENSe1:SWEep:TIME:AUTO ON"

' Query the sweep time
GPIB.Write "SENSe1:SWEep:TIME?"
SweepTime = GPIB.Read
```
Setup the Display using SCPI

This Visual Basic program:

- Sets data formatting
- Turns ON the Trace, Title, and Frequency Annotation
- Autoscales the Trace
- Queries Per Division, Reference Level, and Reference Position
- Turn ON and set averaging
- Turn ON and set smoothing

To run this program, you need:

- An established GPIB interface connection

See Other SCPI Example Programs

GPIB.Write "SYSTem:PRESet"

'Select the measurement
GPIB.Write "CALCulate:PARameter:SELect 'CH1_S11_1'"

'Set the Data Format to Log Mag
GPIB.Write ":CALCulate1:FORMat MLOG"

'Turn ON the Trace, Title, and Frequency Annotation
GPIB.Write "DISPlay:WINDow1:TRACe1:STATe ON"
GPIB.Write "DISPlay:WINDow1:TITle:STATe ON"
GPIB.Write "DISPlay:ANNotation:FREQuency ON"

'Autoscale the Trace
GPIB.Write "DISPlay:WINDow1:TRACe1:Y:Scale:AUTO"

'Query back the Per Division, Reference Level, and Reference Position
GPIB.Write "DISPlay:WINDow1:TRACe1:Y:SCALe:PDIVision?"
Pdiv = GPIB.Read
GPIB.Write "DISPlay:WINDow1:TRACe1:Y:SCALe:RLEVel?"
Rlev = GPIB.Read
GPIB.Write "DISPlay:WINDow1:TRACe1:Y:SCALe:RPOSition?"
Ppos = GPIB.Read

'Turn ON, and average five sweeps
GPIB.Write "SENSe1:AVERage:STATe ON"
GPIB.Write "SENSe1:AVERage:Count 5"

'Turn ON, and set 20% smoothing aperture
GPIB.Write "CALCulate1:SMOothing:STATe ON"
GPIB.Write "CALCulate1:SMothing:APERture 20"
Show Custom Cal Windows during a Guided Calibration

This VBScript program shows how to send commands that allow you to view specific 'custom' windows, and sweep specific channels, during a UI (Cal Wizard) or remote calibration.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as CalWindow.vbs. Learn how to setup and run the macro.

These commands are used to show and sweep windows and channels:

- `SENS:CORR:COLL:DISP:WIND`
- `SENS:CORR:COLL:SWE:CHAN`
- `SENS:CORR:COLL:DISP:WIND:AOFF`
- `SENS:CORR:COLL:SWE:CHAN:AOFF`
- `SENS:CORR:COLL:GUID:PACQuire`

```vbs
Dim app
Dim scpi
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
' A comment
'Preset the analyzer
'This creates an S11 measurement in channel 1, window 1
scpi.Execute "SYST:PReset"
' Create and turn on window 2
scpi.Execute "DISPlay:WINDow2:STATE ON"
'Define an S21 measurement in channel 2
scpi.Execute "CALCulate2:PARameter:DEFine:EXT 'MyMeas',S21"
'Associate ("FEED") the measurement name ('MyMeas') to WINDow2
'and give the new TRACe a number (1).
scpi.Execute "DISPlay:WINDow2:TRACe1:FEED 'MyMeas'"
```

See Other SCPI Example Programs
'The following lines are all you need in order to:
'show and sweep the custom Cal windows during a UI Calibration
'If sending ONLY these commands, make sure you know the
'correct window and channel numbers to show and sweep.
'Flag windows 1 and 2 to show during Ch1 calibration
scpi.Execute "SENS:CORR:COLL:DISP:WIND1 ON"
scpi.Execute "SENS:CORR:COLL:DISP:WIND2 ON"
'Flag channels 1 and 2 to sweep during Ch1 calibration
scpi.Execute "SENS1:CORR:COLL:SWE:CHAN1 ON"
scpi.Execute "SENS1:CORR:COLL:SWE:CHAN2 ON"

' The following code performs a remote guided Cal on Ch1.
' From a remote cal, the Cal window does not normally show and sweep
' after the previous standard has been acquired.
' This shows how to include the PACquire (preview) to view and sweep the Cal Window.
' The Custom window also shows and sweeps due to the flag commands above.
' The flags are cleared at the end of this section.

'Specify the DUT connectors
scpi.Execute "sens:corr:coll:guid:conn:port1 "APC 3.5 female"" 
scpi.Execute "sens:corr:coll:guid:conn:port2 "APC 3.5 male"" 
'Select the Cal Kit for each port being calibrated.
scpi.Execute "sens:corr:coll:guid:ckit:port1 "85052D"" 
scpi.Execute "sens:corr:coll:guid:ckit:port2 "85052D"" 
'Initiate the calibration and query the number of steps
scpi.Execute "sens:corr:coll:guid:init"
MsgBox "Number of steps is " + CStr(numSteps)
'Measure the standards
For i = 1 to numSteps
step = "Step " + CStr(i) + " of " + CStr(numSteps)
'send the Preview Acquire command, then prompt
scpi.Execute "sens:corr:coll:guid:PACquire STAN" + CStr(i)
' Do NOT send any Guided Cal commands here or the cal window will not sweep

MsgBox strPrompt, vbOKOnly, step
scpi.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next
' Conclude the calibration
scpi.Execute "sens:corr:coll:guid:save"
MsgBox "Cal is done!"

' Remove the Custom Window flags
scpi.Execute "SENS:CORR:COLL:DISP:WIND:AOFF"
' Remove the channel sweep flags
Perform a Sliding Load Calibration using GPIB

This Visual Basic program does a **only** the sliding load portion of a Calibration.
To run this program, you need:

- An established **GPIB interface connection**
- A measurement and calibration routine to call this sub-program
- STAN3 set up as a sliding load standard

See Other SCPI Example Programs

Sub slide()
'Measure the sliding load for at least 5 and no more than 7 slides
'Note that "SLSET" and "SLDONE" must be executed before the actual acquisition of a slide
MsgBox "Connect Sliding Load; set to Position 1; then press OK"
GPIB.Write "SENS:CORR:COLL SLSET"
GPIB.Write "SENS:CORR:COLL STAN3;"

MsgBox "Set Sliding Load to position 2; then press OK"
GPIB.Write "SENS:CORR:COLL SLSET"
GPIB.Write "SENS:CORR:COLL STAN3;"

MsgBox "Set Sliding Load to position 3; then press OK"
GPIB.Write "SENS:CORR:COLL SLDONE"
GPIB.Write "SENS:CORR:COLL STAN3;"
End Sub
Socket Client

The following C# example demonstrates how to send SCPI commands to the PNA via a TCP socket connection and how to use a TCP 'control' connection. If the command is a query, the program will read the instrument's response. You can add or replace the SCPI commands in this program with your own.

Learn how to enable Sockets communication on the PNA.

For both of the following methods, first copy the example text below into a Notepad file and name it SocketClient.cs.

To run using Microsoft Visual Studio 2003 or 2005

1. From the Visual Studio File menu, select New, then Project.
2. In the New Project window, select the following items (noting the location of the file folder it is creating for you) then click OK.

- Project Type: Visual C#
- Template: Console Application
- Project Name: SocketClient

1. Copy SocketClient.cs into the folder that was created in the previous step.
2. In the Solution Explorer window pane, right-click Class1.cs (if Visual Studio 2003) or Program.cs (if Visual Studio 2005). Select Delete to delete that file.
3. In the Solution Explorer, right-click SocketClient, and select Add, then Existing Item....
4. Browse to select SocketClient.cs and click OK.

You should then be able to build the project, and test the resulting SocketClient.exe from a command prompt (shell) window.

To run using Mono

Mono is a cross-platform version of .NET. You can download a free version of Mono at http://www.mono-project.com. Once downloaded and installed:

1. Run the Mono command prompt (shell) window.
2. Navigate to the directory where the example SocketClient.cs is stored.
3. Type: MCS SocketClient.cs (builds the .exe and saves in that same folder.)
4. Type mono SocketClient.exe <PNA name or IP address>

This example was compiled and tested successfully with Mono version 1.1.13. It was run on a PC using the Red
Hat version 9.0 distribution of the Linux operating system. It was also run on a PC using Windows XP. This program has not been tested with other versions of Mono, or on other operating systems.

**To run with Agilent T&M Toolkit**

Agilent T&M Toolkit 2.0 is the first version to support communication using Sockets.

Use the following to address the Sockets port: `TCPIP0::<PNA name or IP address>::5025::SOCKET`

```csharp
using System;
using System.Net;
using System.Net.Sockets;

// This C# "Console Application" example program demonstrates SCPI
// communication with an Agilent TCP socket-enabled instrument that
// supports socket "control connections" (such as PNA network analyzers,
// which have support for control connections in their socket
// implementation as of PNA Firmware A.08.33.01).
namespace CSharpSocketClient
{
    /// <summary>
    /// The class supporting the main entry point for the application.
    /// </summary>
    /// <summary>
    class MainClass
    {
        static AsyncCallback m_pCallbackFunc;
        static string m_AsyncReply;

        /// <summary>
        /// The main entry point for the application.
        /// </summary>
        /// <summary>
        [STAThread]
        static int Main(string[] args)
        {
            try
            {
                if (args.Length != 1)
                {
                    Console.WriteLine("\n");
                }
            }
        }
    }
}
```
Console.WriteLine("Usage -- with Microsoft's .NET runtime:");
Console.WriteLine("SocketClient servernameoraddress");
Console.WriteLine("Example: SocketClient 192.168.0.1");
Console.WriteLine("\n");
Console.WriteLine("Usage -- with Mono's (www.mono-project.com) .NET runtime:");
    Console.WriteLine("mono SocketClient.exe servernameoraddress");
    Console.WriteLine("Example: mono SocketClient.exe 192.168.0.1");
    return 1;
}

string server = args[0];
Int32 port = 5025; // default socket port number for the PNA

// Create the primary client socket instance
    Socket client = new Socket(AddressFamily.InterNetwork, SocketType.Stream,
                              ProtocolType.Tcp);

// Get the DNS IP addresses associated with the instrument.
    // (if 'server' string contains the IP address rather than DNS name, this still works)
    IPHostEntry hostInfo = Dns.Resolve(server);
    IPAddress[] IPaddresses = hostInfo.AddressList;
    if (IPaddresses.GetLength(0) < 1)
        return 1;

// Create an endpoint to use for opening the socket connection
    IPEndPoint endpoint1 = new IPEndPoint(IPaddresses[0], port);
// Open the connection to the server instrument
    client.Connect(endpoint1);
    if(!client.Connected)
        return 1;

// Query the instrument's ID string.
    string id = Parse(client, "*IDN?");

// Clear the instrument's Status Byte
Parse(client, "*CLS");

// Enable for the OPC bit (bit 0, which has weight 1) in the instrument's
// Event Status Register, so that when that bit's value transitions from 0
to 1
// then the Event Status Register bit in the Status Byte (bit 5 of that
byte)
// will become set.
Parse(client, "*ESE 1");

// Enable for bit 5 (which has weight 32) in the Status Byte to generate an
// SRQ when that bit's value transitions from 0 to 1.
Parse(client, "*SRE 32");

// Ask the instrument for the number of a port on which a 'control'
// socket connection can be opened.
string controlPortNumStr = Parse(client, "SYSTem:COMMunicate:TCPip:CONTrol?
");
Int32 controlPortNum = System.Convert.ToInt32(controlPortNumStr);

// Create the client "control connection" socket instance
Socket controlClient = new Socket(AddressFamily.InterNetwork,
SocketType.Stream, ProtocolType.Tcp);

// Create an endpoint to use for opening the control connection
IPEndPoint endpoint2 = new IPEndPoint(IPaddresses[0], controlPortNum);
// Connect to the server instrument via the port number that was returned
by the instrument.
controlClient.Connect(endpoint2);
if(!controlClient.Connected)
    return 1;

// Start the control connection listening for an SRQ message.
BeginListeningForAsyncReply(controlClient);

// Now send a preset command to the instrument, accompanied by '*OPC' such
// that when that operation is complete an SRQ event will be generated
// which posts the Status Byte message on the control connection.
Parse(client, "SYSTem:PRESet;*OPC");

    // Normally at this point you would want to have your program do other things
    // right here until the SRQ callback occurs, instead of just idling here waiting
    // for it.
    do {} while (m_AsyncReply == null);

    // Now that the SRQ has occurred, we can issue a Device Clear via the control connection.
    Parse(controlClient, "DCL");

    // The instrument will respond back with "DCL" (and linefeed character appended
    // on the end) via the control connection when it has finished processing the
    // Device Clear request. Note that this 'Response' method uses the synchronous
    // form of 'Receive', so it could potentially time out if the instrument were
    // to take a long time to process the Device Clear. So alternatively the 'BeginListeningForAsyncReply' could be used for this instead of 'Response'.
    string deviceClearResponse = Response(controlClient);

    // Close both of the socket client sessions.
    controlClient.Close();
    client.Close();
} catch (ArgumentNullException e)
{
    Console.WriteLine("ArgumentNullException: {0}", e);
} catch (SocketException e)
{
    Console.WriteLine("SocketException: {0}", e);
}
Console.WriteLine("/n Press Enter to continue...");
Console.Read();
return 0;
}

static string Parse(Socket client, string command)
{
    // Translate the passed command into ASCII and store it as a Byte array.
    Byte[] data = System.Text.Encoding.ASCII.GetBytes(command);
    // Send the command to the socket-enabled instrument.
    client.Send(data);
    // Has to be followed by a linefeed character as terminator.
    Byte[] lf = {(Byte)'/n'};
    client.Send(lf);
    Console.WriteLine("Sent: {0}", command);
    // If the message was a query (involved a question mark), receive the
    instrument response.
    if (command.IndexOf("?") >= 0)
    {
        return Response(client);
    }
    return "";
}

static string Response(Socket client)
{
    // Buffer to store the response bytes.
    // For simplicity of this example, we allocate just for a 256-byte maximum
    // response size.
    Byte[] data = new Byte[256];
    // Read the batch of response bytes.
    Int32 byteCount = client.Receive(data);
    // String to store the response ASCII representation.
    string responseData = System.Text.Encoding.ASCII.GetString(data, 0, byteCount);
    Console.WriteLine("Received: {0}", responseData);
    return responseData;
static void BeginListeningForAsyncReply(Socket client)
{
    if (m_pCallbackFunc == null)
    {
        m_pCallbackFunc = new AsyncCallback(OnMessageReceived);
    }
    SocketPacket socPkt = new SocketPacket();
    socPkt.thisSocket = client;
    // Start asynchronously listening for a response from this client
    IAsyncResult result = client.BeginReceive (socPkt.data, 0, socPkt.data.Length, SocketFlags.None, m_pCallbackFunc, socPkt);
}

class SocketPacket
{
    public Socket thisSocket;
    // For simplicity of this example, we allocate just for a 256-byte maximum response size.
    public Byte[] data = new Byte[256];
}

static void OnMessageReceived(IAsyncResult asyn)
{
    SocketPacket socPkt = (SocketPacket)asyn.AsyncState ;
    Int32 byteCount = socPkt.thisSocket.EndReceive(asyn);
    m_AsyncReply = System.Text.Encoding.ASCII.GetString(socPkt.data, 0, byteCount);
    Console.WriteLine("Received: {0}", m_AsyncReply);
}
}
Last Modified:

28-Aug-2008   Updated with new TCPIP command
This Visual Basic program demonstrates two methods of reading the analyzer's status registers:

- **Polled Bit Method** - reads the Limit1 register continuously.
- **SRQ Method** - enables an interrupt of the program when bit 6 of the status byte is set to 1. The program then queries registers to determine if the limit line failed.

To run this program, you need:

- An established [GPIB interface connection](#)
- A form with two buttons: Poll and SRQ Method
- A means of causing the limit line to fail, assuming it passes initially.

```vbnet
Private Sub Poll_Click()
' POLL THE BIT METHOD
' Clear status registers
GPIB.Write "*CLS"

' Loop FOREVER
Do
  DoEvents
  GPIB.Write "STATus:QUEStionable:LIMit1:EVENt?"
  onn = GPIB.Read
Loop Until onn = 2
MsgBox "Limit 1 Failed"
End Sub

Private Sub SRQMethod_Click()
' SRQ METHOD
GPIB.Write "SYSTem:PRESet"
GPIB.Write "CALCulate:PARameter:SELect 'CH1_S11_1'"
' slow down the trace
GPIB.Write "SENS:BWID 150"

' Setup limit line
GPIB.Write "CALC:LIM:DATA 2,3e9,6e9,-2,-2"
GPIB.Write "CALC:LIMit:DISP ON"
GPIB.Write "CALC:LIMit:STATe ON"

' Clear status registers.
GPIB.Write "*CLS;*wai"
' Clear the Service Request Enable register.
GPIB.Write "*SRE 0"
' Clear the Standard Event Status Enable register.
GPIB.Write "*ESE 0"
```
' Enable questionable register, bit(10) to report to the status byte.
GPIB.Write "STATus:QUEStionable:ENABle 1024"

' Enable the status byte register bit3 (weight 8) to notify controller
GPIB.Write "*SRE 8"

' Enable the onGPIBNotify event
GPIB.NotifyMask = cwGPIBRQS
GPIB.Notify
End Sub

----------------------------------------------------
Private Sub GPIB_OnGPIBNotify(ByVal mask As Integer)
' check to see what failed
' was it the analyzer?
GPIB.Write "*STB?"
onn = GPIB.Read
If onn <> 0 Then
' If yes, then was it the questionable register?
GPIB.Write "STATus:QUEStionable:EVENt?"
onn = GPIB.Read
' Determine if the limit1 register, bit 8 is set.
If onn = 1024 Then
' If yes, then was it trace 1?
GPIB.Write "STAT:QUES:LIMIT1:EVEN?"
onn = GPIB.Read
If onn = 2 Then MsgBox ("Limit Line1 Failed")
End If
End If
End Sub
Transfer Data using GPIB

The following RMB examples transfer data to and from a remote PC using the \texttt{MMEM:TRANSfer} command.

Transferring data FROM the PNA -- TO a remote PC:

\begin{verbatim}
30  ! Set up I/O paths
40  ! Network analyzer address
50  ! File to be stored on local computer
60  ! First time -- need to create the file.
70  ! After file name, number records set to 0 (ignored by WinOS)
80  ! Use "PURGE" command to delete if desired.
90  CREATE "mytestdata.s2p",0
100 ASSIGN @File TO "mytestdata.s2p"
110  ! TRANSFER the data (download)
120  ! Analyzer has file 'testdata.s2p' in default directory
130 OUTPUT @Na;"`:MMEM:TRANS? ""testdata.s2p"
135  ! Now read the bytes coming back from the analyzer in four steps
138  ! (1) Read and dump the first character - '#'
140  ENTER @Na USING ":#;A";A$
141  ! (2) Read the next character which is the number of digits in the file size
150  ENTER @Na USING #:A;Digit$
155  ! (3) Use the value of the number of digits to read back the file byte size
160  ! Create query string using this number of digits
165  Img$="#,"&Digit$&"A"
170  ENTER @Na USING Img$;Byte$
175  ! (4) Read the file contents into a buffer and store the buffer contents to a local file
180  ! Allocate a buffer for holding the data
190  ALLOCATE Dat$[VAL(Byte$)]
195  ! Set up a different image for filling the buffer
200  Img$=Byte$&"A"
205  ! Retrieve the actual file data
210  ENTER @Na USING Img$;Dat$
220  ! Now save the file locally.
230  OUTPUT @File;Dat$
\end{verbatim}
Transferring data FROM the remote PC - TO the PNA:

40 ! Set up I/O paths
50 !
60 ! Network analyzer address
70 ASSIGN @Na TO 716
77 ! File to be retrieved from local computer
78 ASSIGN @File TO "mytestdata.s2p"
79 !
120 ! TRANSFER the data
123 !
230 ! Allocate a buffer for holding the data
240 ALLOCATE Dat$[26236]
250 !
260 ! Get data from the file and fill Dat$
270 ENTER @File;Dat$
280 !
325 ! Data to be transferred to analyzer file 'testupld.s2p'
326 ! in default directory.
327 !
328 ! A specific block transfer designator must follow the
329 ! file name:
330 ! '#' specifies a block transfer.
331 ! '6' specifies 6 digits to follow.
332 ! '026236' matches the buffer size allocated above
333 ! not counting <NL><END> (new line and end of file).
430 OUTPUT @Na;":MMEM:TRAN "'testupld.s2p'",#6026236",Dat$
520 END
Triggering the PNA using SCPI

To understand how to trigger the PNA using SCPI, it is very important to understand the PNA trigger model. Here is a very simple explanation. These three separate functions control PNA triggering:

1. **Trigger:Source** - Where the trigger signals originate:
   - Internal Continuous
   - Internal Manual (Single)
   - External - a trigger source that is connected to the PNA rear panel.

2. **Trigger:Scope** - what gets triggered:
   - Global - each signal triggers all channels in turn.
   - Channel - each signal triggers ONE channel.

3. Channel settings (Sense<ch>:Sweep:Mode) How many triggers will each channel accept before going into hold.
   - HOLD - channel will not trigger.
   - CONTinuous - channel triggers indefinitely.
   - GROups - channel accepts the number of triggers specified with the last SENS:SWE:GRO:COUN <num>.
   - SINGle - channel accepts ONE trigger, then goes to HOLD.
   - Point trigger  SENS1:SWE:TRIG:POINt

When controlling the PNA using SCPI, a SINGLE trigger is used to ensure that a complete sweep is taken. This example demonstrates how to Single trigger the PNA using the following two methods:

- **Simplest Triggering**  This method sets the Trigger Source to Internal Continuous - a stream of trigger signals. Each channel is configured to ACCEPT only a single trigger signal, then HOLD. This method can also be used when an External trigger source sends a continuous stream of trigger signals.

- **Advanced Triggering** This method SENDS a single trigger from the Source, which can be from either Internal (using INIT:IMM) or External triggering. Each channel is configured to accept an unlimited number of triggers. This method is the only way to perform point triggering. When you require some channels to accept continuous triggers and other channels to accept single triggers, see INIT:IMM Advanced to learn how.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text
editor file such as Notepad and save it on the PNA hard drive as Trigger.vbs. Learn how to setup and run the macro.

**Measurement setup example:** This section of code can be used at the start of both methods. It sets up:

- S11 traces on two channels
- 10 data points
- Sweep time of 2 seconds - this is slow enough to allow us to watch as each trace is triggered.

```vbs
Dim app
Dim scpi
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
'============================================
'Setup the PNA
'Preset the analyzer
scpi.Execute ("SYST:FReset")
' Create and turn on window/channel 1
scpi.Execute ("DISPlay:WINDow1:STATE ON")
'Define a measurement name, parameter
scpi.Execute ("CALCulate1:PARameter:DEFine:EXT 'MyMeas1',S11")
'Associate ("FEED") the measurement name ("MyMeas") to WINDow (1)
scpi.Execute ("DISPlay:WINDow1:TRACe1:FEED 'MyMeas1'")
' Create and turn on window/channel 2
scpi.Execute ("DISPlay:WINDow2:STATE ON")
'Define a measurement name, parameter
scpi.Execute ("CALCulate2:PARameter:DEFine:EXT 'MyMeas2',S11")
'Associate ("FEED") the measurement name ("MyMeas") to WINDow (2)
scpi.Execute ("DISPlay:WINDow2:TRACe2:FEED 'MyMeas2'")
'Set slow sweep so we can see
scpi.Execute ("SENS1:SWE:TIME 2")
scpi.Execute ('SENS2:SWE:TIME 2")
'set number of points to 10
scpi.Execute ("SENS1:SWE:POIN 10")
scpi.Execute ("SENS2:SWE:POIN 10")
```
'Put both channels in Hold
scpi.Execute("SENS1:SWE:MODE HOLD")
scpi.Execute("SENS2:SWE:MODE HOLD")
'================================

'Pick Single Send or Single Accept
resp=Msgbox("Single Send? - Click No for Single Accept", 4, "PNA Trigger Demo")
If resp=6 Then
    SingleSend()
Else
    SingleAccept()
End If

Simple Triggering  The following example sends a continuous stream of trigger signals and each PNA channel is set to ACCEPT only a signal trigger signal, then HOLD.

- This example can be used to configure External triggering where the trigger source sends a continuous stream of trigger signals. Configure the type of trigger signal that the PNA responds to using the **CONTrol:SIGNal** command. The command in this example sets the PNA to respond to HIGH TTL signals at the rear-panel BNC1 trigger IN connector. This command also automatically sets Trigger Source to External Trigger.

- The **TRIG SCOPE** (Global or Channel) setting is NOT necessary with a continuous stream of trigger signals. The example program directly controls when each channel is triggered.

- Point triggering can NOT be used with a continuous stream of trigger signals because in point triggering the channel will accept as many triggers as necessary to complete ONE full sweep. Use the single SEND example for point triggering.
Sub SingleAccept()
' PNA sends continuous trigger signals
scpi.Execute("TRIG:SOUR IMMediate")
' Uncomment the following to set External triggering
'scpi.Execute("CONT:SIGN BNC1,TILHIGH")
AcceptOne()
End Sub

Sub AcceptOne()
' The following command makes the channel immediately sweep
'*OPC? allows the measurement to complete before the controller sends another command
scpi.Execute("SENS1:SWE:MODE SINGle;*OPC?")
' You could do something to ch2 here before sweeping it
scpi.Execute("SENS2:SWE:MODE SINGle;*OPC?")
resp=Msgbox("Another trigger?", 1, "PNA Trigger Demo")
If resp=1 Then
AcceptOne()
End If
End Sub

Advanced Trigger
This example section performs Single Send triggering. Here, single triggering is accomplished by SENDING one trigger signal from the Trigger source and each channel is setup to accept unlimited trigger signals. See the INIT:IMM command for more details.

- Using this method, it is possible to change Trigger:Scope to Global or Channel. Set trigger scope to channel if there is some code to execute between channel measurements. Similarly, this method can be used to set Point triggering. Use this method if there is some code to execute between data point measurements.

- In addition, this method can also be used to perform External triggering if the external trigger source is capable of SENDING single triggers. See the CONTrol:SIGNal command to set the type of signal to which the PNA will respond.

- If the external source can only send a continuous stream of trigger signals, then the Single Accept section must be used.

Sub SingleSend()
' Set Source Internal - Manual Triggering
scpi.Execute("TRIG:SOUR MANual")
'If using an External trigger source that is capable of
'sending SINGLE trigger signals, then uncomment the following.
'This command automatically sets trigger source to External
'scp.i.Execute ("CONT:SIGN BNC1,TILHIGH")

'Setup Trigger Scope
'WHAT gets triggered
'Pick one using comments
'Set Channel triggering
'scp.i.Execute ("TRIG:SCOPe CURRent")
'Set Global triggering (Default)
scp.i.Execute ("TRIG:SCOPe ALL")

'Set Channel Settings
'The channels respond to UNLIMITED trigger signals (Default)
scp.i.Execute ("SENS1:SWE:MODE CONTinuous")
scp.i.Execute ("SENS2:SWE:MODE CONTinuous")

'To do Point trigger on one or more channels, uncomment the following.
'Point trigger automatically sets Trig:Scope to Current/Channel
'scp.i.Execute ("SENS1:SWE:TRIG:POINt ON")
'scp.i.Execute ("SENS2:SWE:TRIG:POINt ON")
IntTrig()
End Sub

Sub IntTrig()
'If External triggering, replace this Sub with code
'to single trigger the External Trig Source
Dim resp
'*OPC? allows the measurement to complete before the controller sends another
'command
scpi.Execute ("INITiate:IMMediate;*OPC?")
resp=Msgbox ("Another trigger?", 1, "PNA Trigger Demo")
If resp=1 Then
IntTrig()
End If
End Sub
Last modified:

18-Jun-2007    Updated with Sens:Swe:Mode Single
June 6, 2007    Changed order and wording
April 24, 2007  Updated with links
Oct. 5, 2006    New topic
Perform an Unguided Cal on Multiple Channels

This VBScript program performs an Unguided Calibration simultaneously on two channels. This could be used in the following cases:

- If you need more than the current number of data points per trace, so the additional points must be added to a different channel.
- If you need several channels with independent settings, but you want to calibrate all channels with a minimal number of standard connections. This would be especially critical for on wafer calibration.

The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unguided.vbs. Learn how to setup and run the macro.

```vbscript
Dim app
Dim scpi
Dim NumberOfActiveChannels
NumberOfActiveChannels = 2
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
' Query the list of connectors that the PNA system recognizes
scpi.Execute("SYST:PRES")
' Wait for successful preset before continuing
done=scpi.Execute("*OPC?")
'The following section sets up 2 channels with different frequency ranges
scpi.Execute("DISP:WIND1:STATE OFF")
'Reset Windows
scpi.Execute("DISP:WIND1:STATE ON")
scpi.Execute("DISP:WIND2:STATE ON")
'
' Assign a measurement to the first window
scpi.Execute("CALC1:PAR:DEF:EXT 'Meas1', S21")
scpi.Execute("DISP:WIND1:TRAC1:FEED 'Meas1'")
'Assign a measurement to the second window
scpi.Execute("CALC2:PAR:DEF:EXT 'Meas2', S21")
scpi.Execute("DISP:WIND2:TRAC1:FEED 'Meas2'")
```
'Set up two channels with independent parameters
scpi.Execute("SENS1:FREQ:SPAN 1e9")
scpi.Execute("SENS2:FREQ:SPAN 1e6")
'Wait for changes before continuing
done=scpi.Execute("*OPC?")
',
'This section sets the calibration kits for channel 1 and channel 2
'Select a trace from channel 1 and set calibration type and cal kit
scpi.Execute("CALC1:PAR:SEL 'Meas1'")
scpi.Execute("SENS1:CORR:COLL:METH SPARSOLT")
scpi.Execute("SENS1:CORR:COLL:CKIT 2") '85056D for default settings
'Same standards for forward and reverse direction
scpi.Execute("SENS1:CORR:TST OFF")
'Select a trace from channel 2 and set calibration type and cal kit
scpi.Execute("CALC2:PAR:SEL 'Meas2'")
scpi.Execute("SENS2:CORR:COLL:METH SPARSOLT")
scpi.Execute("SENS2:CORR:COLL:CKIT 2") '85056D for default settings
'Same standards for forward and reverse direction
scpi.Execute("SENS2:CORR:TST OFF")

'Set both channels to manual triggering
scpi.Execute("INIT1:CONT OFF")
scpi.Execute("INIT2:CONT OFF")
',
'The following assumes female port connector on port 1
' and male port connector on port 1
'Step through all active channels and calibrate and measure all standards.
scpi.Execute("SENS1:CORR:SFOR ON") 'Set acquisition to forward
scpi.Execute("SENS2:CORR:SFOR ON") 'Set acquisition to forward
MsgBox("Connect OPEN standard to port 1")
For CurrentChannel  = 1 To NumberOfActiveChannels
scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'")
scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan1")
done= scpi.Execute("*OPC?")
Next
MsgBox("Connect SHORT standard to port 1")
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan2")
    done=scpi.Execute("*OPC?")
Next

MsgBox("Connect LOAD standard to port 1")
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan3")
    done=scpi.Execute("*OPC?")
Next
    scpi.Execute("SENS1:CORR:SFOR OFF") 'Set acquisition to reverse
    scpi.Execute("SENS2:CORR:SFOR OFF") 'Set acquisition to forward

MsgBox("Connect OPEN standard to port 2")
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan1")
    done=scpi.Execute("*OPC?")
Next

MsgBox("Connect SHORT standard to port 2")
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan2")
    done=scpi.Execute("*OPC?")
Next

MsgBox("Connect LOAD standard to port 2")
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":'"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan3")
    done=scpi.Execute("*OPC?")
Next
'Measure thru standard for all channels in both forward and reverse direction
MsgBox("Connect THRU between ports 1 and 2")
scpi.Execute("SENS1:CORR:SFOR ON") ' Set acquisition to forward
scpi.Execute("SENS2:CORR:SFOR ON") ' Set acquisition to forward
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan4")
    done=scpi.Execute("*OPC?"")
Next
scpi.Execute("SENS1:CORR:SFOR OFF") ' Set acquisition to reverse
scpi.Execute("SENS2:CORR:SFOR OFF") ' Set acquisition to reverse
For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL stan4")
    done=scpi.Execute("*OPC?"")
Next

For CurrentChannel = 1 To NumberOfActiveChannels
    scpi.Execute("CALC" & CurrentChannel & ":PAR:SEL 'Meas" & CurrentChannel & ":"")
    scpi.Execute("SENS" & CurrentChannel & ":CORR:COLL:SAVE")
    done=scpi.Execute("*OPC?"")
Next

'Set both channels to continuous triggering
scpi.Execute("INIT1:CONT ON")
scpi.Execute("INIT2:CONT ON")
Upload and Download a Segment List

This VBScript program creates two segments, then uploads the segment data to the PNA. The second part downloads the segment list from the PNA. The SCPI commands in this example are sent over a COM interface using the SCPIStringParser object. You do NOT need a GPIB connection to run this example.

This VBScript (*.vbs) program can be run as a macro in the PNA. To do this, copy the following code into a text editor file such as Notepad and save it on the PNA hard drive as Unguided.vbs. Learn how to setup and run the macro.

See all Segment SCPI commands.

Create and Upload a Segment List

```vbs
Option Explicit
Dim app
Set app = CreateObject("AgilentPNA835x.Application")
' Preset the PNA
app.Preset
Dim scpi
Set scpi = app.ScpiStringParser
' In case of a measurement receiver PNA like N5264A
' which has no source ports, "SOURce:CATalog?" will
' return an empty list (just a pair of quotation marks)
Dim srcPortNames
srcPortNames = Split( scpi.Execute("SOURce:CATalog?"), ",")
Dim numberOfSrcPorts
If Left( srcPortNames(0), 2 ) = Chr(34) & Chr(34) Then
    numberOfSrcPorts = 0
Else
    numberOfSrcPorts = UBound(srcPortNames) + 1
End If
' Building up a string consisting of the sweep segment data
' we want to set up. This example will create two segments.
Dim segData
' Set state of first segment to be ON (1 = ON, 0 = OFF),
' 101 points, start freq of 10 MHz, stop freq of 1 GHz
segData = "1,101,10E6,1E9"
' If you want to include one or more of: IFbandwidth, Dwell Time
```
' or Port Power, remove the comments from these next two lines
'TurnOnOptions 1 'Call the subroutine
'segData = AddOptionalSettings(segData, numberOfSrcPorts)
' Set state of second segment to be ON, 201 points,
' start freq of 1 GHz, stop freq of 3 GHz
segData = segData & ",1,201,1E9,3E9"
' Uncomment this line below only if you uncommented the
' AddOptionalSettings line above for the first segment.
'segData = AddOptionalSettings(segData, numberOfSrcPorts)
Const numSegs = 2
' Upload our segment list to the channel
scpi.Execute "SENSe1:SEGMent:LIST SSTOP," & numSegs & "," & segData
' Set segment sweep type on Channel 1
scpi.Execute "SENSe1:SWEep:TYPE SEGment"
' Having the PNA display the segment sweep table for the channel
scpi.Execute "DISPlay:WINDow1:TABLe SEGMent"
Sub TurnOnOptions(ByVal chan)
    scpi.Execute "SENSe"&chan&":SEGMent:BWIDth:CONTrol ON"
    scpi.Execute "SENSe"&chan&":SEGMent:SWEep:TIME:CONTrol ON"
    scpi.Execute "SENSe"&chan&":SEGMent:POWer:CONTrol ON"
    ' Turning off coupling allows power to vary per each port
    scpi.Execute "SOURce"&chan&":POWer:COUPle OFF"
End Sub
Function AddOptionalSettings(ByVal inStr, ByVal numSrcPorts)
    ' Specifying 1 kHz IF bandwidth and Dwell Time of 0
    inStr = inStr & ",1E3,0"
    ' -10 dBm power for each of the source ports
    Dim i
    For i = 0 To numSrcPorts - 1
        inStr = inStr & ",-10"
    Next
    AddOptionalSettings = inStr
End Function

Download a Segment List
This example assumes that the active trace is in Window 1
Option Explicit
Dim app
Set app = CreateObject("AgilentPNA835x.Application")
Dim scpi
Set scpi = app.ScpiStringParser
' Set the display-active channel's sweep type to segment sweep
' (if the PNA's currently active measurement window doesn't
' contain any traces, this querying for active channel will
' result in a SCPI error which scpi.Parse will trap and throw)
Dim chan
chan = CLng( scpi.Parse("SYSTem:ACTive:CHANnel?"))
scpi.Execute "SENSe" & chan & ":SWEep:TYPE SEGment"
' Having the PNA display the segment sweep table for the channel
scpi.Execute "DISPlay:WINDow1:TABLE SEGment"
' Get the total number of segments
Dim numSegs
numSegs = CLng( scpi.Execute("SENSe" & chan & ":SEGment:COUNt?"))
' Read the segment listing
Dim segDataStr
segDataStr = scpi.Execute("SENSe" & chan & ":SEGment:LIST?"")
Dim segData
segData = Split(segDataStr, ",")
' Get upper bound of the array of data values
' (lower bound of array resulting from VB 'Split' function is 0)
Dim segArrayUB
segArrayUB = UBound(segData)
Dim numDataElementsPerSeg
numDataElementsPerSeg = (segArrayUB + 1) / numSegs
WScript.Echo "Number of segments = " & numSegs
WScript.Echo "Number of data values per segment = " & numDataElementsPerSeg
Dim segInfStr
segInfStr = "Segment 1: state = " & CBool(segData(0))
segInfStr = segInfStr & ", num points = " & CLng(segData(1))
segInfStr = segInfStr & ", start freq = " & CDbI(segData(2))
segInfStr = segInfStr & ", stop freq = " & CDbI(segData(3))
segInfStr = segInfStr & " IFBW = " & CDbI(segData(4))
segInfStr = segInfStr & ", dwell time = " & CDbl(segData(5))
' In case of a measurement receiver PNA like N5264A
' which has no source ports, "SOURce:CATalog?" will
' return an empty list
Dim srcPortNames
srcPortNames = Split( scpi.Execute("SOURce"&chan&":CATalog?")", ",")
Dim srcPortNamesUB
srcPortNamesUB = UBound(srcPortNames)
' First source port name will be preceded by a quotation mark
' and the last name will be followed by one of those, so stripping
' those off now.
srcPortNames(0) = Right( srcPortNames(0), Len(srcPortNames(0)) - 1 )
srcPortNames(srcPortNamesUB) = Left( srcPortNames(srcPortNamesUB),
InStrRev(srcPortNames(srcPortNamesUB), Chr(34)) - 1 )
Dim firstPortIndex
firstPortIndex = 6
Dim lastPortIndex
lastPortIndex = numDataElementsPerSeg - 1
Dim j
For j = firstPortIndex To lastPortIndex
    segInfStr = segInfStr & ", " & srcPortNames(j - firstPortIndex) & " power = " & CDbl(segData(j))
Next
WScript.Echo segInfStr
Uploading a Source Power Cal using SCPI

Programming the PNA using COM or using SICL/VISA over LAN (as in this example) leaves the PNA free to control GPIB devices as needed. This Visual Basic program demonstrates:

- Uploading a source power calibration of Port 2 for Channel 1.
- Reading the calibration data.

Learn more about Power Calibrations

Other SCPI Example Programs

To run this program, you need:

- Your PC and PNA both connected to a LAN (if using VISA LAN server / client).
- The SICL and VISA components of Agilent’s I/O Libraries software installed on your PC (both are included when you install the software, unless you already have another vendor’s VISA installed. Then specify Full SICL and VISA installation to overwrite the other vendor’s VISA.
- The module visa32.bas added to your VB project.
- A form with two buttons: cmdRun and cmdQuit.
- A VISA interface configured on your remote PC to control the PNA. This could be GPIB interface or a VISA LAN Client.

' Session to VISA Default Resource Manager
Private defRM As Long
' Session to PNA
Private viPNA As Long
' VISA function status return code
Private status As Long
Private Sub Form_Load()
defRM = 0
End Sub
Private Sub cmdRun_Click()

' String to receive data from the PNA.
' Dimensioned large enough to receive scalar comma-delimited values
' for 21 frequency points (20 ASCII characters per point)
Dim strReply As String * 420
Dim strPower As String, strCalPower As String
Dim strStimulus, strCalValue
Dim strResult As String

' Open the VISA default resource manager
status = viOpenDefaultRM(defRM)
If (status < VI_SUCCESS) Then HandleVISAError

' Open a session (viPNA) to the PNA at "address 16" on the VISA
' interface configured as "GPIB0" on this PC.
status = viOpen(defRM, "GPIB0::16::INSTR", 0, 0, viPNA)
If (status < VI_SUCCESS) Then HandleVISAError

' Set the number of sweep points to 2 on Channel 1.
status = myGPIBWrite(viPNA, "SENS1:SWE:POIN 2")
If (status < VI_SUCCESS) Then HandleVISAError

' Ensure there's currently no source power cal on for this channel and port.
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR OFF")
If (status < VI_SUCCESS) Then HandleVISAError

' Specify if the cal power level is offset (positive value for a gain, negative
' value for a loss) from the PNA port power setting on the channel when no source
' power cal is active. This is to account for components between the PNA test
' port and cal reference plane. In this example, let's set up our calibration
' at the output of an amplifier with 15 dB gain.
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:OFFS 15 DB")
If (status < VI_SUCCESS) Then HandleVISAError

' Prepare for doing data transfer in ASCII format.
status = myGPIBWrite(viPNA, "FORM:DATA ASCII")
If (status < VI_SUCCESS) Then HandleVISAError

' Send our source power correction data to the PNA. For purpose of simplicity
' in this example, we'll set up for no correction (0) at our start stimulus and
' 0.5 dB at our stop stimulus (recall that our sweep currently has just 2 points).
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:DATA 0,0.5")
If (status < VI_SUCCESS) Then HandleVISAError

' Set the number of sweep points to 21 on Channel 1.
status = myGPIBWrite(viPNA, "SENS1:SWE:POIN 21")
If (status < VI_SUCCESS) Then HandleVISAError

' Read the fixed power level for this port on Channel 1.
status = myGPIBWrite(viPNA, "SOUR1:POW2:LEV?")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError
strPower = strReply

' Turn the source power cal on.
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR ON")
If (status < VI_SUCCESS) Then HandleVISAError

' Again read the fixed power level for this port on Channel 1
' (with our calibration turned on, this should now include the 15 dB offset
' we indicated our power amplifier provides).
status = myGPIBWrite(viPNA, "SOUR1:POW2:LEV?")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError
strCalPower = strReply
'
' Read the stimulus values from Channel 1.
status = myGPIBWrite(viPNA, "SENS1:X?")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError
'
' Tokenize the reply string into an array containing the values
strStimulus = Split(strReply, ",")
'
' Read back the source power correction data, now interpolated for 21 points
status = myGPIBWrite(viPNA, "SOUR1:POW2:CORR:DATA?")
If (status < VI_SUCCESS) Then HandleVISAError
status = myGPIBRead(viPNA, strReply)
If (status < VI_SUCCESS) Then HandleVISAError
'
' Tokenize the reply string into an array containing the values
strCalValue = Split(strReply, ",")
'
' Print the data using a message box (here, Chr returns the ASCII characters
' for Tab (9) and Linefeed (10)).
strResult = "PNA port power = " & Val(strPower) & Chr(10)
strResult = strResult & "Power at reference plane = " & Val(strCalPower) & Chr(10)
strResult = strResult & "Stimulus" & Chr(9) & Chr(9) & "Cal Value" & Chr(10)
For i = 0 To UBound(strStimulus)
    strResult = strResult & Val(strStimulus(i)) & Chr(9) & Val(strCalValue(i)) & Chr(10)
Next
MsgBox strResult
End Sub
Private Sub cmdQuit_Click()
'
' Close the resource manager session (which also closes
' the session to the PNA).
If defRM <> 0 Then Call viClose(defRM)
'
' End the program
End Sub
Private Function myGPIBWrite(ByVal viHandle As Long, ByVal strOut As String) As Long
'
' The "+ Chr$(10)" appends an ASCII linefeed character to the
' output, for terminating the write transaction.
myGPIBWrite = viVPrintf(viHandle, strOut + Chr$(10), 0)
End Function
Private Function myGPIBRead(ByVal viHandle As Long, strIn As String) As Long
myGPIBRead = VIbsScanf(viHandle, "%t", strIn)

' Remove trailing linefeed character
If Right(strIn, 1) = Chr(10) Then strIn = Left(strIn, Len(strIn) - 1)
End Function

Sub HandleVISAError()
Dim strVisaErr As String * 200
Call viStatusDesc(defRM, status, strVisaErr)
MsgBox "*** Error : " + strVisaErr, vbExclamation

' Close the resource manager session (which also closes
' the session to the PNA).
If defRM <> 0 Then Call viClose(defRM)
End
End Sub
GPIB Fundamentals

The General Purpose Interface Bus (GPIB) is a system of hardware and software that allows you to control test equipment to make measurements quickly and accurately. This topic contains the following information:

- The GPIB Hardware Components
- The GPIB / SCPI Programming Elements
- Specifications
- GPIB Interface Capability Codes

Note: All of the topics related to programming assume that you already know how to program, preferably using a language that can control instruments.

Other Topics about GPIB Concepts

The GPIB Hardware Components

The system bus and its associated interface operations are defined by the IEEE 488 standard. The following sections list and describe the main pieces of hardware in a GPIB system:

Early PNA models had only ONE GPIB connector. These models could control other GPIB devices using one of, or a combination of, the following methods:

- Use the SCPI `SYST:COMM:GPIB:RDEV:` commands.
- Use VISA or SICL over LAN to accomplish this. See an example.
- Use USB / GPIB Interface

Note: Current PNA models have dedicated Controller and Talker/Listener GPIB ports. See how to configure these ports.

Controllers

Controllers specify the instruments that will be the talker and listener in a data exchange. The controller of the bus must have a GPIB interface card to communicate on the GPIB.

- The Active Controller is the computer or instrument that is currently controlling data exchanges.
- The System Controller is the only computer or instrument that can take control and give up control of the GPIB to another computer or instrument, which is then called the active controller.

Talker / Listener Instruments and GPIB Addresses

- Talkers are instruments that can be addressed to send data to the controller.
- **Listeners** are instruments that can be addressed to receive a command, and then respond to the command. All devices on the bus are required to listen.

Every GPIB instrument must have its own unique address on the bus. The PNA address (default = 716) consists of two parts:

1. **The Interface select code** (typically 7) indicates which GPIB port in the system controller is used to communicate with the device.

2. **The primary address** (16) is set at the factory. You can change the primary address of any device on the bus to any number between 0 and 30. To change the analyzer address click **System / Configure / SICL-GPIB**.

A secondary address is sometimes used to allow access to individual modules in a modular instrument system, such as a VXI mainframe. The PNA does NOT have a secondary address.

**Cables**

GPIB Cables are the physical link connecting all of the devices on the bus. There are eight data lines in a GPIB cable that send data from one device to another. There are also eight control lines that manage traffic on the data lines and control other interface operations.

You can connect instruments to the controller in any arrangement with the following limitations:

- Do not connect more than 15 devices on any GPIB system. This number can be extended with the use of a bus extension.

- Do not exceed a total of 20 meters of total cable length or 2 meters per device, whichever is less.

- Avoid stacking more than three connectors on the back panel of an instrument. This can cause unnecessary strain on the rear-panel connector.

**The GPIB / SCPI Programming Elements**

The following software programming elements combine to become a GPIB program:

- **GPIB / SCPI Commands**
- **Programming Statements**
- **Instrument Drivers**

**GPIB Commands**

The GPIB command is the basic unit of communication in a GPIB system. The analyzer responds to three types of GPIB commands:

1. **IEEE 488.1 Bus-management Commands**

These commands are used primarily to tell some or all of the devices on the bus to perform certain interface operations.
All of the functions that can be accomplished with these commands can also be done with IEEE 488.2 or SCPI commands. Therefore, these commands are not documented in this Help system. For a complete list of IEEE 488.1 commands refer to the IEEE 488 standard. Examples of IEEE 488.1 Commands

- CLEAR – Clears the bus of any pending operations
- LOCAL – Returns instruments to local operation

2. IEEE 488.2 Common Commands

These commands are sent to instruments to perform interface operations. An IEEE 488.2 common command consists of a single mnemonic and is preceded by an asterisk ( * ). Some of the commands have a query form which adds a “?” after the command. These commands ask the instrument for the current setting. See a complete list of the Common Commands that are recognized by the analyzer. Examples of IEEE 488.2 Common Commands

- *OPC – Operation Complete
- *RST – Reset
- *OPT? – Queries the option configuration

3. SCPI Commands

The Standard Commands for Programmable Instruments (SCPI) is a set of commands developed in 1990. The standardization provided in SCPI commands helps ensure that programs written for a particular SCPI instrument are easily adapted to work with a similar SCPI instrument. SCPI commands tell instruments to do device specific functions. For example, SCPI commands could tell an instrument to make a measurement and output data to a controller. Examples of SCPI Commands:

```plaintext
CALCULATE:AVERAGE:STATE ON
SENSE:FREQUENCY:START?
```

For more information on SCPI:

- The Rules and Syntax of SCPI Commands provides more detail of the SCPI command structure.
- SCPI Command Tree is a complete list of the SCPI commands for the analyzer

**Programming Statements**

SCPI commands are included with the language specific I/O statements to form program statements. The programming language determines the syntax of the programming statements. SCPI programs can be written in a variety of programming languages such as VEE, HP BASIC, or C++. Example of a Visual Basic statement:

```plaintext
GPIB.Write "SOURCE:FREQUENCY:FIXED 1000 MHz"
```

**Note about examples**

**Instrument Drivers**

Instrument drivers are subroutines that provide routine functionality and can be reused from program to program. GPIB industry leaders have written standards for use by programmers who develop drivers. When programmers write drivers that comply with the standards, the drivers can be used with predictable results. To comply with the
standard, each instrument driver must include documentation describing its functionality and how it should be implemented.

**GPIB Specifications**

**Interconnected devices** - Up to 15 devices (maximum) on one contiguous bus.

**Interconnection path** - Star or linear (or mixed) bus network, up to 20 meters total transmission path length or 2 meters per device, whichever is less.

**Message transfer scheme** - Byte-serial, bit-parallel, asynchronous data transfer using an interlocking 3-wire handshake.

**Maximum data rate** - 1 megabyte per second over limited distances, 250 to 500 kilobytes per second typical maximum over a full transmission path. The devices on the bus determine the actual data rate.

**Address capability** - Primary addresses, 31 Talk and 31 Listen; secondary addresses, 961 Talk and 961 Listen. There can be a maximum of 1 Talker and up to 14 Listeners at a time on a single bus. See also previous section on GPIB addresses.

**GPIB Interface Capability Codes**
The IEEE 488.1 standard requires that all GPIB compatible instruments display their interface capabilities on the rear panel using codes. The codes on the analyzer, and their related descriptions, are listed below:

- **SH1** full source handshake capability
- **AH1** full acceptor handshake capability
  - **T6** basic talker, serial poll, no talk only, unaddress if MLA (My Listen Address)
- **TEO** no extended talker capability
  - **L4** basic listener, no listen only, unaddress if MTA (My Talk Address)
- **LEO** no extended listener capability
- **SR1** full service request capability
- **RL1** full remote / local capability
- **PPO** no parallel poll capability
- **DC1** full device clear capability
- **DT1** full device trigger capability
- **C1** system controller capability
- **C2** send IFC (Interface Clear) and take charge controller capability
- **C3** send REN (Remote Enable) controller capability
C4 respond to SRQ (Service Request)

Last Modified:

30-Oct-2008 Removed legacy content
The Rules and Syntax of SCPI

Most of the commands used for controlling instruments on the GPIB are SCPI commands. The following sections will help you learn to use SCPI commands in your programs.

Branches on the Command Tree

Command and Query

Multiple Commands

Command Abbreviation

Bracketed (Optional) Keywords

Vertical Bars (Pipes)

MIN and MAX Parameters

Other Topics about GPIB Concepts

Branches on the Command Tree

All major functions on the analyzer are assigned keywords which are called ROOT commands. (See GPIB Command Finder for a list of SCPI root commands). Under these root commands are branches that contain one or more keywords. The branching continues until each analyzer function is assigned to a branch. A root command and the branches below it is sometimes known as a subsystem.

For example, under `SOURcE:POWer` are several branch commands.

Sometimes the same keyword, such as `STATE`, is used in several branches of the command tree. To keep track of the current branch, the analyzer's command parser uses the following rules:

- **Power On and Reset** - After power is cycled or after `*RST`, the current path is set to the root level commands.

- **Message Terminators** - A message terminator, such as a `<NL>` character, sets the current path to the root command level. Many programming language output statements send message terminators automatically. Message terminators are described in Sending Messages to the Analyzer.

- **Colon (:)** - When a colon is between two command keywords, it moves the current path down one level in the command tree. For example, the colon in `:SOURCE:POWER` specifies that `POWER` is one level below `SOURCE`. When the colon is the first character of a command, it specifies that the following keyword is a root level command. For example, the colon in `:SOURCE` specifies that source is a root level command.

**Note:** You can omit the leading colon if the command is the first of a new program line. For example, the following two commands are equivalent:

```
SOUR:POW:ATT:AUTO
:SOUR:POW:ATT:AUTO
```
Whitespace characters, such as `<tab>` and `<space>`, are generally ignored. There are two important exceptions:

- Whitespace inside a keyword, such as `:CALCULATE`, is not allowed.
- Most commands end with a parameter. You must use whitespace to separate these ending parameters from commands. **Always refer to the command documentation**. In the following example, there is whitespace between `STATE` and `ON`.

```
CALCULATE1:SMOOTHING:STATE ON
```

**Comma (,)** - If a command requires more than one parameter, you must separate adjacent parameters using a comma. For example, the `SYSTEM:TIME` command requires three values to set the analyzer clock: one for hours, one for minutes, and one for seconds. A message to set the clock to 8:45 AM would be `SYSTEM:TIME 8,45,0`. Commas do not affect the current path.

**Semicolon (;)** - A semicolon separates two commands in the same message without changing the current path. See **Multiple Commands** later in this topic.

**IEEE 488.2 Common Commands** - Common commands, such as `*RST`, are not part of any subsystem. An instrument interprets them in the same way, regardless of the current path setting.

### Command and Query

A SCPI command can be an **Event command**, **Query command** (a command that asks the analyzer for information), or both. The following are descriptions and examples of each form of command. GPIB Command Finder lists every SCPI command that is recognized by the analyzer, and its form.

<table>
<thead>
<tr>
<th>Form</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Event commands</strong> - cause an action to occur inside the analyzer.</td>
<td><code>:INITIATE:IMMEDIATE</code></td>
</tr>
<tr>
<td><strong>Query commands</strong> - query only; there is no associated analyzer state to set.</td>
<td><code>:SYSTem:ERRor?</code></td>
</tr>
<tr>
<td><strong>Command and query</strong> - set or query an analyzer setting. The query form appends a question mark (?) to the set form</td>
<td><code>:FORMat:DATA ! Command</code>  <code>:FORMat:DATA? ! Query</code></td>
</tr>
</tbody>
</table>

### Multiple Commands

You can send multiple commands within a single program message. By separating the commands with semicolons the current path does not change. The following examples show three methods to send two commands:

1. **Two program messages**:

```
SOURCE:POWER:START 0DBM
SOURCE:POWER:STOP 10DBM
```
2. **One long message.** A colon follows the semicolon that separates the two commands causing the command parser to reset to the root of the command tree. As a result, the next command is only valid if it includes the entire keyword path from the root of the tree:

```
SOURCE:POWER:START 0DBM; :SOURCE:POWER:STOP 10DBM
```

3. **One short message.** The command parser keeps track of the position in the command tree. Therefore, you can simplify your program messages by including only the keyword at the same level in the command tree.

```
SOURCE:POWER:START 0DBM; STOP 10DBM
```

### Common Commands and SCPI Commands

You can send Common commands and SCPI commands together in the same message. (For more information on these types of commands see [GP-IB Fundamentals](#).) As in sending multiple SCPI commands, you must separate them with a semicolon.

**Example** of Common command and SCPI commands together

```
*RST; SENSE:FREQUENCY:CENTER 5MHZ; SPAN 100KHZ
```

### Command Abbreviation

Each command has a long form and an abbreviated short form. The syntax used in this Help system use uppercase characters to identify the short form of a particular keyword. The remainder of the keyword is lower case to complete the long form.

<table>
<thead>
<tr>
<th>Short form</th>
<th>Long form</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOUR - Short form</td>
<td>SOURce - Long form</td>
</tr>
</tbody>
</table>

Either the complete short form or complete long form must be used for each keyword. However, the keywords used to make a complete SCPI command can be a combination of short form and long form.

The following is **unacceptable** - The first three keywords use neither short or long form.

```
SOURc:Powe:Atten:Auto on
```

The following is **acceptable** - All keywords are either short form or long form.

```
SOUR:POWer:ATT:AUTO on
```

In addition, the analyzer accepts lowercase and uppercase characters as equivalent as shown in the following equivalent commands:

```
source:POW:att:auto ON
Source:Pow:Att:Auto on
```

### Optional [Bracketed] Keywords

You can omit some keywords without changing the effect of the command. These optional, or default, keywords are used in many subsystems and are identified by brackets in syntax diagrams.

**Example** of Optional Keywords

The **HCOPY** subsystem contains the optional keyword **IMMediate** at its first branching point. Both of the following commands are equivalent:
Vertical Bars | Pipes

Vertical bars, or "pipes", can be read as "or". They are used in syntax diagrams to separate alternative parameter options.

Example of Vertical Bars:

SOURce:POWer:ATTenuation:AUTO <on|off>

Either ON or OFF is a valid parameter option.

MIN and MAX Parameters

The special form parameters "MINimum" and "MAXimum" can be used with commands that specify single frequency (Hz) and time (seconds) as noted in the command documentation. Note: Also with these commands, kHZ, MHz, and GHz are accepted as suffixes/units.

The short form (min) and long form (minimum) of these two keywords are equivalent.

- MAXimum refers to the largest value that the function can currently be set to
- MINimum refers to the smallest value that the function can currently be set to.

For example, the following command sets the start frequency to the smallest value that is currently possible:

SENS:FREQ:START MIN

In addition, the max and min values can also be queried for these commands.

For example, the following command returns the smallest value that Start Frequency can currently be set to:

SENS:FREQ:START? MIN

An error will be returned if a numeric parameter is sent that exceeds the MAX and MIN values.

For example, the following command will return an "Out of range" error message.

SENS:FREQ:START 1khz

Last Modified:

10-Jul-2007  Removed image
Configure for GPIB, SCPI, and SICL

The following settings are used to configure the PNA for remote control using SCPI commands.

### How to Configure for SICL / GPIB Operation

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then [<strong>Configure</strong>]</td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then [<strong>SICL/GPIB</strong>]</td>
<td>3. then <strong>Configure</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>SICL/GPIB/SCPI</strong></td>
</tr>
</tbody>
</table>

**SICL / GPIB** dialog box help

**GPIB**

**Talker/Listener Address**  Sets the PNA address used to send and receive GPIB/SCPI commands to the system controller (external computer).

Use the National Instruments interface or the ACE (Agilent Connection Expert) interface to change the
System Controller address. Use the PNA as the system controller of external devices. Learn about the PNA as controller.

See the rear panels of the PNA-X or PNA ‘C’ models.

**SICL**

**SICL Enabled** When checked, the analyzer is capable of running GPIB programs on its computer to control analyzer functions. The programs must be run from a GPIB-capable programming environment (VEE, Visual Basic). This mode does not allow control of external GPIB instruments. To uncheck this box, exit the PNA application - (Click File, then Exit). The PNA restarts with the SICL enabled box unchecked unless **Automatically Enable on Startup** is checked.

Learn more about Configuring for VISA and SICL.

**Address** Sets the PNA address.

**Automatically Enable on Startup** When checked, SICL Enabled is automatically selected when starting the PNA application.

**LAN Sockets/Telnet**

Provides ability to communicate with the PNA from a PC that uses a Windows, or non-Windows, operating system.

- These settings are checked by default. If you have security concerns, clear these check boxes.

- These settings remain after the PNA is shutdown and restarted.

**Sockets Enabled** When checked, provides the ability to control the PNA from a remote SCPI program using port number 5025. See the C# example that illustrates how this is done.

**Telnet Enabled** When checked, provides the ability to send single SCPI commands from a remote Windows, or non-Windows, PC to the PNA using port number 5024.

How to send single SCPI commands using Telnet:

1. On the remote PC, click **Start**, then **Run**

2. Type: `telnet <computer name> 5024`
   where `<computer name>` is the full computer name of the PNA. See how to find the computer name of the PNA.

3. A Telnet window with a **SCPI>** prompt should appear on the remote PC screen.

4. From the SCPI prompt:

   - Type single SCPI commands
   - If an invalid SCPI command is sent, the prompt will disappear. Press **Enter** or **Ctrl C** to recover the SCPI prompt.
     - To exit the telnet window click **X** in the upper-right corner.
     - To get a normal telnet prompt, press **Ctrl ]** (closing bracket).
To close the normal telnet window, type Quit and press Enter.

**SCPI Monitor / Input**

**GPIB Command Processor Console** Launches a window that is used to send single SCPI/GPIB commands from the PNA keyboard.

*Note:* Press Control+Z, then enter, to close the console window.

*Note:* The Status Register system can NOT be used from the GPIB Console.

- Type a valid command, with appropriate arguments and press enter.
- Use the arrow keys to recall previous commands.
- The console window may launch behind the PNA application. Press Control+Tab to bring the console window to the top.

**Monitor GPIB Bus** Enables monitoring activity on the GPIB.

**Show GPIB Bus Monitor Window** Shows and hides the window monitoring GPIB activity.

**Local and Remote Operation**

The analyzer LCL and RMT (Local and Remote) operation labels appear in the lower right corner of the status bar.

*Note:* The status bar is NOT visible when the analyzer is preset. See [how to make the status bar visible](#).

- **LCL** appears when NOT under SCPI control
- **RMT** appears when under SCPI control. The RMT label does NOT appear when under COM control. Remote operation disables the front panel keys except for the Macro/Local key.

To return to Local (front panel) operation, press the Macro / Local key

Sending the GPIB "GTL" (go to local) command also returns the analyzer to Local operation.

Sending the GPIB "LLO" (local lockout) command disables the front panel Local button.

---

Last Modified:

- 1-May-2009  Added note about Status Register and Console
- 30-Oct-2008  Removed legacy content.
- 21-Feb-2008  Include Windows OS in Telnet/Sockets
Getting Data from the Analyzer

Data is sent from the analyzer in response to program queries. Data can be short response messages, such as analyzer settings, or large blocks of measurement data. This topic discusses how to read query responses and measurement data from the analyzer in the most efficient manner.

Response Message Syntax

Clearing the Output Queue

Response Data Types

Transferring Measurement Data

Note: Some PCs use a modification of the IEEE floating point formats with the byte order reversed. To reverse the byte order for data transfer into a PC, use the FORMat:BORDer command.

Other Topics about GPIB Concepts

Response Message Syntax

Responses sent from the analyzer contain data, appropriate punctuation, and message terminators. <NL><^END> is always sent as a response message terminator. Most programming languages handle these terminators transparent to the programmer.

Response messages use commas and semicolons as separators in the following situations:

- a comma separates response data items when a single query command returns multiple values

  FORM:DATA? 'Query
  ASC, +0 'Analyzer Response

- a semicolon separates response data when multiple queries are sent within the same messages

  SENS:FREQ:STAR?;STOP? --Example Query
  +1.23000000000E+008; +7.89000000000E+008<NL><^END> 'Analyzer Response

Clearing the Output Queue

After receiving a query, the analyzer places the response message in it's output queue. Your program should read the response immediately after the query is sent. This ensures that the response is not cleared before it is read. The response is cleared when one of the following conditions occur:

- When the query is not properly terminated with an ASCII carriage return character or the GPIB <^END> message.

- When a second program query is sent.
When a program message is sent that exceeds the length of the input queue

- When a response message generates more response data than fits in the output queue.
- When the analyzer is switched ON.

**Response Data Types**

The analyzer sends different response data types depending on the parameter being queried. You need to know the type of data that will be returned so that you can declare the appropriate type of variable to accept the data. For more information on declaring variables see your programming language manual. The GPIB Command Finder lists every GPIB command and the return format of data in response to a query. The analyzer returns the following types of data:

- **Numeric Data**
- **Character Data**
- **String Data**
- **Block Data**

**Numeric Data**

All numeric data sent over the GPIB is ASCII character data. Your programming environment may convert the character data to numeric data for you. Boolean data (1 | 0) is a type of numeric data.

**Character Data**

Character data consists of ASCII characters grouped together in mnemonics that represent specific analyzer settings. The analyzer always returns the short form of the mnemonic in upper-case alpha characters. Character data looks like string data. Therefore, refer to the GPIB Command Finder to determine the return format for every command that can be queried.

**Example** of Character Data

```
MLOG
```

**String Data**

String data consists of ASCII characters. String parameters can contain virtually any set of ASCII characters. When sending string data to the analyzer, the string **must** begin with a single quote (') or a double quote (" ) and end with the same character (called the delimiter).

**Note**: The analyzer responds best to all special characters if the string is enclosed in single quotes. If quotes are not used, the analyzer will convert the text to uppercase. The analyzer may not respond as you expect.

The analyzer always encloses data in double quotes when it returns string data.

**Example** of String Data

```
GPIB.Write "DISP:WINDow:TITLe:DATA?"

"This is string response data."
```

**Block Data**
Block data is used to transfer measurement data. Although the analyzer will accept either definite length blocks or indefinite length blocks, it always returns definite length block data in response to queries unless the specified format is ASCII. The following graphic shows the syntax for definite block data:

```
<num_digits> specifies how many digits are contained in <byte_count>
<byte_count> specifies how many data bytes will follow in <data bytes>
```

**Example** of Definite Block Data

```
#17ABC+XYZ<nl><end>
```

Where:

- `#` - always sent before definite block data
- `1` - specifies that the byte count is one digit (7)
- `7` - specifies the number of data bytes that will follow, not counting <NL><END>
- `ABC+XYZ` - Data
- `<NL><END>` - always sent at the end of block data

**Transferring Measurement Data**

Measurement data is blocks of numbers that result from an analyzer measurement. Measurement data is available from various processing arrays within the analyzer. For more information on the analyzer’s data processing flow, see [Accessing Data Map](#). Regardless of which measurement array is read, transferring measurement data is done the same.

See an example.

When transferring measurement data, the **FORMat:DATA** command allows you to choose from the following two data types:

- **REAL**
- **ASCII**

The following graphic shows the differences in transfer times between the two:
REAL Data

REAL data (also called floating-point data) types transfer faster. This is because REAL data is binary and takes about half the space of ASCII data. The disadvantage of using REAL data is that it requires a header that must be read. See definite length block data. The binary floating-point formats are defined in the IEEE 754-1985 standard. The following choices are available in REAL format:

- **REAL,32** - IEEE 32-bit format - single precision (not supported by HP BASIC)
- **REAL,64** - IEEE 64-bit format - double precision

ASCII Data

The easiest and slowest way to transfer measurement data is to use ASCII data. ASCII data is sent if the data contains both numbers and characters (the setting of FORMat:DATA is ignored). ASCII data is separated by commas.

Last Modified:

- 28-Jul-2009 Added three zeros to example
- 26-Jul-2007 Added link to example
Synchronizing the PNA and Controller

In this topic, Synchronizing the PNA and Controller means to keep PNA and the controller working at approximately the same pace.

See Also
- Synchronize an External PSG Source
- Triggering the PNA using SCPI

The Problem
The controller sends commands to the PNA as fast as the GPIB bus will allow. The PNA stores these commands in the PNA Input queue. However, the PNA executes those commands at a slower rate than they are accepted. If left unchecked, the PNA input buffer will contain a long list of commands waiting to be executed.

At some point, the controller will send a query command which requires a response from the PNA. The controller will not send more commands until a response is received. It will wait for a response from the PNA for the amount of time set by the Timeout setting. If the PNA is working off a long list of commands in the input buffer, it may not execute and respond to the query command until the controller has quit waiting, or "timed out".

The Solution
The easiest way to keep the controller and the PNA "synched" is to send query commands often. This stops the controller from sending more commands until the PNA executes and responds to the query. This limits the number of commands that are waiting in the PNA input queue to be processed.

Although any query will stop the controller from sending more commands, a good practice is to send *OPC? Most of the time, as soon as this query is executed, it will immediately reply. The exception to this is the Overlapped command.

- **Sequential** commands are executed quickly and in the order in which they are received.

- **Overlapped** commands take longer to execute. Therefore, they allow the PNA to execute other commands while waiting. However, the programmer may want to prevent the analyzer from processing new commands until the overlapped command has completed. If the PNA is executing an overlapped command when a *OPC? is received, it will wait until the overlapped command is complete.

Note: The analyzer has two overlapped commands:

- **INITiate:IMMediate**
- **SENSe:SWEep:MODE GROUPS** (when INIT:CONT is ON)
- **Calibration Acquire commands using the ASYN argument**

Analyzer Queues
Queues are memory buffers that store messages until they can be processed. The analyzer has the following queues:
**Input Queue**

**Output Queue**

**Error Queue**

**Input Queue**

The controller sends statements to the analyzer without regard to the amount of time required to execute the statements. The input queue is very large (31k bytes). It temporarily stores commands and queries from the controller until they are read by the analyzer's command parser. The input queue is cleared when the analyzer is switched ON.

**Output Queue**

When the analyzer parses a query, the response is placed in the output queue until the controller reads it. Your program should immediately read the response or it may be cleared from the output queue. The following conditions will clear a query response:

- When a second query is sent before reading the response to the first. This does not apply when multiple queries are sent in the same statement.
- When a program statement is sent that exceeds the length of the input queue.
- When a response statement generates more data than fits in the output queue.
- When the analyzer is switched ON.

**Error Queue**

Each time the analyzer detects an error, it places a message in the error queue. When the `SYSTEM:ERROR?` query is sent, one message is moved from the error queue to the output queue so it can be read by the controller. Error messages are delivered to the output queue in the order they were received. The error queue is cleared when any of the following conditions occur:

- When the analyzer is switched ON.
- When the `*CLS` command is sent to the analyzer.
- When all of the errors are read.

If the error queue overflows, the last error is replaced with a "Queue Overflow" error. The oldest errors remain in the queue and the most recent error is discarded.

**Synchronization Methods**

The following common commands are used to synchronize the analyzer and controller. Examples are included that illustrate the use of each command in a program. See the SCPI command details to determine if a command is an overlapped command.

`*WAI`

`*OPC?`

`*OPC`
**WAI**

The *WAI command:

- **Stops the analyzer** from processing subsequent commands until all overlapped commands are completed.
- **It does NOT stop the controller** from sending commands to this and other devices on the bus. This is the easiest method of synchronization.

**Example** of the *WAI command

GPIB.Write "ABORT;:INITIATE:IMMEDIATE" 'Restart the measurement.
GPIB.Write "CALCULATE:MARKER:SEARCH:MAXIMUM" 'Search for max amplitude.
GPIB.Write "CALCULATE:MARKER:X?" 'Which frequency?

The following time line shows how the processing times of the three commands relate to each other:

```
 ABORT;:INITIATE:IMMEDIATE

   CALCULATE:MARKER:MAXIMUM

   CALCULATE:MARKER:X?
```

**INITIATE:IMMEDIATE** is an overlapped command; it allows the immediate processing of the sequential command, **CALCULATE:MARKER:SEARCH:MAXIMUM**. However, the **INITIATE:IMMEDIATE** is not considered complete until the measurement is complete. Therefore, the marker searches for maximum amplitude before the measurement completes. The **CALCULATE:MARKER:X?** query could return an inaccurate value.

To solve the problem, insert a *WAI command.

GPIB.Write "ABORT;:INITIATE:IMMEDIATE" 'Restart the measurement.
GPIB.Write "*WAI" 'Wait until complete.
GPIB.Write "CALCULATE:MARKER:MAXIMUM" 'Search for max amplitude.
GPIB.Write "CALCULATE:MARKER:X?" 'Which frequency

The time line now looks like this:

```
 ABORT;:INITIATE:IMMEDIATE

   *WAI

   CALCULATE:MARKER:MAXIMUM

   CALCULATE:MARKER:X?
```

The *WAI command keeps the **MARKER:SEARCH:MAXIMUM** from taking place until the measurement is completed. The **CALCULATE:MARKER:X?** query returns the correct value.

**Note**: Although *WAI stops the analyzer from processing subsequent commands, it does not stop the controller. The controller could send commands to other devices on the bus.

**OPC?**
The *OPC? query stops the controller until all pending commands are completed.

In the following example, the Read statement following the *OPC? query will not complete until the analyzer responds, which will not happen until all pending commands have finished. Therefore, the analyzer and other devices receive no subsequent commands. A "1" is placed in the analyzer output queue when the analyzer completes processing an overlapped command. The "1" in the output queue satisfies the Read command and the program continues.

Example of the *OPC? query

This program determines which frequency contains the maximum amplitude.

```
GPIB.Write "ABORT; :INITIATE:IMMEDIATE"! Restart the measurement
GPIB.Write "*OPC?" 'Wait until complete
Meas_done = GPIB.Read 'Read output queue, throw away result
GPIB.Write "CALCULATE:MARKER:MAX" 'Search for max amplitude
GPIB.Write "CALCULATE:MARKER:X?" 'Which frequency?
Marker_x = GPIB.Read
PRINT "MARKER at " & Marker_x & " Hz"
```

*OPC

The *OPC command allows the analyzer and the controller to process commands while processing the overlapped command.

When the analyzer completes processing an overlapped command, the *OPC command sets bit 0 of the standard event register to 1. This requires polling of status bytes or use of the service request (SRQ) capabilities of your controller. See Reading the Analyzer's Status Registers for more information about the standard event status register, generating SRQs, and handling interrupts.

**Note:** Be careful when sending commands to the analyzer between the time you send *OPC and the time you receive the interrupt. Some commands could jeopardize the integrity of your measurement. It also could affect how the instrument responds to the previously sent *OPC.

Example of polled bit and SRQ processes.

**When To Synchronize the Analyzer and Controller**

The need to synchronize depends upon the situation in which the overlapped command is executed. The following section describes situations when synchronization is required to ensure a successful operation.

**Completion of a Measurement**

**Averaged Measurements**

Completion of a Measurement

To synchronize the analyzer and controller to the completion of a measurement, use the ABORT;INITIATE:IMMEDIATE command sequence to initiate the measurement.

This command sequence forces data collection to start (or restart) under the current measurement configuration. A restart sequence, such as ABORT;INITIATE:IMMEDIATE is an overlapped command. It is complete when all operations initiated by that restart command sequence, including the measurement, are finished. The *WAI,*OPC? and *OPC commands allow you to determine when a measurement is complete. This ensures that valid measurement data is available for further processing.
Measurements with External Trigger

See Triggering the PNA using SCPI.

External Triggering

Averaged Measurements

Averaged measurements are complete when the average count is reached. The average count is reached when the specified number of individual measurements is combined into one averaged measurement result. Use synchronization to determine when the average count has been reached.

If the analyzer continues to measure and average the results after the average count is reached, use synchronization to determine when each subsequent measurement is complete.

Last Modified:

6-Apr-2009  Added links to External trig and triggering
17-Feb-2009  Added Cal ACQ to Overlapped commands
Calibrating the PNA Using SCPI

There are several ways to calibrate the PNA using SCPI depending on your measurement needs. As from the Cal Wizard, you can perform a Guided Cal, Unguided Cal, or ECal. This topic explains the differences in these calibration choices when using SCPI commands.

- **Guided Calibrations**
- **ECal**
- **Perform Computer Operations During Acquire**
- **Creating Cal Sets**
- **Applying Cal Sets and Cal Types**
- **Uploading Error Terms**
- **Unguided Cals and Calibration Classes**

**See SCPI Calibration Examples**

**Guided Calibrations**

- To perform a **Guided Calibration**, use ONLY `Sens:Corr:Coll:Guided` commands.
- To perform an **Unguided Calibration**, use ONLY `Sens:Corr` commands (NOT Guided).
- These commands calibrate the ACTIVE channel. Activate a channel by selecting a measurement on the channel to be calibrated using `Calc:Par:Select`.
- Full 1,2,3,4-port SOLT and TRL calibrations - No response cals.
- All of the advanced calibration features (Thru method, specify DUT connectors and Cal kits for each port, port pairings).
- A Cal Set is applied to the channel and saved at the completion of a guided cal according to the preference setting `SENS:CORR:PREF:CSET:SAVE`.

**ECal**

From the Cal Wizard or from a SCPI program, ECal is fast, accurate, and very repeatable. Unlike from the Cal Wizard, you can use SCPI to perform ECal using either the Guided or Unguided commands. The Unguided commands are easiest to use. However, the following situations require that you use the Guided commands.

- To maximize accuracy, all ECal calibrations on the PNA perform an Unknown Thru measurement of the ECal module Thru state **IF** the PNA model being used has **1 reference receiver per port**. If your PNA does **NOT** have 1 reference receiver per port, use Guided ECal commands and specify a Thru method.
If your ECal module connectors do NOT match the DUT connectors, and you choose not to perform a User Characterization, use Guided ECal commands and specify the Thru method.

**ECAL Notes:**

- When using either Guided or Unguided ECal commands under low power situations, use the Orientation settings. The Guided example shows the use of these commands. When using Unguided, they must appear before the Acquire command.
- The frequency range of the measurement must be within the range of the ECal module. Otherwise, the calibration will fail.
- Although we recently provided the command, you do NOT have to send the ECal module state command. The ECal algorithm switches ECal states automatically.
- All of these ECal choices are listed in the Programming Command Search function in this Help file.

See [Using ECal](#) to learn about all of the ECal features.

**Perform Computer Operations During Acquire**

During a calibration with slow sweep speeds, such as when using a narrow IF bandwidth, you may want to have your program perform other operations, such as checking for the click event of a Cancel button.

To do this, use the optional ASYNchronous argument with the ACQuire command as shown in several calibration example programs. The PNA parser returns immediately while the cal step measurement proceeds. It does NOT block commands and wait for the measurement step to finish. You can send *ESR? or *STB? queries to monitor the status register bytes to see when the OPC (operation complete) bit gets set, which indicates the cal measurement step has finished. Learn more about [status registers](#).

If your program is using the ScpiStringParser, then you can ONLY use *OPC? to detect when the OPC bit is set. When using the ASYN argument, set the timeout value in the IO settings to at least 5 seconds. There are intervals during the cal acquires when the PNA takes a several seconds to respond to additional commands, such as when the processor is calculating error terms.

The following Acquire commands have this argument:

<table>
<thead>
<tr>
<th>Command</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:CORR:COLL:GUID:ACQuire (Guided Cal)</td>
<td>Guided 2-Port or 4-Port Cal</td>
</tr>
<tr>
<td>SENS:CORR:COLL:ACQuire (Unguided Cal)</td>
<td>Perform Unguided ECAL</td>
</tr>
<tr>
<td>SENS:CORR:COLL:SESS:ACQuire (FCA)</td>
<td></td>
</tr>
</tbody>
</table>

Learn more about [Synchronizing the PNA and Controller](#).
Creating Cal Sets

There are several ways to store guided cal data into a unique Cal Set. The following is probably the easiest. It does not require the name of an existing Cal Set and it allows you to name the Cal Set.

```
SENS:CORR:COLL:GUID:INIT  'start the cal with no cal set argument
'Sens the cal
SENS:CORR:COLL:GUID:SAVE  'create cal set with auto-generated name or to cal register
SENS:CORR:CSET:NAME 'MyCalSet'  'name the current cal set.
```

Applying Cal Sets and Cal Types

A Cal Set is applied to the channel and saved at the completion of a guided cal according to the preference setting `SENS:CORR:PREF:CSET:SAVE`.

When you select a Cal Set to apply to an uncalibrated channel, the PNA attempts to find the most comprehensive calibration type in the Cal Set and turn it ON. In addition, changing a measurement parameter (for example, from S11 to S21) will also initiate an attempt to apply the best Cal Type and turn correction ON.

There may be times when you do not want the most comprehensive Cal Type. For example, say there is a Full 2-port Cal Set applied, but there is only an S11 measurement displayed. If measurement speed is a concern, you can apply a Full 1-Port Cal Type from that same Cal Set and save time by not doing the extra background sweeps. Learn more about background sweeps.

If you change the measurement parameter, the PNA will reapply the Full 2-Port Cal Type.

See the SCPI and COM commands for Cal Sets and Cal Types.

Uploading Error Terms

There are two ways to upload error terms using SCPI: the old way and the recommended way. The old way will still works but requires a 'preference' setting.

**The old way is this:**

```
SENS:CORR:COLL:METHod <cal type>
SENS:CORR:COLL:SAVE
CALC:DATA SCORR[n]  < data to upload >
```

This technique, used in WinCal software, starts a calibration and immediately saves it without acquiring any standards. In PNA Rev 6.0, executing SAVE without acquiring data will return an error. To suppress the error and continue to use the above technique to upload error terms, send the following command to set the preference:

`SENS:CORR:PREF:SIMCal 1`

Or you can execute the script that is saved on the PNA at `C:/Program Files/Agilent/Network Analyzer/System/wincal32.reg`.

Setting this preference defeats some error checking when performing unguided cal using SCPI. This is not recommended unless needed for backward compatibility.

**The recommended way** is to upload error terms into a created or selected Cal Set:

```
SENS:CORR:CSET:CREATE or SENS:CORR:CSET:GUID
SENS:CORR:CSET:Data <term> <port> <port> <data>
```
This method puts error terms into a Cal Set, outside of a Guided or Unguided calibration session. The Cal Set can then be applied at any time.

See SENS:CORR:CSET commands.

**Unguided Cals and Calibration Classes**

- Use Sens:Correction commands.
- 1-port, 2-port, Response.
- Can select 2 sets of standards.
- TRL is NOT recommended.

The following describes how to perform an unguided calibration using SCPI. The objective here is to make clear the relationship between the physical port on which a standard is being measured, the actual device in the cal kit, and the SCPI command used to acquire the device.

Calibration standards classes are ‘categories’ of standard types. To perform a 2 port calibration, the cal wizard requires the following types of standards to be measured:

**3 reflection standards on the forward port:**

- Class S11A typically an open
- Class S11B typically a short
- Class S11C typically a load

**Likewise, 3 reflection standards are required for the reverse port:**

- Class S22A typically an open
- Class S22B typically a short
- Class S22C typically a load

**There is also a transmission standard that is measured in both directions:**

- Class S21T typically a thru

The following illustrates the relationship between cal kit physical standards and calibration classes. Here is a list of the physical devices in my calibration kit.

Standard #1 = "3.5 mm male short"
Standard #2 = "3.5 mm male open"
Standard #3 = "3.5 mm male broadband load"
Standard #4 = "Insertable thru standard"
When you perform a calibration remotely using SCPI, you don’t specify the device number directly. Rather, you specify the class you want to measure. Each device in the calibration kit is assigned to a class. And since more than one device can be assigned to the same class, each class contains an ordered list of devices. The class assignments are set using the Advanced Modify Cal Kit dialog or the SCPI command:

```
SENS:CORR:COLL:CKIT:ORDER <class>, <std>, <std>, <std>, <std>, <std>, <std>, <std>
```

The 85052B kit used in the example program has the following standard list for each class: The list was obtained by issuing the corresponding SCPI query:

```
SENS:CORR:COLL:CKIT:OLIST1?  S11A = +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST2?  S11B = +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST3?  S11C = +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST4?  S21T = +4,+8,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST5?  S22A = +2,+15,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST6?  S22B = +1,+7,+0,+0,+0,+0,+0
SENS:CORR:COLL:CKIT:OLIST7?  S22C = +6,+5,+3,+12,+13,+14,+0
SENS:CORR:COLL:CKIT:OLIST8?  S12T = +4,+8,+0,+0,+0,+0,+0
```

When you perform the calibration, you acquire data by issuing the ACQuire command:

```
SENS:CORR:COLL:ACQ <class>[, <subst>]
```

For example:

```
SENS:CORR:COLL:SFOR 1
SENS:CORR:COLL:ACQ STANA, SST2
```

The SFOR command tells the wizard to make the next acquisition in the forward direction. The ACQuire command specifies that we are measuring the 2nd device in the list for STANA. And since we are measuring SFORward, then STANA refers to class #1 or S11A. The list of devices for this class are specified in the OLIST1 query above.

Alternately, you could modify the device order for the S11A class to move device #15 into the first position (SENS:CORR:COLL:CKIT:ORDER1). When the desired device is in the first position, you need not specify the order number in the ACQuire command. The default is the first device in the OLIST. This works well for two port network analyzers where the order for S11A,B,C classes is set up for port 1 and the order for S22A,B,C is set up for port 2. With the kit set up in the proper order, you eliminate the need to specify the substandard number.
See an example: Perform an Unguided 2-port Cal on a 4-port PNA.
The PNA as a USB Device

Beginning with PNA Rev. A.09.00, the PNA can be controlled as a USB Device using SCPI. This is done through the Agilent I/O Libraries which must be installed on your remote computer.

All data types, especially Binary block data, transfer MUCH faster using USB as compared to GPIB.

To communicate with the PNA as a USB device

1. Connect the PNA to the remote computer using the rear-panel device-side USB connector. This connector is available ONLY on PNA models with a 1.1 GHz CPU board, and the PNA-X.

2. The ‘Found New Hardware’ wizard is launched. Follow the prompts to install the PNA driver software.

3. The Agilent I/O Libraries will recognize the PNA as a Test and Measurement device and show the following dialog.

Note: The PNA is not a USB Mass Storage Device. Therefore, Windows Explorer does NOT recognize it as a USB device. You can NOT use Windows Explorer to transfer files to and from the PNA. For file transfer, use the SCPI command MMEM:TRANsfer.

Alias name Change this to a name that is easy to recognize. Once configured, use the Alias name to communicate with the USB device using applications such as VISA and SICL:

- VISA: viOpen (...,"UsbDevice1",...)
- SICL: iopen ("UsbDevice1")

For more information, see the Connectivity Guide in the Agilent I/O libraries.
Reading the Analyzer's Status Register

The PNA has several status registers that your program can read to know when specific events occur. There are two methods of reading the status registers in the analyzer: the Polled Bit method and the Service Request method.

- The Status Registers
- Setting and Reading Bits in Status Registers
- Polled Bit Method
- Service Request Method

See Also
IEE 482 Common commands
Example: Status Reporting
Status Commands

Other Topics about GPIB Concepts

Important Notes:

- A new **Limit Line Fail command** that makes it easy to determine if Limit Line testing has failed.
- **OPC?** can be used to easily determine when a channel has completed a sweep. This requires no interaction with the Status Register system. Most [PNA programming examples](#) use *OPC*.
- Most of the Status Register system can NOT be used with the [SCPIStringParser Object](#). However, *OPC?* can be used.

The Status Registers

Most of the status registers in the analyzer have sixteen bits. For simplicity, this topic will illustrate their use with 8-bit registers. Bits in registers represent the status of different conditions inside of the analyzer. In the following graphic, a register is represented by a row of boxes; each box represents a bit. Bit 3 is ON.

```
7 6 5 4 3 2 1 0
```

Each PNA Status Register is actually comprised of the following registers. [See an image of the PNA Status registers.](#)

- **Enable Registers** - When using the [SRQ method of polling](#), you first set bits in the enable register which tells the PNA which events to monitor. This is not necessary using the [Polled Bit method](#), as you can only monitor a single event. A *CLS (clear status) command will not clear the enable register. The *ESE and
*ESE? commands are used to set and query Enable bits, while *ESR is used to read and clear an Enable register. Learn how to set bits.

- **Condition Registers** - A condition register continuously monitors events in the PNA. Bits in the condition register change real time as conditions occur. These bits are not latched, so this register is used mainly for diagnostic purposes. The registers that only summarize lower level registers do NOT have a condition register.

- **Event Registers** - This is the register that is read to determine if an event has occurred. An event register latches the bits from the corresponding condition register. When an event register bit is set, subsequent changes to the corresponding condition register bit are ignored. The bit remains set until a query command such as *CLS clears the bit. Learn how to read the Event Register.

- **Positive and Negative Transition Registers** - Transition registers control what type of change in a condition register will set the corresponding bit in the event register.
  
  - **Positive** transitions (0 to 1) are only reported to the event register if the corresponding positive transition bit is set to 1.
  
  - **Negative** transitions (1 to 0) are only reported to the event register if the corresponding negative transition bit is set to 1.

  Setting both transition bits to 1 causes both positive and negative transitions to be reported.

Transition registers are read-write and are unaffected by *CLS (clear status) or queries. They are reset to their default settings at power-up and after *RST and SYSTem:PRESet commands. The following are the default settings for the transition registers:

- All Positive Transition registers = 1
- All Negative Transition registers = 0

This means that, by default, the analyzer will latch all event registers on the negative to positive transition (0 to 1).

The following is an example of why you would set transition registers:

A critical measurement requires that you average 10 measurements and then restart averaging. You decide to poll the averaging bit. When averaging is complete, the bit makes a positive transition. After restart, you poll the bit to ensure that it is set back from 1 to 0, a negative transition. You set the negative transition bit for the averaging register.

### Setting and Reading Bits in Status Registers

Both the Polled-Bit method and Service Request method require that you set and read status register bits. Most of the PNA status registers contain 16 bits, numbered 0 to 15. Each bit has a weighted value. The following example shows how to set the bits in a 8-bit status register.

<table>
<thead>
<tr>
<th>Bit</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>1</td>
<td>2</td>
<td>4</td>
<td>8</td>
<td>16</td>
<td>32</td>
<td>64</td>
<td>128</td>
</tr>
</tbody>
</table>

How to set bits 4 and 5 in the Standard Event Status Enable register:
Step | Example
---|---
1. Determine the weighted bit value for these bits weights 16 and 32 (respectively)
2. Add these values together $16 + 32 = 48$
3. Send this number as an argument in the appropriate command. (see Status Commands) STAT:QUES:LIMIT1:ENAB 48

The Polled Bit Method
With the Polled Bit Method, your program monitors a bit in the status register that represents the condition of interest to you. When the PNA sets the bit to 1, your program sees it and responds accordingly.

- If your program **periodically** monitors a bit in the status register, it is free to do other things as well. However, your program can respond only as fast as the bit is polled.
- If your program **continually** monitors a bit, it can respond immediately, but will be unavailable to do anything other than poll the bit.

**Advantage:** This method requires very little programming.

**Procedure:**

1. Decide which condition to monitor. The Status Commands topic lists all of the possible conditions that can be monitored in the analyzer.
2. Determine the command to be used to monitor the bit.
3. Determine how often to poll the bit until it is set.
4. Construct the routine to respond when the bit is set.

The Service Request (SRQ) Method
Your program enables the bits in the status registers representing the condition of interest. When the condition occurs, the PNA actively interrupts your program from whatever it is doing, and an event handler in your program responds accordingly. Do this method if you have several conditions you want to monitor or the conditions are such that it is not practical to wait for the condition to occur.

**Advantage:** This method frees your program to do other things until the condition occurs. The program is interrupted to respond to the condition.

**Disadvantage:** This method can require extensive programming depending on the number and type of conditions that you want to monitor.

**Procedure:**

1. Decide which conditions to monitor. The Status Commands topic lists all of the possible analyzer conditions that can be monitored.
2. Set the enable bits in the **summary** registers and the **status byte** register.
Enabling is like making power available to a light. Without power available, the switch can be activated, but the light won't turn ON. In the analyzer, without first enabling a bit, the condition may occur, but the controller won't see it unless it is enabled.

The condition, and the bit in the summary registers in the reporting path, must be enabled. This is like streams (conditions) flowing into rivers (summary registers), and rivers flowing into the ocean (controller). See the diagram of status registers in Status Commands.

Bit 6 of the status byte register is the only bit that can interrupt the controller. When any representative bit in the status byte register goes ON, bit 6 is automatically switched ON.

3. Enable your program to interrupt the controller. This is done several ways depending on the programming language and GPIB interface card you use. An example program is provided showing how this is done with in Visual Basic with a National Instruments GPIB card.

4. Construct a subroutine to handle the interrupt event. If you are monitoring more than one condition in your system, your event handler must determine which condition caused the interrupt. Use the *SPE command to determine the instrument that caused the interrupt, then poll the summary registers, then poll condition registers to determine the cause of the interrupt.

Last Modified:

4-Jun-2009    Modified Polled Bit method
1-May-2009    Several edits
Configure for SCPI LAN using SICL / VISA

**PNA Supported Interfaces**

**Agilent I/O Libraries**

**SICL / VISA Programs Running on the PNA**

**Configure the PNA for SICL / VISA**

**Configure the External Controller**

---

**Other Topics about GPIB Concepts**

**PNA Supported Interfaces**

The PNA supports the following interfaces for SICL / VISA communication:

- **LAN** - as a remote GPIB interface. The PNA LAN is presented as a virtual GPIB interface. It does NOT support simple TCPIP-based control. Therefore, when configuring the Agilent IO libraries on your PC, add a **REMOTE GPIB** interface, which uses the LAN client interface.

- **GPIB** - requires that your external controller have a GPIB card.

**Note:** For optimum LAN interface performance, use **COM** to control the PNA. SCPI commands can be sent to the PNA using the COM **SCPIStringParser** object.

The following interfaces are **NOT** supported:

- **USB**
- **Serial**
Important Note:
To enable VISA or SICL communication over LAN, you must do the following:

1. On the PNA, click **System**, point to **Configure**, then click **SICL/GPIB**.

2. Check **SICL Enabled**. To automatically enable SICL when the PNA is booted, check **Automatically enable on Startup**.

3. Click **OK**.

The PNA is now ready to be controlled over LAN.

Learn more about this dialog box.

Agilent I/O Libraries
The Agilent I/O libraries includes the drivers to allow you to communicate with Agilent test instruments. Every PNA is shipped with the Agilent I/O libraries installed. We recommend you do NOT upgrade the Agilent I/O libraries on the PNA as unexpected results may occur. If you choose to upgrade the Agilent I/O libraries on the PNA, do NOT change the default folder path in the InstallShield Wizard.

To communicate with the PNA, the Agilent I/O libraries must also be installed on your external controller. To purchase the Agilent I/O libraries, or download a free upgrade, go to [www.agilent.com](http://www.agilent.com) and search for IO Libraries. Scroll to find Software, Firmware & Drivers.

SICL / VISA Programs Running on the PNA
You can run your SICL / VISA program on the PNA to control the PNA. Although the Agilent I/O libraries are already installed on the PNA, it is configured as the **Host**. You must also configure a SICL or VISA LAN **Client** interface on the PNA, specifying the LAN hostname of that same PNA.

If your program uses the COM interface to VISA, and is compiled on a PC with the Agilent IO Libraries Suite (version 14 or later), and the resulting executable is copied and run on the PNA, it will produce a “type mismatch error”. This is because the PNA has the ‘M’ version of Agilent I/O libraries. The following Visual Basic code is an example of how to avoid this error when communicating with the PNA from within the PNA:

```vbnet
Dim rm As IResourceManager
Dim fmio As IFormattedIO488
Set rm = CreateObject("AgilentRM.SRMCls")
Set fmio = CreateObject("VISA.BasicFormattedIO")
Set fmio.IO = rm.Open("GPIB0::22")
fmio.WriteString "*IDN?" & Chr(10)
MsgBox fmio.ReadString()
```

Controlling the PNA over LAN while controlling other instruments over GPIB
The PNA can NOT be both a controller and talker/listener on the same GPIB bus. Using SICL / VISA, you can use LAN to control the PNA, leaving the PNA free to use the rear-panel GPIB interface to control other GPIB devices.
Configure the PNA for SICL / VISA

1. On the PNA, click **System** then check **Windows Taskbar**

2. Click **Start** then point to **Program Files, Agilent IO Libraries**, then click **IO Config**

3. Select each GPIB Interface and click **Edit** to verify (or make) the default settings in the following table. These settings are REQUIRED when using a **82357A USB / GPIB** Interface with the PNA.

4. When complete, click **OK** to close the edit dialog.

5. Click **OK** to close the IO Config dialog.

<table>
<thead>
<tr>
<th>VISA Interface Name</th>
<th>SICL Interface Name</th>
<th>Dialog box title</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GPIB0</td>
<td>gpib0</td>
<td>GPIB Using NI-488.2</td>
<td>PNA Rear-panel GPIB connector. This GPIB interface can be used to control the PNA <strong>OR</strong> for the PNA to control external equipment. <strong>IT CAN NOT DO BOTH IN THE SAME PROGRAM. Learn more about pass-through options.</strong></td>
</tr>
<tr>
<td>GPIB1</td>
<td>hpib7</td>
<td>Internal Instrument Configuration</td>
<td>Internal interface for programs running on the PNA to control itself.</td>
</tr>
<tr>
<td>GPIB4</td>
<td>inst0</td>
<td>Internal Instrument Configuration</td>
<td>Used for <strong>LXI compliance. Do NOT delete this interface.</strong></td>
</tr>
</tbody>
</table>

Configure the External Controller

Please refer to the Agilent I/O libraries documentation to learn how to configure your controller to communicate with the PNA. These links can show you how to find the following PNA information:

- **PNA full computer name**
- **GPIB Address**
- **IP Address**

This **example program** can help test your VISA configuration.

Last Modified:

13-Aug-2008 Added GPIB4
Beginning in Dec. 2005, Agilent VEE Pro RunTime is installed on new PNAs. This means that programs written with Agilent VEE (.vxe files) can be run directly on the PNA. New PNA models with 1.1 GHz CPU have VEE 7.5 runtime with Agilent I/O Libraries suite 14.

PNAs without Agilent VEE installed can go to the Agilent VEE website and download Agilent VEE Pro 6.2 RunTime to the PNA and begin to run VEE programs directly on the PNA. This version does not require Agilent I/O Libraries suite 14. Do NOT upgrade to Agilent I/O Libraries suite 14 on the PNA.

With Agilent VEE Pro RunTime installed on the PNA, the following examples can be run directly on the PNA:

- Basic Control of the PNA

For more VEE examples, see the PNA support website.
For more information on Agilent VEE, see www.agilent.com/find/VEE
Basic Control using VEE

This VEE Pro 6.0 example does the following:

- Controls PNA windows and traces.
- Changes stimulus settings.
- Measures all four S parameters.
- Create markers and displays marker readout.

If this Help file is on a PNA and VEE Pro RunTime is installed, then:

1. Run the BasicControl.vxe example
2. Then click Open on the following dialog box to run the program.

Otherwise, you can modify the example program using VEE, save the VEE BasicControl.vee
Learn how to run this program as a Macro on the PNA.

The following dialog box will be visible on the PNA when the example program is running.

- Click Fwd to activate the Forward (S11 and S21) measurements.
- Click Rev to activate the Reverse (S22 and S12) measurements.
- Click Update Markers to sweep the PNA.
- Type values to change Marker Frequencies.
ECal with Confidence Check using VEE

This VEE Pro 6.0 example performs an ECal and subsequent ECal confidence Check. If this Help file is on a PNA and VEE Pro RunTime is installed:

- Run the .vxe example
- Then click Open on the following dialog box to run the program.

Or to modify the example program using VEE, save the VEE BasicControl.vee

Learn how to run this program as a Macro on the PNA.

The following dialog box will be visible on the PNA when the example program is running.

- Click Fwd to activate the Forward (S11 and S21) measurements.
- Click Rev to activate the Reverse (S22 and S12) measurements.
- Click Update Markers to sweep the PNA.
- Type values to change Marker Frequencies.
Interface Control

The Interface Control feature allows you to send remote commands and data to the following PNA rear-panel Interfaces: GPIB, Material Handler I/O, Test Set I/O, and Auxiliary I/O.

- Overview
- How to Access Interface Control Settings
- Interface Control Dialog Box
- Z5623A H08 Test Set Commands

Other System Configuration Topics

Overview

The Interface Control feature allows you to send data to control external equipment such as GPIB instruments, a material handler, test set, or other equipment, without needing to create a remote program. The PNA manages the timing and required interface setup. See Rear Panel Tour.

- A unique set of control data can be sent for each channel. In addition, a unique set of control data can be sent before the channel sweep starts, and after the sweep ends.
- Interface Control settings can be saved and recalled from the Interface Control dialog box, or with Instrument State Save and Recall.
- Interface Control settings can be copied to other channels using Copy Channels.
- Control data can only be WRITTEN to the interfaces, NOT READ from the interfaces.
- Control data is sent in the following order. This order cannot be changed.

1. GPIB Interface
2. Material Handler Interface
3. Test Set Interface
4. Aux Interface
5. Dwell Time
# How to access Interface Control settings

## Using front-panel HARDKEY [softkey] buttons

1. Press **TRACE/CHAN**
2. then **[Channel]**
3. then **[More]**
4. then **[Interface Control]**

## Using a mouse with PNA Menus

1. Click **Trace/Chan**
2. then **Channel**
3. then **More**
4. then **Interface Control**

## Interface Control dialog box help

See Interface Control Overview (scroll up)
An Instrument Preset will reset all of the fields to their default settings.

**Note:** If an error is encountered when sending Interface Control data, an error message is displayed on the PNA screen. The Channel Trigger State is set to Hold. You must fix the condition that caused the error, then change the Channel Trigger State to its original setting.

**Enable Interface Control** Enables and disables ALL Interface Control communication. When cleared (default setting) Interface Control is disabled and NO data is sent. To send data, the individual interfaces must also be enabled.

**Channel** Specifies the channel number for dialog settings. Each channel is configured individually. The list box shows the channels that currently have measurements. There must be at least one measurement present in order to make settings.

**Channel Label** Specifies the label to be displayed on the second status bar at the bottom of the PNA screen. This field is shared with External Testset control. The second status bar is automatically displayed when Interface Control is enabled.

Before Sweep Start - After Sweep End Tabs

Commands / data for all four interfaces can be sent both Before Sweep Start and After Sweep End. However, they are configured and enabled on separate tabs of the Interface Control dialog box. For example, to send GPIB commands both Before and After a PNA sweep, the Enable Control checkbox must be selected and commands entered on BOTH the Before Sweep Start and After Sweep End tabs.

**Before Sweep Start** The data is sent BEFORE the first trace on the channel begins sweeping.

**After Sweep End** The data is sent AFTER the last trace on the channel completes sweeping.

**GPIB Commands**

**Notes:**

- While using the rear-panel GPIB port with Interface Control, the PNA must be in GPIB System Controller mode. If the PNA is NOT in System Controller mode, an error message appears AND Interface Control is disabled. To correct this situation,
  
  1. Put the PNA in System Controller mode, and
  2. Re-enable Interface Control.

- GPIB instruments CAN be connected to the PNA using a USB/GPIB adapter. In this case, the PNA can be in talker-listener mode.

- GPIB Queries are NOT supported.

**Enable Control** Enables and disables sending commands out the GPIB interface.

**Multi-line edit control** Each line contains a GPIB command using the following syntax:

```
address command
```
Where:

- **address** a number between 0 and 31. The PNA will look through all of the GPIB interfaces for an instrument connected to the specified address. If an instrument with that address is not recognized, an error is returned.

- **command** a GPIB command, with or without enclosing quotes. Enclosing quotes are ignored.

Address and command are separated by at least one space.

Commands should be separated by a new line, or carriage return. For example:

```
19 ":init:cont off"
16 init:imm
```

The front-panel **Enter** key inserts a new line into the field.

The number of GPIB commands that can be entered is limited only by the available memory of the PNA. See [Z5326A H08 Test Set Commands](#).

### Material Handler I/O

**Enable Control** Enables and disables sending data out the **Material Handler I/O connector**.

**Ports A, B, C, D** Sends values to the respective Handler I/O port. Although ports C and D are normally bidirectional, ONLY Output mode is allowed using the Interface Control feature. It cannot read from these, or any other, ports.

### Test Set I/O

Note: The PNA has a separate interface for controlling the **E5091A Test Set**.

**Enable Control** Enables and disables sending data out the **External Test Set I/O connector**.

**Multi-line edit control** Each line contains a Write command using the following syntax:

```
address.value
```

Where:

- **address** any positive integer.
- **value** numeric character. Entries that require **alpha** characters should use the **GPIB interface**.

Address and value are separated by a period. For example:

```
18.2
27.3
```

Entries should be separated by a new line, or carriage return. The PNA front-panel **Enter** key inserts a new line into the field.

All entries are sent out the Test Set I/O port using the **WriteData Method**.

The number of entries is limited only by the available memory of the PNA.

### Aux I/O

**Enable Control** Enables and disables sending data out the **Auxiliary I/O connector**.

**DAC1, DAC2** Sets voltages on the Aux I/O connector pins 2 (DAC1) and pin 3 (DAC2).
**Dwell After Command**  Specifies a wait time, in milliseconds, after all commands to all interfaces are sent. Any positive integer is allowed. This is used to allow all external devices to settle before beginning a measurement. An erratic trace could indicate that more settling time is necessary.

**Reset All**  Sets ALL fields on ALL channels to their default values.

**Save and Recall**  Saves and recalls the contents of this dialog box. If the Interface Control dialog box is populated with settings during an Instrument State Save, the settings are automatically recalled with the Instrument State settings.

Interface control uses an *.xml file type. An example file is stored on the PNA hard drive. You can recall it into the dialog, or you can open and edit it with a word processor, such as Word Pad.

**OK**  Applies the settings and closes the dialog box.

**Cancel**  Does not apply changes that were made, and closes the dialog box.

---

**Z5623A H08 Test Set Commands**

The following table lists the commands that are used to control the popular Agilent Z5623A H08 Test Set. These commands can be entered into the GPIB Interface control.

<table>
<thead>
<tr>
<th>Connection Path</th>
<th>Test Set Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reflection to Port 1</td>
<td>refl_01</td>
</tr>
<tr>
<td>Reflection to Port 2</td>
<td>refl_02</td>
</tr>
<tr>
<td>Reflection to Port 3</td>
<td>refl_03</td>
</tr>
<tr>
<td>Reflection to Port 4</td>
<td>refl_04</td>
</tr>
<tr>
<td>Reflection to Port 5</td>
<td>refl_05</td>
</tr>
<tr>
<td>Reflection to Port 6</td>
<td>refl_06</td>
</tr>
<tr>
<td>Reflection to Port 7</td>
<td>refl_07</td>
</tr>
<tr>
<td>Reflection to Port 8</td>
<td>refl_08</td>
</tr>
<tr>
<td>Transmission to Port 1</td>
<td>tran_01</td>
</tr>
<tr>
<td>Transmission to Port 2</td>
<td>tran_02</td>
</tr>
<tr>
<td>Transmission to Port 3</td>
<td>tran_03</td>
</tr>
<tr>
<td>Transmission to Port 4</td>
<td>tran_04</td>
</tr>
<tr>
<td>Transmission to Port 5</td>
<td>tran_05</td>
</tr>
<tr>
<td>Transmission to Port 6</td>
<td>tran_06</td>
</tr>
<tr>
<td>-----------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Transmission to Port 7</td>
<td>tran_07</td>
</tr>
<tr>
<td>Transmission to Port 8</td>
<td>tran_08</td>
</tr>
<tr>
<td>Reset</td>
<td>*rst</td>
</tr>
<tr>
<td>Reflection Termination</td>
<td>*r_term</td>
</tr>
<tr>
<td>Transmission Termination</td>
<td>*t_term</td>
</tr>
<tr>
<td>All Termination</td>
<td>*all_term</td>
</tr>
</tbody>
</table>

Last Modified:

4-Sep-2008   Removed legacy content
**General Description**

This DB-25 male connector provides a variety of analog I/O, digital I/O, timing I/O, and supply lines. You can change the settings on the Auxiliary IO connector through SCPI and COM programming commands. The settings are NOT accessible through the front-panel keys or display menu.

**Note:** This connector does NOT exist on the PNA-X model. Instead, most of this functionality is present on the PNA-X Power IO connector.

**Note:** The AUX IO configuration settings REMAIN after an Instrument Preset and Hibernation. However, Preset will clear the DAC values. The settings will revert to their default settings ONLY after the PNA is restarted, or until they are changed by you. AUX IO settings are saved and recalled with Instrument State.

<table>
<thead>
<tr>
<th>Pin</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ACOM</td>
<td>Ground reference for analog signals</td>
</tr>
<tr>
<td>2</td>
<td>Analog Out 2</td>
<td>-10 to +10Vdc output, 10mA max</td>
</tr>
<tr>
<td>3</td>
<td>Analog Out 1</td>
<td>-10 to +10Vdc output, 10mA max</td>
</tr>
<tr>
<td>4</td>
<td>no connect</td>
<td>for future enhancements</td>
</tr>
<tr>
<td>5</td>
<td>DCOM</td>
<td>Ground reference for digital signals</td>
</tr>
<tr>
<td>6</td>
<td>reserved</td>
<td>for future enhancements</td>
</tr>
<tr>
<td>7</td>
<td>reserved</td>
<td>for future enhancements</td>
</tr>
<tr>
<td>8</td>
<td>reserved</td>
<td>for future enhancements</td>
</tr>
<tr>
<td>9</td>
<td>+5V</td>
<td>+5Vdc output, 100mA max.</td>
</tr>
<tr>
<td>10</td>
<td>Pass/Fail Write Strobe</td>
<td>Indicates pass/fail line is valid (active low)</td>
</tr>
<tr>
<td>11</td>
<td>Sweep End</td>
<td>Indicates sweep is done (programmable modes)</td>
</tr>
<tr>
<td>No.</td>
<td>Description</td>
<td>Notes</td>
</tr>
<tr>
<td>-----</td>
<td>--------------------------------------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>12</td>
<td><strong>Pass/Fail</strong></td>
<td>Indicates pass/fail (programmable logic, modes and scope)</td>
</tr>
<tr>
<td>13</td>
<td><strong>Output Port Write Strobe</strong></td>
<td>Writes I/O port data (active low)</td>
</tr>
<tr>
<td>14</td>
<td><strong>Analog In</strong></td>
<td>-10 to +10VDC analog input</td>
</tr>
<tr>
<td>15</td>
<td><strong>ACOM</strong></td>
<td>Ground reference for analog signals</td>
</tr>
<tr>
<td>16</td>
<td><strong>Power Button In</strong></td>
<td>Grounding replicates front panel power button press</td>
</tr>
<tr>
<td>17</td>
<td><strong>DCOM</strong></td>
<td>Ground reference for digital signals</td>
</tr>
<tr>
<td>18</td>
<td><strong>Ready for Trigger</strong></td>
<td>Indicates ready for external trigger (active low)</td>
</tr>
<tr>
<td>19</td>
<td><strong>External Trigger In</strong></td>
<td>Measurement trigger input (programmable to be active high or low)</td>
</tr>
<tr>
<td>20</td>
<td><strong>Footswitch In</strong></td>
<td>Active low input latches a user-readable status bit.</td>
</tr>
<tr>
<td>21</td>
<td><strong>+22V</strong></td>
<td>+22Vdc output, 100mA max.</td>
</tr>
<tr>
<td>22</td>
<td><strong>In/Out port C0</strong></td>
<td>General purpose input / output</td>
</tr>
<tr>
<td>23</td>
<td><strong>In/Out port C1</strong></td>
<td>General purpose input / output</td>
</tr>
<tr>
<td>24</td>
<td><strong>In/Out port C2</strong></td>
<td>General purpose input / output</td>
</tr>
<tr>
<td>25</td>
<td><strong>In/Out port C3</strong></td>
<td>General purpose input / output</td>
</tr>
</tbody>
</table>

**ACOM (pins 1, 15)**

**Description**

Analog common (ground) - To be used with the Analog Out and Analog In lines.

ACOM and DCOM are connected to system ground at a star ground point inside the analyzer.

---

**Analog Out 1, 2 (pins 2, 3)**

**Description**
Two analog outputs programmable to +/-10V; \( I_{\text{out}} < 10 \text{mA} \); \( R_{\text{out}} = 100 \text{ ohms} \n\)
12-bit DACs with voltage resolution of approximately 5mV/count.

The DACs are set to constant values using SCPI or COM, and can be read using SCPI or COM commands.

Preset state for both pins is 0 volts.

**HW Details**

Looking into this output pin is a 100-ohm series resistor followed by two diodes tied to +/-15V for static protection, then the output or an op-amp.

The voltage output is provided by a 12-bit DAC with an op amp buffer.

Specifics:

- Maximum output current = 10mA
- Settling time = 3us

**Timing**

The DACs are set after the last data point is measured, during retrace. If the analyzer is in single sweep mode, the DACs are set as part of the presweep process, before the sweep is triggered.

---

**DCOM (pins 5, 17)**

**Description**

Digital common (ground).

Used with the digital input and output lines.

ACOM and DCOM are connected to system ground at a star ground point inside the analyzer.

---

**Pins 6, 7, 8**

**Description**

Reserved

---

**+5V (pin 9)**

**Description**

+5V nominal output (100mA max).

Protected by self-healing fuse:

---

**Pass/Fail Write Strobe (pin 10)**

**Description**

See Handler IO connector.

---

**Sweep End (pin 11)**

**Description**
See Handler IO connector.

**Pass/Fail (pin 12)**
**Description**
See Handler IO connector.

**Output Port Write Strobe (pin 13)**
**Description**
See Handler IO connector.

**Analog In (pin 14)**
Beginning with PNA Rev. A.07.21, voltages on this pin can be read using ADC receiver measurements. Learn more.
**Description**
Analog input, +/-10V range, Rin=100k ohm
Bandwidth = 40kHz (2-pole lowpass filter).
This analog input may be read using the SCPI or COM commands.
**HW Details**
Looking into this pin there is 1k-ohm series resistor followed by 100k-ohm resistor to ground, static protection diodes after the 1k resistor limit the signal to +/-15V, then a high impedance buffer and active filter limiting the bandwidth to 40kHz with a lowpass filter.

**Power Button In (pin 16)**
**Description**
Short this pin to ground to replicate a front panel power button key press.
**HW Details**
Looking into the pin there is a 215-ohm series resistor followed by a 10k pull-up to the 3V standby supply, static protection diodes to the 0V/5V and then connects to the front panel power key circuit.
CAUTION: Because this line is internally pulled up to 3V, it should not be driven by a TTL driver.
**Timing**
Grounding this line for 1us to 2 seconds will simulate pressing the front panel power button.
Grounding this line for >4 seconds will perform a hard reset (similar to a personal computer) and is not recommended.

**Ready for Trigger (pin 18)**
**Description**
TTL output.
Active Low signal indicates that system is ready for an external trigger.
Remains High if system is not in External Trigger mode.
Goes High after an External Trigger is acknowledged.
Goes Low after the system has finished with its measurements, the source has been set up, and the next data point is ready to be measured.

**HW Details**
Looking into this pin there is a 215-ohm series resistor followed by a 10k pullup, diodes to 0V/5V for static protection, then the output of an "ABT" TTL buffer.
This line is enabled only when the analyzer is in External Trigger mode.
Refer to External Trigger In (following pin) for more information.

**Timing**
Refer to [External Trigger In](following pin)

---

**External Trigger In (pin 19)**

**Description**
This input accepts level trigger signals (High / Low) on all PNA models, or edge trigger signals on some PNA models.
The external trigger configuration is set from the front panel, [SCPI](#) or [COM](#).
For more information, see [External triggering](#).
A single trigger is achieved by asserting the external trigger for a period from 1us to 50us. Continuous triggering is achieved by holding the external trigger in the "asserted" mode (either Low or High).

**HW Details**
Looking into this pin is a 215-ohm series resistor followed by a 4.64k pullup, 1000pF to ground and then a "FAST" TTL buffer input.

**Timing**
A level trigger width should be between 1us and 50us.

---

**Footswitch In (pin 20)**

**Description**
TTL input.
A Low level input such as shorting this line to ground using a footswitch (where the input stays low for >1us) will be latched.
The latched status may be read using the [SCPI](#) or [COM](#) commands.
Only one footswitch press can be latched (remembered) by the system.
Reading the latch status will reset it if Footswitch In has returned to a high level.
**HW Details**

Looking into this pin is a 215-ohm series resistor followed by a 4.64k pullup to 5V and 1000pF to ground. This line is an input to a "FAST" TTL buffer.

**Timing**

Footswitch In must be Low for at least 1us.

---

**+22V (pin 21)**

**Description**

+22V nominal output (100mA max).

Protected by self-healing fuse.

---

**In/Out Port C0-C3 (pins 22-25)**

**Description**

See [Handler IO connector](#).

---

Last Modified:

23-Mar-2009    Added links to ADC Meas from pin 14
External Test Set I/O Connector

General Description

This DB-25 female connector is used to control external test sets. The external test set bus consists of 13 multiplexed address and data lines, three control lines, and an open-collector interrupt line. The Test Set IO is not compatible with the 8753 test sets.

You can change the settings on the External Test Set IO connector through SCPI and COM programming commands. The settings are NOT accessible through the front-panel keys or display menu.

Notes:

- The External Test Set pin settings are NOT affected by Instrument State Save/Recall or Instrument Preset.
- At PNA Power Up and return from Hibernation, the External Test Set bus data lines, address lines, and control lines are set HIGH, and no strobe lines are pulsed.

Caution: Do not mistake this connector with a Parallel Printer port. A printer may be damaged if connected to this port.

Other System Configuration Topics

<table>
<thead>
<tr>
<th>Pin</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>SEL0</td>
<td>Test set select bit 0; tied to GND</td>
</tr>
<tr>
<td>2</td>
<td>Sweep Holdoff In</td>
<td>TTL input - state may be read with SCPI or COM command</td>
</tr>
<tr>
<td>3</td>
<td>AD12</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>4</td>
<td>AD10</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>5</td>
<td>AD9</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>6</td>
<td>AD8</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>7</td>
<td>GND</td>
<td>0V</td>
</tr>
<tr>
<td>8</td>
<td>LAS</td>
<td>TTL output Low = Address Strobe</td>
</tr>
<tr>
<td>Pin</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>-----</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>AD4</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>10</td>
<td>AD3</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>11</td>
<td>AD2</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>12</td>
<td>GND</td>
<td>0V</td>
</tr>
<tr>
<td>13</td>
<td>Interrupt In</td>
<td>TTL input - state may be read with a SCPI or COM command</td>
</tr>
<tr>
<td>14</td>
<td>No connect</td>
<td>CAUTION: Older PNAs have +22V on this line; this will damage a printer.</td>
</tr>
<tr>
<td>15</td>
<td>SEL1</td>
<td>Test set select bit 1; tied to GND</td>
</tr>
<tr>
<td>16</td>
<td>SEL2</td>
<td>Test set select bit 2; tied to GND</td>
</tr>
<tr>
<td>17</td>
<td>AD11</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>18</td>
<td>SEL3</td>
<td>Test set select bit 3; tied to GND</td>
</tr>
<tr>
<td>19</td>
<td>AD7</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>20</td>
<td>AD6</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>21</td>
<td>AD5</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>22</td>
<td>AD0</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>23</td>
<td>AD1</td>
<td>Address and latched data</td>
</tr>
<tr>
<td>24</td>
<td>LDS</td>
<td>TTL output - active low data strobe</td>
</tr>
<tr>
<td>25</td>
<td>RLW</td>
<td>TTL output - high-read, low write</td>
</tr>
</tbody>
</table>

**SEL0-SEL3 (pins 1,15,16,18)**

**Description**
Selects addresses of test sets that are "daisy chained" to this port. The select code is set to zero at the PNA connector and is incremented by one as it goes through each successive external test set. Therefore, the first test set in the chain has address zero and so on, for up to 16 test sets.

**HW Details**
Connected to ground inside the PNA.

**Timing**
None

**Sweep Holdoff In (pin 2)**

**Description**
Input line used by the test set for holding off a sweep. Holding off a sweep is one way of introducing a delay that allows an external device to settle before the PNA starts taking data. You must write a program that will query the line and perform the delay. The program needs to query the line and keep PNA from sweeping while the line remains low. When a subsequent query detects that the line went high the program would then trigger the PNA to start the sweep.

Use either Single or External trigger mode to control the PNA sweep.

**HW Details**

This pin has a series 215-ohms resistor followed by 4.7k-ohm pull-up and then an “ABT” TTL buffered register.

**Timing**

This input is not latched by the PNA hardware. Therefore the input level must be held at the desired state by the test set until it’s read by your program.

---

**AD0-AD12 (pins 3-6, 9-11, 17, 19-23)**

**Description**

Thirteen lines are used to output data addresses or input / output data. Several SCPI and COM commands are available for reading and writing to these lines. You can choose to use commands where the PNA provides the appropriate timing signals needed for strobing the addresses and data. Or you can choose to control the timing signal directly. The timing signals are RLW, LAS and LDS. If you decide to do direct control refer to the corresponding SCPI and COM command details. Close attention to detail is needed to insure the desired results.

After a write command, lines AD0-AD12 are left in the state they were programmed. Default setting for Mode is Read / Input).

After a read command, lines AD0-AD12 are left in input mode. While in this mode an external test set attached to the IO is free to set the level on each line.

**HW Details**

Each of these I/O pins has a series 215-ohm resistor followed by 4.7k-ohm pull-up resistor.

Write/Read is implemented by an output tri-state TTL buffer / latch for latching and enabling write data in parallel with a TTL input buffer for reading.

**Timing**

Output Address and data setup and hold times are 1us minimum.

---

![Timing Diagram](image)

**Address & Data I/O Write**
**Address & Data I/O Read** - Data must be valid for 1us before and after strobe

---

**GND (pins 7, 12)**

**Description**
Two ground pins used as ground references by the test set.

**HW Details**
Connected to digital ground.

**Timing**
None.

---

**LAS (Low Address Strobe) (pin 8)**

**Description**
This line has two behaviors that are command dependent. Refer to the SCPI and COM commands for further details.

In one behavior LAS is one of the lines used by the PNA to provide appropriate timing for writing Address and Data to the Test Set. In this case LAS is controlled automatically by the PNA and is intended to be used as the strobe for the Address. When LAS is low, lines AD0 - AD12 represent the Address. LAS will return to its normally high state when the transaction is finished.

In the second behavior the PNA will NOT provide appropriate timing. In this case LAS is controlled directly by the user through a SCPI or COM command. When the transaction is finished LAS is left set to the state it was programmed to until another command changes it. (Default for LAS is TTL High).

**HW Details**
This output pin is driven by a TTL latched buffer with a series 215-ohm resistor followed by 2.15k-ohm pull-up.

**Timing**
Strobe length, setup and hold times are all 1us minimum.
See the description for AD0-AD12 for more timing information.

---

**Interrupt In (pin 13)**

**Description**
Query this line with a SCPI or COM command.

**HW Details**
This line is a non-latched TTL input, has series 215-ohms followed by 4.64k-ohm pullup.

**Timing**
The Test Set must maintain at the desired TTL level until its read.

---

**LDS (Low Data Strobe) (pin 24)**

**Description**

This line has two behaviors that are command dependent. Refer to the External Test Set IO SCPI and COM commands for further details. (Default setting for LDS is TTL High)

In one behavior LDS is one of lines used by the PNA to provide appropriate timing for writing Address and Data to the Test Set. In this case LDS is controlled automatically by the PNA and is intended to be used as the strobe for the Data. When LDS is low, lines AD0 - AD12 represents Data. LDS will return to its normally high state when the transaction is finished.

In the second behavior the PNA will NOT provide appropriate timing. In this case LDS is controlled directly by the user through a SCPI or COM command. When the transaction is finished the LDS is left set to the state it was programmed to.

**HW Details**

This output pin is driven by a TTL latched buffer with a series 215-ohm resistor followed by 2.15k-ohm pull-up.

**Timing**

Strobe length, setup and hold times are all 1us minimum.

See the description for AD0-AD12 for more timing information.

---

**RLW (pin 25)**

**Description**

This line is the output for the Read Write signal. It has two behaviors that are command dependent. Refer to the External Test Set IO SCPI and COM commands for further details. (Default setting for RLW is TTL High)

---

**Warning:** Connecting a printer to this port will usually damage the printer.
In one behavior RWL is controlled automatically by the PNA during a Read Write operation. When RLW is low, lines AD0 - AD12 represent output Data. When RLW is high, the lines represent input Data.

In the second behavior the PNA does NOT provide the timing. The user must control it directly through the SCPI or COM command. In this case the line is left set to the state it was programmed to.

**HW Details**

This pin is a TTL latched output with a series 215-ohm resistor followed by 2.15k-ohm pull-up resistor.

**Timing**

Strobe length, setup and hold times are all 1us minimum.

See the description for **AD0-AD12** for more timing information.
Material Handler I/O Connector

This rectangular 36-pin female connector provides communication signals between the PNA and a material parts handler. You can change the settings on the Material Handler IO connector using SCPI and COM commands. The settings are NOT accessible through the front-panel keys or display menu.

- **Overview - Controlling a Material Handler**

- **Pin Assignments**

- **Pin Descriptions**

- **Timing Diagrams**

- **Input Output Electrical Characteristics**

**Note:** On early PNAs this connector is labeled "GPIO". It is covered to indicate that the connector is not functional.

**Overview - Controlling a Material Handler**

The PNA is capable of interacting with an external material handler or part handler. This allows the PNA to be used in an automated test environment, where devices to be tested are inserted into a test fixture by a part handler, and sorted into pass/fail bins by the handler after testing is complete. By connecting the part handler to the PNA Auxiliary or Material Handler I/O ports, the PNA and part handler can synchronize their activities in a way that makes automated testing possible.

**PNA and Part Handler Preparation**

1. **Define the measurements** you want to make.

2. **Define limits** for each of the measurements.

3. Configure the PNAs Material Handler port so that it is compatible with your part handler. This usually involves setting the **handler logic**, **pass/fail logic**, **pass/fail scope**, and **pass/fail mode**. These settings are made remotely using SCPI or COM commands.

4. Use a cable to connect the PNA to your part handler.

5. Put the PNA in **External Trigger** mode.

6. Load parts in handler per manufacturer instructions.

**Note:** The Material Handler configuration settings REMAIN after an Instrument Preset and Hibernation. The settings will revert to their default settings ONLY after the PNA is restarted, or until they are changed by you. Material Handler settings are saved and recalled with Instrument State.

**Flow Diagram**

The following diagram and descriptions summarizes the events that occur during automated testing. 'DUT' refers to Device Under Test.
Text Descriptions

0. (Optional). The PNA sends values out the Material Handler and/or Auxiliary I/O connectors to configure external instruments. The A,B,C, and D ports of the Material Handler can be used to control devices used in testing, such as step attenuators, part handlers, or even the DUT itself. Also, the DAC1 and DAC2 lines on the Aux I/O connector can be used to provide bias voltages for devices and instruments. If you wish to use the Material Handler or Aux connectors for testing, you will need to write a program to send values out the various lines and ports, as there is no activity on these lines by default.

1. The part handler receives a Ready for Trigger signal from the PNA. This indicates that the PNA is properly configured and ready to take a measurement.

2. The part handler sends an External Trigger signal to the PNA. This signals that the part handler has settled, and allows the PNA to begin taking measurements.

3. The PNA takes measurements on all triggerable channels.

4. The Index line on the material handler goes to a Low state, which means that all required data has been collected by the PNA.

5. The part handler removes the DUT from the test fixture, and inserts a new DUT into the fixture. This operation is often referred to as part handler indexing. The device just tested is staged (removed from the fixture and prepared for binning), and the next part to be tested is put into the fixture. The removed DUT cannot be assigned to a Pass/Fail bin yet, as the Pass/Fail status is not available.

6. The PNA sends the Pass/Fail Status.

7. The PNA sends the Pass/Fail Strobe meaning that the Pass/Fail status has been determined.
8. The part handler reads the Pass/Fail Status line.

9. The part handler bins the staged part based on the Pass/Fail Status.

10. The test process repeats at step 1, waiting for Ready for Trigger from the PNA.

### Material Handler IO Pin Assignments

There are three different Handler IO pin assignment configurations depending on the PNA model:

- **Type 1** - All E835xA, E880xA, N338xA models. You can change the pinout configuration to Type 2 on these models. This requires opening the instrument and changing a connector internally. Refer to the procedure in the Service Guide, Chapter 7. You can download the Service Guide for your PNA model from our website: [http://na.tm.agilent.com/pna](http://na.tm.agilent.com/pna)  **Caution:** Changing this connection should be done by qualified service personnel.

- **Type 2** - E8362x, E8363x, E8364x and N5230 Opt 220, 225, 420, 425, 520, 525;

- **Type 3** - All PNA-X and N5230 Opt 020, 025, 120, 125, 140, 145, 146, 245, 240, 245, 246.

See PNA models and options.

Shaded/bold indicates changes from Type 1

**Note:** A slash (/) preceding the signal names indicates that the signal uses negative (active low) logic. A low pulse is a logical 1.

<table>
<thead>
<tr>
<th>Pin</th>
<th>Type 1</th>
<th>Type 2</th>
<th>Type 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Ground</td>
<td>Ground</td>
<td>Ground</td>
</tr>
<tr>
<td>2</td>
<td>/ INPUT1</td>
<td>/ INPUT1</td>
<td>/ INPUT1</td>
</tr>
<tr>
<td>3</td>
<td>/ OUTPUT1</td>
<td>/ OUTPUT1</td>
<td>/ OUTPUT1</td>
</tr>
<tr>
<td>4</td>
<td>/ OUTPUT2</td>
<td>/ OUTPUT2</td>
<td>/ OUTPUT2</td>
</tr>
<tr>
<td>5</td>
<td>/ Output port A0</td>
<td>/ Output port A0</td>
<td>/ Output port A0</td>
</tr>
<tr>
<td>6</td>
<td>/ Output port A1</td>
<td>/ Output port A1</td>
<td>/ Output port A1</td>
</tr>
<tr>
<td>7</td>
<td>/ Output port A2</td>
<td>/ Output port A2</td>
<td>/ Output port A2</td>
</tr>
<tr>
<td>9</td>
<td>/ Output port A4</td>
<td>/ Output port A4</td>
<td>/ Output port A4</td>
</tr>
<tr>
<td>10</td>
<td>/ Output port A5</td>
<td>/ Output port A5</td>
<td>/ Output port A5</td>
</tr>
<tr>
<td></td>
<td>Port C Status</td>
<td>Port D Status</td>
<td>Port C Status</td>
</tr>
<tr>
<td>---</td>
<td>---------------</td>
<td>---------------</td>
<td>---------------</td>
</tr>
<tr>
<td>11</td>
<td>/ Output port A6</td>
<td>/ Output port A6</td>
<td>/ Output port A6</td>
</tr>
<tr>
<td>12</td>
<td>/ Output port A7</td>
<td>/ Output port A7</td>
<td>/ Output port A7</td>
</tr>
<tr>
<td>13</td>
<td>/ Output port B0</td>
<td>/ Output port B0</td>
<td>/ Output port B0</td>
</tr>
<tr>
<td>14</td>
<td>/ Output port B1</td>
<td>/ Output port B1</td>
<td>/ Output port B1</td>
</tr>
<tr>
<td>15</td>
<td>/ Output port B2</td>
<td>/ Output port B2</td>
<td>/ Output port B2</td>
</tr>
<tr>
<td>16</td>
<td>/ Output port B3</td>
<td>/ Output port B3</td>
<td>/ Output port B3</td>
</tr>
<tr>
<td>17</td>
<td>/ Output port B4</td>
<td>/ Output port B4</td>
<td>/ Output port B4</td>
</tr>
<tr>
<td>18</td>
<td>no connect</td>
<td>/ Output port B5</td>
<td>/ Ext. Trigger</td>
</tr>
<tr>
<td>19</td>
<td>/ Output port B5</td>
<td>/ Output port B6</td>
<td>/ Output port B5</td>
</tr>
<tr>
<td>20</td>
<td>/ Output port B6</td>
<td>/ Output port B7</td>
<td>/ Output port B6 -or / Index Signal Learn more</td>
</tr>
<tr>
<td>21</td>
<td>/ Output port B7</td>
<td>/ In/Out port C0</td>
<td>/ Output port B7 or / Ready for Trigger Learn more</td>
</tr>
<tr>
<td>22</td>
<td>/ In/Out port C0</td>
<td>/ In/Out port C1</td>
<td>/ In/Out port C0</td>
</tr>
<tr>
<td>23</td>
<td>/ In/Out port C1</td>
<td>/ In/Out port C2</td>
<td>/ In/Out port C1</td>
</tr>
<tr>
<td>24</td>
<td>/ In/Out port C2</td>
<td>/ In/Out port C3</td>
<td>/ In/Out port C2</td>
</tr>
<tr>
<td>25</td>
<td>/ In/Out port C3</td>
<td>/ In/Out port D0</td>
<td>/ In/Out port C3</td>
</tr>
<tr>
<td>26</td>
<td>/ In/Out port D0</td>
<td>/ In/Out port D1</td>
<td>/ In/Out port D0</td>
</tr>
<tr>
<td>27</td>
<td>/ In/Out port D1</td>
<td>/ In/Out port D2</td>
<td>/ In/Out port D1</td>
</tr>
<tr>
<td>28</td>
<td>/ In/Out port D2</td>
<td>/ In/Out port D3</td>
<td>/ In/Out port D2</td>
</tr>
<tr>
<td>29</td>
<td>/ In/Out port D3</td>
<td>Port C Status</td>
<td>/ In/Out port D3</td>
</tr>
<tr>
<td>30</td>
<td>Port C Status</td>
<td>Port D Status</td>
<td>Port C Status</td>
</tr>
<tr>
<td>31</td>
<td>Port D Status</td>
<td>/ Output Port Write Strobe</td>
<td>Port D Status</td>
</tr>
<tr>
<td>32</td>
<td>/ Output Port Write Strobe</td>
<td>no connect</td>
<td>/ Output Port Write Strobe</td>
</tr>
<tr>
<td>33</td>
<td>/ Pass/Fail</td>
<td>/ Pass/Fail</td>
<td>/ Pass/Fail</td>
</tr>
<tr>
<td>34</td>
<td>/ Sweep End</td>
<td>+5V</td>
<td>/ Sweep End</td>
</tr>
</tbody>
</table>
Pin Descriptions

**Input1**
When this Input line receives a Low pulse from the material handler, data is latched on the OUTPUT1 and OUTPUT2 lines. See OUTPUT1|2 Data Output Write Timing

**Note: Type 1 and Type 2 Behavior:** The Input line responds to a High (rising edge) pulse.

The Input Line activity can be read:

```
SCPI                          COM
CONTro1:HANDle:INPut?
```

The current state of these latched TTL outputs may be set High or Low (Default setting) using the (non-user) SCPI put_Output (COM) commands.

The next state (following a negative edge on the INPUT1 line) may be pre-loaded to High or Low (Default setting) using the user commands.

For example, on the next negative pulse on the INPUT1 line, you want the OUTPUT1 line to go from 0 to 1. To do this:

```
CONT:HAND:OUTP1:DATA 0 'Force the OUTPUT1 line to 0
CONT:HAND:OUTP1:USER 1 'Set the OUTPUT1:USER buffer to 1, indicating the next state
```

**Output1, Output2**
See OUTPUT1|2 Data Output Write Timing

The current state of these latched TTL outputs may be set High or Low (Default setting) using the user commands.

For example, on the next negative pulse on the INPUT1 line, you want the OUTPUT1 line to go from 0 to 1. To do this:

```
CONT:HAND:OUTP1:DATA 0 'Force the OUTPUT1 line to 0
CONT:HAND:OUTP1:USER 1 'Set the OUTPUT1:USER buffer to 1, indicating the next state
```

```
SCPI                          COM
Write User Data               put_Output Method
CONT:HAND:OUTP<pin>:USER
Read last value written       get_Output Method
CONT:HAND:OUTP<pin>:USER
Write non-user data            put_Output Method
CONT:HAND:OUTP<pin>:DATA
Read last value written       get_Output Method
CONT:HAND:OUTP<pin>:DATA
```
Output Ports A and B
These two general purpose, 8-bit output ports are used to write data to the material handler. When any line changes state, all output lines are latched to the I/O connector as the Output Write Strobe goes Low.

The default state for data is Low.

See Data Output Write Timing Diagram

Set Port Logic:
The logic for the data lines can be set to either: Positive (1 = High) or Negative (1 = Low). This setting affects all data ports. They cannot be set independently.

```
SCPI
CONTrol:HANDler:LOGic
COM
PortLogic Property
```

Combine to read or write data to Port F:
Ports A and B can be virtually combined to write data to one 16-bit I/O port F.

```
SCPI
CONTrol:HANDler:F <num>
COM
put Port (F)
```

Input/Output Ports C and D
These two general purpose 4-bit Input/Output ports are used to write data (Output) or read data (Input). These lines could be used to write to an external device such as a step attenuator.

When any line changes state, all output lines are latched to the I/O connector as the Output Write Strobe goes Low. See Data Output Write Timing

The four lines of Port C are connected internally to the Auxiliary IO connector.

Set Input | Output Mode:
Each port may be independently defined as Output or Input.

```
SCPI
CONTrol:HANDler:C:MODE
COM
PortMode Property

CONTrol:HANDler:D:MODE
```
Set Port Logic:
The logic for the data lines can be set to either: Positive (1 = High) or Negative (1 = Low). This setting affects all data ports. They cannot be set independently.

SCPI

CONTrol:HANDler:LOGic

COM

PortLogic Property

Read or write data:
Ports C and D can be virtually combined to read or write data to one 8-bit I/O port E. When combined, both C and D ports must be set to either INPUT or OUTPUT mode.

SCPI

CONTrol:HANDler:<port>[:DATa]>

COM

gget Port(x)

put Port(x)

Port C Status, Port D Status
These two output lines indicate the Read / Write mode of the C and D ports.

- A Low level indicates that the associated port is in INPUT mode (read only).
- A High level indicates that the associated port is in OUTPUT mode (write only).

These logic of these status outputs cannot be changed.
See Input/Output Ports C and D to learn how to set I/O Mode
See Data Output Write Timing

Output Port Write Strobe
This Output line goes Low to write data from Ports A and B and Ports C and D when a change is detected on any of the data lines.
These logic of this strobe output cannot be changed.
This line is shared with Auxiliary IO connector.
See Data Output Write Timing
**External Trigger**

When trigger source is set to external, this Input line accepts a trigger signal from the material handler. This usually means that a part is in place and ready to be tested.

*See Trigger Timing Diagram*

---

**Index**

A Low signal on this Output line indicates to the material handler that the measurement is complete. This usually means that the handler can connect the next device. However, measurement data is not available until data is calculated. *See Trigger Timing Diagram*.

**Set Function:**

This line also serves as a data line. Set the function using the following commands:

```
SCPI

CONTrol:HANDler:INDex:STATe

COM

IndexState
```

---

**Ready for Trigger**

When this output line goes low, it indicates to the material handler that the PNA is ready for a trigger signal. Available only on Type 3 PNA Models (Pin 17). See also *Aux IO connector*.

*See Trigger Timing Diagram*

*See Pass/Fail Timing Diagram*

**Set Function:**

This line also serves as a data line. Set the function using the following commands:

```
SCPI

CONTrol:HANDler:RTRigger:STATe

COM

ReadyForTriggerState
```
Pass/Fail State
This Output line indicates to the handler whether the limit test has passed or failed.
Pass/Fail state is valid only when the limit test function is ON and while Pass/Fail strobe line is Low. See Pass/Fail Timing Diagram
This line is shared with the Auxiliary IO connector.

Set Pass / Fail Logic:

- Positive Logic: High=Pass, Low=Fail. (Default setting)
- Negative Logic: High=Fail, Low=Pass.

SCPI

CONTrol:HANDler:PASSfail:LOGic

PassFailLogic Property

Set Default Conditions:

- **PASS**- the line stays in PASS state. When a device fails, then the line goes to fail after the Sweep End line is asserted.
- **FAIL**- the line stays in FAIL state. When a device passes, then the line goes to PASS state after the Sweep End line is asserted.
- **No Wait**- the line stays in PASS state. When a device fails, then the line goes to fail IMMEDIATELY. (Default setting)

SCPI

CONTrol:HANDler:PASSfail:MODE

PassFailMode Property

Set Pass / Fail Scope:

- **Channel scope**: The line resets to the default state after the measurements on a channel have completed.
- **Global scope**: The line resets to the default state after the measurements on all triggerable channels have completed. (Default setting)

SCPI

CONTrol:HANDler:PASSfail:SCOPe

PassFailScope Property
**Pass/Fail Write Strobe**
A Low pulse indicates that Pass/Fail line is valid and the Pass / Fail State is output to the material handler. This line is shared with the Auxiliary IO connector.

The Pass/Fail Strobe is fixed in duration and timing. However, when the strobe occurs depends on the Pass/Fail Mode and Pass/Fail Scope (Channel or Global) settings. See Pass/Fail State

See Pass/Fail Timing Diagram

---

**+5V**
+5V nominal output (100mA max).
Protected by self-healing fuse.

---

**Sweep End**
This output line indicates the status of the PNA sweep. The sweep includes sweeping the source and taking data.

- **Low** (falling edge) indicates that the specified sweep event has finished. This does NOT indicate that all calculations have finished.
- **High** indicates that the specified sweep event is active.

See Trigger Timing Diagram
This line is shared with the Auxiliary IO connector.

**Set Sweep Event Mode:**

- **Sweep**: indicates that a single source sweep has finished. (Default setting)
- **Channel**: indicates that a single channel has finished.
- **Global**: indicates that all enabled channels have finished.

```scpi
CONTrol:HANDler:SWEepend
```

SweepEndMode Property
**Timing Diagrams**

**Trigger Timing**

- **/ External Trigger (to PNA)**
- **/ Index (from PNA) meas. complete**
- **/ Sweep End (from PNA) sweep complete**
- **/ Ready for Trigger (from PNA)**

All signals are active low.

**T1 = 1 μs**  External Trigger pulse width

**T2 > 10μs**  Sweep End pulse width (both High and Low)

**Pass / Fail Timing**

- **Pass / Fail Status (from PNA)**
- **/ Pass /Fail Write Strobe (from PNA)**
- **/ Ready for Trigger (from PNA)**

**T1 = 1 μs**  Pulse width and response time of Pass / Fail Strobe

**T2 > 10 μs**  Ready for Trigger lag
Ports A-F Data Output Write Timing

\[ T1 = 1 \, \mu s \] Write Strobe response time
\[ T2 = 1 \, \mu s \] Write Strobe pulse width

OUTPUT1|2 Data Output Write Timing

The old state to new state transition can be either low to high (as shown) or high to low.
\[ T1 = 0.6 \, \mu s \] Output1|2 response time
\[ T2 = 1 \, \mu s \] Input1 Strobe pulse width

Input / Output Electrical Characteristics

All Material Handler I/O Input and Output lines are TTL compatible.

Input and Output lines

Lines carrying information IN (or bidirectional) to the PNA from the material handler.

<table>
<thead>
<tr>
<th>Maximum Input Voltages:</th>
<th>-0.5 V to 5.5 V</th>
</tr>
</thead>
<tbody>
<tr>
<td>TTL High level:</td>
<td>2.0 V to 5.0 V</td>
</tr>
<tr>
<td>TTL Low level:</td>
<td>0 V to 0.5 V</td>
</tr>
</tbody>
</table>
PNA Input and Input/Output Circuit Diagram

![Circuit Diagram]

To Handler I/O Port

Note: The INPUT1 line does NOT have the 10K pullup resistor.

**Output Lines**

Lines carrying information OUT of the PNA to the material handler.

<table>
<thead>
<tr>
<th></th>
<th>Maximum Output Current: -10 mA to 10 mA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Output Current</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TTL High level: -5 mA</td>
</tr>
<tr>
<td></td>
<td>TTL Low level: 3 mA</td>
</tr>
<tr>
<td>Output Voltage</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TTL High level: 2.0 V to 3.3 V</td>
</tr>
<tr>
<td></td>
<td>TTL Low level: 0 V to 0.8 V</td>
</tr>
</tbody>
</table>

PNA Output Circuit Diagram

![Circuit Diagram]

To Handler I/O Port

---

Last modified:

6-Apr-2009  Replaced N5242A with PNA-X
8-Feb-2008  Clarified Types
Nov 3, 2006  Added link from pin 20
The PNA-X and N522x models offer the 15 pin D connector provides access to Pulse Modulators and Generators.

- See IF Path Configuration Dialog and block diagram, which includes the Pulse Modulators and Generators.
- See the Pulsed Application (Opt H08 / 008)

<table>
<thead>
<tr>
<th>Pin</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>IFGateAIn</td>
<td>IF pulse gate input A (TTL)</td>
</tr>
<tr>
<td>2</td>
<td>IFGateBIn</td>
<td>IF pulse gate input B (TTL)</td>
</tr>
<tr>
<td>3</td>
<td>IFGateCIn</td>
<td>IF pulse gate input C (TTL)</td>
</tr>
<tr>
<td>4</td>
<td>IFGateDIn</td>
<td>IF pulse gate input D (TTL)</td>
</tr>
<tr>
<td>5</td>
<td>IFGateRIn</td>
<td>IF pulse gate input R (TTL)</td>
</tr>
<tr>
<td>6</td>
<td>DCOM</td>
<td>Ground</td>
</tr>
<tr>
<td>7</td>
<td>PulseSyncIn</td>
<td>Pulse gen. synchronization trigger input (TTL)</td>
</tr>
<tr>
<td>8</td>
<td>RFPulseModIn</td>
<td>RF source pulse modulation drive input (TTL)</td>
</tr>
<tr>
<td>9</td>
<td>DCOM</td>
<td>Ground</td>
</tr>
<tr>
<td>10</td>
<td>Pulse1Out</td>
<td>Hardwired pulse train output #1 (TTL)</td>
</tr>
<tr>
<td>11</td>
<td>Pulse2Out</td>
<td>Hardwired pulse train output #2 (TTL)</td>
</tr>
<tr>
<td>12</td>
<td>Pulse3Out</td>
<td>Hardwired pulse train output #3 (TTL)</td>
</tr>
<tr>
<td>13</td>
<td>Pulse4Out</td>
<td>Hardwired pulse train output #4 (TTL)</td>
</tr>
<tr>
<td>14</td>
<td>N.C.</td>
<td>No connect -- for future use</td>
</tr>
<tr>
<td>15</td>
<td>DCOM</td>
<td>Ground</td>
</tr>
</tbody>
</table>

See Pulse SCPI and COM commands
N1966A Pulse I/O Adapter

This D connector to RF adapter makes accessing the Pulse I/O connector more convenient.

Last Modified:

4-May-2011  Added N522x
16-Jul-2007  Clarification
18-Jan-2007  MX New topic
**Power I/O Connector**

The 9-pin D connector replaces much of the functionality of the AUX I/O connector on older PNA models. See Rear Panel

![Power I/O Connector Image]

<table>
<thead>
<tr>
<th>Pin</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+15V</td>
<td>+15V @ 400 mA</td>
</tr>
<tr>
<td>2</td>
<td>-15V</td>
<td>-15V @ 400 mA</td>
</tr>
<tr>
<td>3</td>
<td>AnalogOut1</td>
<td>Analog Output Voltage</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Programmable +/-10V @100 mA out</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Nominally 0 ohms</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.44mV typical resolution</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1MHz BW</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Read and write voltage programmatically using:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• CONT:AUX:OUTP1:VOLT (SCPI - read and write)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• get OutputVoltage Method (COM - read)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• put OutputVoltage Method (COM - write)</td>
</tr>
<tr>
<td>4</td>
<td>AnalogOut2</td>
<td>Analog Output Voltage</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Programmable +/-10V @100 mA out</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Nominally 0 ohms</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.44mV typical resolution</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1MHz BW</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Read and write voltage programmatically using:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• CONT:AUX:OUTP2:VOLT (SCPI - read and write)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• get OutputVoltage Method (COM - read)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• put OutputVoltage Method (COM - write)</td>
</tr>
<tr>
<td>5</td>
<td>ACOM</td>
<td>System ground</td>
</tr>
<tr>
<td>6</td>
<td>GndSense</td>
<td>Ground sense for Analog In &amp; Out</td>
</tr>
</tbody>
</table>
| 7  | AnalogIn1 | Analog input:  
 +/-10V @ 1.22mV typical resolution  
 Rin >1 M-ohm  
 BW ~ 1 MHz  
 ADC conversion time < 1 us typical  
 Read voltage programmatically using:  
  - `CONT:AUX:INP1:VOLT` (SCPI)  
  - `InputVoltageEX Property` (COM) |
|----|-----------|--------------------------------------------------|
| 8  | AnalogIn2 | Analog input:  
 +/-10V @ 1.22mV typical resolution  
 Rin >1 M-ohm  
 BW ~ 1 MHz  
 ADC conversion time < 1 us typical  
 Read voltage programmatically using:  
  - `CONT:AUX:INP2:VOLT` (SCPI)  
  - `InputVoltageEX Property` (COM) |
| 9  | Power Button | Open collector input  
 Active low replicates power button key press. |

Connected with 51.1-ohm to ACOM
Read voltage programmatically using:

- `InputVoltageEX Property` (COM)
- `CONT:AUX:OUTP3:VOLT` (SCPI)

Voltages on this pin can also be read using ADC receiver measurements. [Learn more.](#)
4-May-2011   Modified for N522x
23-Mar-2009   Added links to ADC meas.
13-Aug-2008   Edited for InputVoltageEx
10-Jul-2007   Added COM commands
18-Jan-2007   MX New topic
# New Programming Commands

The following are new programming commands for PNA release A.09.42  
See What's New

## Noise Figure Receiver Characterization

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSE:NOISe:CALibration:RMEThod</td>
<td>RcvCharMethod</td>
</tr>
</tbody>
</table>

## Misc Commands

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSE:PATH:CONF:COPY</td>
<td>CopyFrom</td>
</tr>
<tr>
<td>CALC:PAR:WNUM?</td>
<td>WindowNumber</td>
</tr>
<tr>
<td>CALC:PAR:TNUM?</td>
<td>N/A</td>
</tr>
</tbody>
</table>

## Source / Role Commands (for apps)

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSE:ROLE:CATalog</td>
<td>DefinedRoles</td>
</tr>
<tr>
<td>SENSE:ROLE:DEVice</td>
<td>RoleDevice</td>
</tr>
</tbody>
</table>

---

**Noise Figure Receiver Characterization**

Select method

**Misc Commands**

Read CalSet stimulus values

Write CalSet stimulus values

Copy mechanical settings from specified channel

Return a measurement handle of the trace object

Read the PNA port which is connected to the DUT input.

Read the PNA port which is connected to the DUT output.

Read the window number of the selected trace.

Read the trace number of the selected trace.

---

**Source / Role Commands (for apps)**

Returns the roles for which sources can be used for the channel.

Set and returns the source to be used in the specified role.
### IMD and IM Spectrum Tone Power settings

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:IMD:TPOwr:EQualize:STATe</td>
<td>EqualTonePower</td>
</tr>
<tr>
<td>SENS:IMD:TPOwr:SET</td>
<td></td>
</tr>
<tr>
<td>SENS:IMS:TPOwr:EQualize:STATe</td>
<td>TonePowerSetAt</td>
</tr>
<tr>
<td>SENS:IMS:TPOwr:SET</td>
<td></td>
</tr>
</tbody>
</table>

### Pulse Commands

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:SWE:PULSe:MASTer:FREQ</td>
<td>MasterFrequency</td>
</tr>
<tr>
<td>SENS:SWE:PULSe:MASTer:PERiod</td>
<td>MasterPeriod</td>
</tr>
<tr>
<td>SENS:SWE:PULSe:MASTer:WIDTh</td>
<td>MasterWidth</td>
</tr>
<tr>
<td>SENS:SWE:PULSe:CWTime</td>
<td>AutoCWSweepTime</td>
</tr>
</tbody>
</table>

### Capabilities Commands

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>SupportedParameters</td>
</tr>
<tr>
<td>None</td>
<td>IFBWList</td>
</tr>
<tr>
<td>None</td>
<td>ResBWList</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release A.09.33**  See What’s New

### FCA Commands

<table>
<thead>
<tr>
<th>SCPI</th>
<th>COM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sense:Correction:Guided</td>
<td>Not Applicable</td>
</tr>
<tr>
<td>Sense:Correction:Collect:Guided:SMC</td>
<td></td>
</tr>
<tr>
<td>Sense:Correction:Collect:Guided:VMC</td>
<td></td>
</tr>
</tbody>
</table>
### Multiple Power Sensors

<table>
<thead>
<tr>
<th>Command</th>
<th>Command String</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable multiple sensors</td>
<td><code>SENS:CORR:COLL:GUID:PSEN:MULT</code></td>
<td>UseMultipleSensors</td>
</tr>
<tr>
<td>Assign power sensor name</td>
<td><code>SENS:CORR:COLL:GUID:PSEN:MULT:NAME</code></td>
<td>Name</td>
</tr>
<tr>
<td>Read the number of configured sensors</td>
<td><code>SENS:CORR:COLL:GUID:PSEN:MULT:COUN?</code></td>
<td>Count</td>
</tr>
<tr>
<td>Set connector type</td>
<td><code>SENS:CORR:COLL:GUID:PSEN:MULT:CONN</code></td>
<td>PowerSensorConnectorType</td>
</tr>
<tr>
<td>Set Cal Kit</td>
<td><code>SENS:CORR:COLL:GUID:PSEN:MULT:CKIT</code></td>
<td>PowerSensorCalKitType</td>
</tr>
</tbody>
</table>

### Phase Control

<table>
<thead>
<tr>
<th>Command</th>
<th>Command String</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase Sweep type</td>
<td><code>SENSe:SWEep:TYPE</code></td>
<td>SweepType</td>
</tr>
<tr>
<td>Set fixed phase value</td>
<td><code>SOURce:PHASE [:FIXed]</code></td>
<td>FixedPhase</td>
</tr>
<tr>
<td>Phase sweep start value</td>
<td><code>SOURce:PHASE:STARt</code></td>
<td>StartPhase</td>
</tr>
<tr>
<td>Phase sweep stop value</td>
<td><code>SOURce:PHASE:STOP</code></td>
<td>StopPhase</td>
</tr>
<tr>
<td>Phase parameter</td>
<td><code>SOURce:PHASE:PARameter</code></td>
<td>PhaseParameter</td>
</tr>
<tr>
<td>Set Phase control mode</td>
<td><code>SOURce:PHASE:PARameter:MODE</code></td>
<td>PhaseControlMode</td>
</tr>
<tr>
<td>Set reference port</td>
<td><code>SOURce:PHASE:PARameter:PORT</code></td>
<td>PhaseReferencePort</td>
</tr>
<tr>
<td>Read available phase control modes for the port</td>
<td><code>SOURce:PHASE:PARameter:MODE:CAT?</code></td>
<td>PhaseParameterModes</td>
</tr>
<tr>
<td>Couple sweep settings</td>
<td><code>SOURce:PHASE:CONTrol:COUple</code></td>
<td>CouplePhasePortSettings</td>
</tr>
<tr>
<td>Set number of sweep iterations</td>
<td><code>SOURce:PHASE:CONTrol:ITERation</code></td>
<td>PhaseIterationNumber</td>
</tr>
<tr>
<td>Set sweep tolerance</td>
<td><code>SOURce:PHASE:CONTrol:TOLerance</code></td>
<td>PhaseTolerance</td>
</tr>
</tbody>
</table>
Set and read an array of phase offsets.

SOURce:PHASe:CORRection:DATA

PhaseCorrectionData

Use phase offset array.

SOURce:PHASe:CORRection:STATe

PhaseCorrectionEnabled

Set and read an array of ratioed power offsets.

SOURce:PHASe:POFFset:CORR:DATA

RatioedPowerCorrectionData

Use power offset array.

SOURce:PHASe:POFFset:CORR:STATe

RatioedPowerCorrectionEnabled

Set the fixed power ratioed value

SOUR:PHAS:POFFset:FIXed

FixedRatioedPower

Set the start power ratioed value.

SOUR:PHAS:POFFset:STARt

StartRatioedPower

Set the stop power ratioed value.

SOUR:PHAS:POFFset:STOP

StopRatioedPower

2 and 4-port Fixture De-embed

2-port Reverse


Reverse2PortAdapter

4-port remap

CALC:FSIM:EMB:NETW<n>:PMAP

NetworkPortMap

Extrapolate

CALC:FSIM:SNP:EXTR

EnableSnPDataExtrapolation

Extrapolate added to Cal Set De-embedding

CSET:FIXT:DEEM

Deembed

Extrapolate added to Cal Set Embedding

CSET:FIX:EMBED

Embed

Mixer Segment Sweep

Recalculate

SENS:MIX:RECalculate

ReCalculate

Segment Calculate

SENS:MIX:SEGM<n>:CALC

SegmentCalculate

Query Count

SENS:MIX:SEGM:COUNt?

SegmentCount

Add Segments

SENS:MIX:SEGM<n>:ADD

AddSegment

Delete Segments

SENS:MIX:SEGM<n>:DEL

DeleteSegment

Remove All Segments

SENS:MIX:SEGM:DEL:ALL

DeleteAllSegments

State

SENS:MIX:SEGM<n>[:STATe]

SegmentState
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number Of Points</td>
<td>SENS:MIX:SEGM&lt;n&gt;:POINTs</td>
<td>Segment Points</td>
</tr>
<tr>
<td>IFBW</td>
<td>SENS:MIX:SEGM&lt;n&gt;:BWIDth</td>
<td>Segment IF Bandwidth</td>
</tr>
<tr>
<td>Input Start Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:INP:FREQ:START</td>
<td>Segment Start Frequency</td>
</tr>
<tr>
<td>Input Stop Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:INP:FREQ:STOP</td>
<td>Segment Stop Frequency</td>
</tr>
<tr>
<td>Input Power</td>
<td>SENS:MIX:SEGM&lt;n&gt;:INP:POWer</td>
<td>Segment Fixed Power</td>
</tr>
<tr>
<td>Output Start Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:OUTP:FREQ:START</td>
<td>Segment Start Frequency</td>
</tr>
<tr>
<td>Output Stop Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:OUTP:FREQ:STOP</td>
<td>Segment Stop Frequency</td>
</tr>
<tr>
<td>Output (+/-)</td>
<td>SENS:MIX:SEGM&lt;n&gt;:OUTP:FREQ:SIDeband</td>
<td>Segment Mixing Mode</td>
</tr>
<tr>
<td>Output Power</td>
<td>SENS:MIX:SEGM&lt;n&gt;:OUTP:POWer</td>
<td>Segment Fixed Power</td>
</tr>
<tr>
<td>LO Fixed Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:FREQ:FIXed</td>
<td>Segment Fixed Frequency</td>
</tr>
<tr>
<td>LO Start Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:FREQ:START</td>
<td>Segment Start Frequency</td>
</tr>
<tr>
<td>LO Stop Freq</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:FREQ:STOP</td>
<td>Segment Stop Frequency</td>
</tr>
<tr>
<td>LO Fixed/Swept</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:FREQ:MODE</td>
<td>Segment Range Mode</td>
</tr>
<tr>
<td>Input &gt;LO</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:FREQ:ILTI</td>
<td>Segment Is Input Greater Than LO</td>
</tr>
<tr>
<td>LO Power</td>
<td>SENS:MIX:SEGM&lt;n&gt;:LO&lt;x&gt;:POWer</td>
<td>Segment Fixed Power</td>
</tr>
<tr>
<td>IF (+/-)</td>
<td>SENS:MIX:SEGM&lt;n&gt;:IF:FREQ:SIDeband</td>
<td>Segment Mixing Mode</td>
</tr>
</tbody>
</table>
### Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Syntax</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Read available Meas Classes</td>
<td>SYST:MClass:CAT?</td>
<td>AvailableMeasurementClasses</td>
</tr>
<tr>
<td>Set receiver ratio to be used with receiver leveling.</td>
<td>SOURce:POWer:ALC:MODE:REC:RATio</td>
<td>ReceiverRatio</td>
</tr>
<tr>
<td>Removes adapters</td>
<td>SENS:CORR:COLL:GUID:ADAP:COUNt:ZERO</td>
<td>None</td>
</tr>
<tr>
<td>Perform Linear Interpolation</td>
<td>SYST:CORR:INT:LINear</td>
<td>None</td>
</tr>
<tr>
<td>Return if a CalSet exists</td>
<td>CSET:EXISts?</td>
<td>Exists</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release A.09.30**  See What's New

### Marker Display

<table>
<thead>
<tr>
<th>Command</th>
<th>Syntax</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Readouts Per Trace</td>
<td>DISP:WIND:ANN:MARK:NUMB</td>
<td>MarkerReadoutsPerTrace</td>
</tr>
<tr>
<td>Marker symbol</td>
<td>DISP:WIND:ANN:MARK:SYMB</td>
<td>MarkerSymbol</td>
</tr>
</tbody>
</table>

**GCX** - No new commands. Learn more .

### Overload Preferences

<table>
<thead>
<tr>
<th>Command</th>
<th>Syntax</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turn source power OFF when a receiver is overloaded?</td>
<td>SYST:PREF:ITEM:REC:OVER:POW</td>
<td>RFOffOnReceiverOverload</td>
</tr>
</tbody>
</table>
### Miscellaneous

**Use Last Result for Source Power Cal**

```
SOUR:POW:ALC:MODE:REC:LSPC LastLevelingAsSPC
```

**Selectively delete a channel**

```
SYSTem:CHANnels:DELeTe Remove Method RemoveChannelNumber
```

### Guided Power Cal

**Perform Power Cal**

```
```

**Match-correction ON/OFF**

```
SENS:CORR:METHods:MATCh MatchCorrectPower
```

### Cal Set Items

**Set or get name-value pair from calset**

None Item

**Remove name-value pair from calset**

None RemoveItem

**Enumerate name-value pair items in the calset.**

None EnumerateCalSets

---

The following are new programming commands for **PNA release A.09.22**  See What's New

**Returns the error correction state for the measurement.**

```
CALC:CORR:INDicator? ErrorCorrectionIndicator
```

**Calibrating specific channels**

```
SENS:CORR:COLL:GUID:CHAN:MODE N/A
```

### Port Mapping - Noise Figure Opt 028

**Write port mapping**

```
SENSe:NOISe:PMAP SetPortMap
```

**Read input port mapping**

```
SENSe:NOISe:PMAP:INPut? DeviceInputPort
```

**Read output port mapping**

```
SENSe:NOISe:PMAP:OUTPut? DeviceOutputPort
```

---

The following are new programming commands for **PNA release A.09.20**  See What's New
## Configure Pulse Measurements

<table>
<thead>
<tr>
<th>Feature</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pulse Meas Mode</td>
<td>SENS:SWE:PULS:MODE</td>
<td>PulseMeasMode</td>
</tr>
<tr>
<td>Autodetect Pulse mode</td>
<td>SENS:SWE:PULS:DTECT:mode</td>
<td>AutoDetection</td>
</tr>
<tr>
<td>Set Pulse Mode (Narrow</td>
<td>Wide)</td>
<td>SENS:SWE:PULS:WIDeband</td>
</tr>
<tr>
<td>Autoselect IFBW</td>
<td>SENS:SWE:PULS:IFBW</td>
<td>AutoIFBandWidth</td>
</tr>
<tr>
<td>Autoselect IF Gain</td>
<td>SENS:SWE:PULS:IFGain</td>
<td>AutoIFGain</td>
</tr>
<tr>
<td>Autoselect Pulse clock period</td>
<td>SENS:SWE:PULS:PRF</td>
<td>AutoOptimizePRF</td>
</tr>
<tr>
<td>Autoselect Width and Delay</td>
<td>SENS:SWE:PULS:TIMing</td>
<td>AutoPulseTiming</td>
</tr>
<tr>
<td>Autoselect Pulse Gens</td>
<td>SENS:SWE:PULS:DRIVE</td>
<td>AutoSelectPulseGen</td>
</tr>
</tbody>
</table>

## External Pulse Configuration

<table>
<thead>
<tr>
<th>Feature</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>PulseSyncIn Trigger Polarity</td>
<td>Sense:Pulse:TPolarity</td>
<td>TriggerInPolarity</td>
</tr>
<tr>
<td>PulseSyncIn Trigger Type</td>
<td>Sense:Pulse:TType</td>
<td>TriggerInType</td>
</tr>
</tbody>
</table>

## PSAT Marker Search

<table>
<thead>
<tr>
<th>Feature</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initiate a PSAT search</td>
<td>CALC:MARK:PSAT:BACKoff</td>
<td>SearchPowerSaturation</td>
</tr>
<tr>
<td>Set and read PSAT backoff</td>
<td>CALC:MARK:PSAT:BACKoff</td>
<td>PMaxBackOff</td>
</tr>
<tr>
<td>Read PSat Out</td>
<td>CALC:MARK:PSAT:POUT?</td>
<td>POut</td>
</tr>
<tr>
<td>Read PSat In</td>
<td>CALC:MARK:PSAT:PIN?</td>
<td>Pin</td>
</tr>
<tr>
<td>Read PMax Out</td>
<td>CALC:MARK:PSAT:POUT:MAXimum?</td>
<td>PMaxOut</td>
</tr>
<tr>
<td>Read PMax In</td>
<td>CALC:MARK:PSAT:PIN:MAXimum?</td>
<td>PMaxIn</td>
</tr>
<tr>
<td>Read Gain Sat</td>
<td>CALC:MARK:PSAT:GAIN?</td>
<td>GainSaturation</td>
</tr>
<tr>
<td>Read Gain Max</td>
<td>CALC:MARK:PSAT:GAIN:MAXimum?</td>
<td>GainMax</td>
</tr>
<tr>
<td>Read Gain Linear</td>
<td>CALC:MARK:PSAT:GAIN:LINEar?</td>
<td>GainLinear</td>
</tr>
</tbody>
</table>
Read Comp Sat
CALC:MARK:PSAT:COMP:SAT? CompressionSaturation
Read Comp Max

**PNOP Marker Search**

Initiate a PNOP search
CALC:MARK:PNOP:BACKoff SearchPowerNormalOperatingPoint
Set and read PNOP backoff
CALC:MARK:PNOP:BACKoff BackOff
Set and read PNOP Power Offset
CALC:MARK:PNOP:POFFset PinOffset
Read Pnop Out
CALC:MARK:PNOP:OUT? POut
Read Pnop in
CALC:MARK:PNOP:PIN? Pin
Read Pnop Gain
CALC:MARK:PNOP:GAIN? Gain
Read Pnop Comp
CALC:MARK:PNOP:COMPression? Compression
Read PMax Out
CALC:MARK:PNOP:OUT:MAXimum? PMaxOut
Read PMax In
CALC:MARK:PNOP:PIN:MAXimum? PMaxIn
Read Gain Max
Read Comp Max
CALC:MARK:PNOP:COMPression:MAXimum? CompressionMax
Read PBO Out
Read PBO In
CALC:MARK:PNOP:BACKoff:PIN? BackOffPin
Read PBO Gain

**Include Phase in SMC measurements**

Enable Phase
SENS:MIX:PHASe EnablePhase
Set normalize point
SENS:MIX:NORMalize NormalizePoint
Choose known delay or S2P
Set known delay
Set Cal Mixer Char S2P filename
<table>
<thead>
<tr>
<th><strong>Group Delay Aperture</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Set aperture using frequency</td>
</tr>
<tr>
<td>Set aperture using percent of span</td>
</tr>
<tr>
<td>Set aperture using fixed number of points</td>
</tr>
<tr>
<td>Set Preference to 2 points</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Calibrations</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns the names of the mechanical cal kits for unguided calibrations.</td>
</tr>
<tr>
<td>Set/get by name which cal kit is currently selected for use by unguided cal.</td>
</tr>
<tr>
<td>Read pass fail status of tolerance limits on the target cal power</td>
</tr>
<tr>
<td>Gets the power correction dB values from the prior iteration of the source power cal.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Fixturing</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Create Cal Set with De-embedded fixture removed.</td>
</tr>
<tr>
<td>Create Cal Set with Matching Network included.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Noise Receiver (Opt 028)</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Select Noise Receiver</td>
</tr>
<tr>
<td>Miscellaneous</td>
</tr>
<tr>
<td>---------------------------------------------------</td>
</tr>
<tr>
<td>Include SC12 Sweep</td>
</tr>
<tr>
<td>IncludeReverseSweep</td>
</tr>
<tr>
<td>Active Window Background</td>
</tr>
<tr>
<td>Disp:Color:ABACKground</td>
</tr>
<tr>
<td>ActiveBackground</td>
</tr>
<tr>
<td>Sweep Delay</td>
</tr>
<tr>
<td>SweepDelay</td>
</tr>
<tr>
<td>Preference: On PRESET always turn power ON</td>
</tr>
<tr>
<td>Send and return Instrument state file to remote computer</td>
</tr>
<tr>
<td>SYSTem:SET</td>
</tr>
<tr>
<td>None</td>
</tr>
<tr>
<td>Read the display image into remote computer</td>
</tr>
<tr>
<td>HCOPy:SDUMp:DATA?</td>
</tr>
<tr>
<td>None</td>
</tr>
<tr>
<td>Set format of display image</td>
</tr>
<tr>
<td>HCOPy:SDUMp:DATA:FORM</td>
</tr>
<tr>
<td>None</td>
</tr>
<tr>
<td>Set and return LXI status</td>
</tr>
<tr>
<td>LXI:IDEN</td>
</tr>
<tr>
<td>LXIDeviceIDState</td>
</tr>
<tr>
<td>GCA Safe mode - max power</td>
</tr>
<tr>
<td>Sens:GCS:SAFE:MLIM</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Set Line type of the window grid</td>
</tr>
<tr>
<td>DISP:WIND:TRAC:GRAT:GRID:LTYPE GridLineType</td>
</tr>
<tr>
<td>Save Data</td>
</tr>
<tr>
<td>MOMEM:STOR:DATA</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 9.10**  See What's New

<table>
<thead>
<tr>
<th>NFX</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sets the power sensor connector</td>
</tr>
<tr>
<td>SENS:CORR:COLL:GUID:PSEN:CONN PowerSensorConnectorType</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets the power sensor calkit</td>
</tr>
<tr>
<td>SENS:CORR:COLL:GUID:PSEN:CKIT PowerSensorCalKitType</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets power level for source power cal</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets auto orientation state for noise tuner</td>
</tr>
<tr>
<td>SENS:NOISE:TUNer:ORIent AutoOrientTuner</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets LO power calibration state</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets the source pull technique to compute DUT S-parameters</td>
</tr>
<tr>
<td>SENS:NOISE:PULL SourcePullForSParameters</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Sets the state of ENR adapter de-embedding.</td>
</tr>
<tr>
<td>SENS:CORR:COLL:NOIS:ENR:ADAP:DEEM ForceDeEmbedENRAAdapter</td>
</tr>
</tbody>
</table>
Sets the state of Power Sensor adapter de-embedding.

**Help About commands**

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSP Revision</td>
<td>SYST:CONF:REVision:DSP?</td>
<td>DSP Revision</td>
</tr>
<tr>
<td>DSP FPGA</td>
<td>SYST:CONF:REVision:DSPFpga?</td>
<td>DSP FPGA</td>
</tr>
<tr>
<td>CPU Speed</td>
<td>SYST:CONF:REVision:CPU?</td>
<td>CPU Speed</td>
</tr>
<tr>
<td>Hostname</td>
<td>SYST:COMM:LAN:HOSTname?</td>
<td>Hostname</td>
</tr>
</tbody>
</table>

**Miscellaneous**

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Perform and apply Response (Normalization) cal</td>
<td>SENS:CORR:COLL:METHod</td>
<td>DoResponseCal</td>
</tr>
<tr>
<td>PulseSyncIn Trigger Polarity</td>
<td>Sense:Pulse:TPolarity</td>
<td>TriggerInPolarity</td>
</tr>
<tr>
<td>PulseSyncIn Trigger Type</td>
<td>Sense:Pulse:TTtype</td>
<td>TriggerInType</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 9.0**  See What's New

**Configure External Devices**

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adds an external device to the system.</td>
<td>SYST:CONF:EDEV:ADD</td>
<td>Add (External Device)</td>
</tr>
<tr>
<td>Returns names of all configured devices</td>
<td>SYST:CONF:EDEV:CAT?</td>
<td>Items</td>
</tr>
<tr>
<td>Set driver for the external device.</td>
<td>SYST:CONF:EDEV:DRIV</td>
<td>Driver</td>
</tr>
<tr>
<td>Set type of device.</td>
<td>SYST:CONF:EDEV:DTYP</td>
<td>DeviceType</td>
</tr>
<tr>
<td>Configuration path for external device.</td>
<td>SYST:CONF:EDEV:IOC</td>
<td>IOConfiguration</td>
</tr>
<tr>
<td>Enable or disable communication with device.</td>
<td>SYST:CONF:EDEV:IOEN</td>
<td>IOEnable</td>
</tr>
<tr>
<td>Activation state of the device.</td>
<td>SYST:CONF:EDEV:STAT</td>
<td>Active</td>
</tr>
<tr>
<td>Time out value for external device.</td>
<td>SYST:CONF:EDEV:TOUT</td>
<td>TimeOut</td>
</tr>
</tbody>
</table>

3201
**External Source Config**

- **Set Dwell per Point**
  - Command: `SYST:CONF:EDEV:SOUR:DPP`
  - Description: DwellPerPoint

- **Set Trigger Mode**
  - Command: `SYST:CONF:EDEV:SOUR:TMOD`
  - Description: Trigger Mode

- **Set Trigger Port**
  - Command: `SYST:CONF:EDEV:SOUR:TPORt`
  - Description: TriggerPort

**Power Meter As Receiver (PMAR) Config**

- **Enable min and max freqs**
  - Command: `SYST:CONF:EDEV:PMAR:FLIM`
  - Description: LimitFrequency

- **Set Max freq**
  - Description: MaximumFrequency

- **Set Min freq**
  - Description: MinimumFrequency

- **Set max number of PM readings**
  - Description: ReadingsPerPoint

- **Set tolerance level**
  - Description: ReadingsTolerance

- **Select sensor**
  - Description: SensorIndex

- **Set Cal Factor data**
  - Description: ReferenceCalFactor

- **Set Cal Factor frequencies**
  - Description: Frequency

- **Set Power loss data**
  - Description: Loss

- **Set Power loss frequencies**
  - Description: Frequency

- **Enable Power loss data**
  - Description: UsePowerLossSegments

- **Set reference cal factor**
  - Description: ReferenceCalFactor

**Power Limit**

- **Set power limit**
  - Command: `SYST:POWer:LIMit`
  - Description: Limit

- **Power limit ON/OFF**
  - Command: `SYST:POWer:LIMit:STATe`
  - Description: State

- **Power limit UI lock**
  - Command: `SYST:POWer:LIMit:LOCK`
  - Description: Lock
### Scale Coupling

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set method</td>
<td>DISP:WIND:TRAC:Y:COUP:METH</td>
<td>ScaleCouplingMethod</td>
</tr>
<tr>
<td>Enable window</td>
<td>DISP:WIND:TRAC:Y:COUP</td>
<td>ScaleCouplingState</td>
</tr>
</tbody>
</table>

### Display and Print Colors

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set background color</td>
<td>DISP:COL:BACK</td>
<td>Background</td>
</tr>
<tr>
<td>Set labels and grid frame colors</td>
<td>DISP:COL:GRAT1</td>
<td>ActiveLabels</td>
</tr>
<tr>
<td>Set inner lines of all grids in all windows colors</td>
<td>DISP:COL:GRAT2</td>
<td>Grid</td>
</tr>
<tr>
<td>Set Inactive window label colors</td>
<td>DISP:COL:ILAB</td>
<td>InactiveLabels</td>
</tr>
<tr>
<td>Set limit line colors</td>
<td>DISP:COL:LIM1</td>
<td>FailedTraces</td>
</tr>
<tr>
<td>Set trace data and Limit Line colors</td>
<td>DISP:COL:TRAC:DATA</td>
<td>DataAndLimits</td>
</tr>
<tr>
<td>Set data trace marker colors</td>
<td>DISP:COL:TRAC:MARK</td>
<td>Markers</td>
</tr>
<tr>
<td>Set memory trace colors</td>
<td>DISP:COL:TRAC:MEM</td>
<td>Memory</td>
</tr>
<tr>
<td>Set memory trace marker colors</td>
<td>DISP:COL:TRAC:MMAR</td>
<td>MemoryMarkers</td>
</tr>
<tr>
<td>Load a color theme</td>
<td>DISP:COL:LOAD</td>
<td>LoadTheme</td>
</tr>
<tr>
<td>Saves the current color theme.</td>
<td>DISP:COL:STOR</td>
<td>StoreTheme</td>
</tr>
<tr>
<td>Resets to the default PNA colors.</td>
<td>DISP:COL:RES</td>
<td>ResetTheme</td>
</tr>
</tbody>
</table>

### IMD and IMS Limited Port Mapping

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set port map</td>
<td>SENS:IMD:PMAP</td>
<td>SetPortMap</td>
</tr>
<tr>
<td></td>
<td>SENS:IMS:PMAP</td>
<td></td>
</tr>
<tr>
<td>Read Input</td>
<td>SENS:IMD:PMAP:INP?</td>
<td>DeviceInputPort</td>
</tr>
<tr>
<td></td>
<td>SENS:IMS:PMAP:INP?</td>
<td></td>
</tr>
<tr>
<td>Read Output</td>
<td>SENS:IMD:PMAP:OUTP?</td>
<td>DeviceOutputPort</td>
</tr>
<tr>
<td></td>
<td>SENS:IMS:PMAP:OUTP?</td>
<td></td>
</tr>
</tbody>
</table>
### ECal User Char to Disk (new and modified commands)

<table>
<thead>
<tr>
<th>Action</th>
<th>Command</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns the factory defined and user-defined connectors.</td>
<td>SENSE:CORR:CKIT:ECAL:CHAR:CONN:CAT?</td>
<td>None</td>
</tr>
<tr>
<td>Initiate a User Char - optionally check module memory.</td>
<td>SENSE:CORR:CKIT:ECAL:CHAR:INIT</td>
<td>InitializeEx</td>
</tr>
<tr>
<td>Saves a new characterization to disk memory</td>
<td>SENSE:CORR:CKIT:ECAL:CHAR:DMEM:SAVE</td>
<td>SaveToDiskMemory</td>
</tr>
<tr>
<td>Delete disk memory characterizations.</td>
<td>SENSE:CORR:CKIT:ECAL:DMEMory:CLEar</td>
<td>None</td>
</tr>
<tr>
<td>Saves a disk memory characterization to an archive file.</td>
<td>SENSE:CORR:CKIT:ECAL:EXPort</td>
<td>None</td>
</tr>
<tr>
<td>Imports the ECal characterization from the specified archive file.</td>
<td>SENSE:CORR:CKIT:ECAL:DMEMory:IMPort</td>
<td>None</td>
</tr>
<tr>
<td>Reads the user-characterization info from ECal module or PNA disk memory.</td>
<td>SENSE:CORR:CKIT:ECAL:KNAM:INF?</td>
<td>None</td>
</tr>
</tbody>
</table>

### Gain Compression Analysis

<table>
<thead>
<tr>
<th>Action</th>
<th>Command</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable a compression analysis trace</td>
<td>CALC:GCM:ANAL:ENAB</td>
<td>AnalysisEnable</td>
</tr>
<tr>
<td>Set CW frequency</td>
<td>CALC:GCM:ANAL:CWFR</td>
<td>AnalysisCWFreq</td>
</tr>
<tr>
<td>Set to discrete or interpolated CW frequencies</td>
<td>CALC:GCM:ANAL:ISD</td>
<td>AnalysisIsDiscreteFreq</td>
</tr>
<tr>
<td>Sets X-axis display</td>
<td>CALC:GCM:ANAL:XAX</td>
<td>AnalysisXAxis</td>
</tr>
</tbody>
</table>
### Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYST:PREF:ITEM:RTOF</td>
<td>Red Trace On Failures</td>
</tr>
<tr>
<td>CALC:X?</td>
<td>Returns the X-axis values for the selected trace.</td>
</tr>
<tr>
<td>SENS:CORR:CKIT:EXP</td>
<td>Saves a Cal Kit to a file.</td>
</tr>
<tr>
<td>SENS:CORR:COLL:GUID:CKIT:CAT?</td>
<td>Returns the list of cal kits that use the specified connector.</td>
</tr>
<tr>
<td>SENS:GCS:COMP:SAT:LEV</td>
<td>Gain Compression Saturation level</td>
</tr>
<tr>
<td>SENS:CORR:PREF:CAL:RANG</td>
<td>Set Cal FOM Range Preference</td>
</tr>
<tr>
<td>SENS:CORR:CSET:STIMulus?</td>
<td>Return the Calset X-axis FOM frequency range</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.60** See What's New

### Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENSe:SEGm:LIST</td>
<td>Reads or writes the segment sweep table.</td>
</tr>
<tr>
<td>Sens:Corr:Ckit:CLEar</td>
<td>Optional arguments for...</td>
</tr>
<tr>
<td>Sens:Corr:Ckit:Init</td>
<td></td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.55** See What's New

### Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sens:pulse:SUBPointTrig</td>
<td>Enable subpoint triggering</td>
</tr>
</tbody>
</table>

### Description

**IMD-X for Converters**
### Create or Change a Custom (Application) Measurement

Create a custom measurement

Change a custom measurement

Return handle to a converter object

### Configure a Mixer

Calculate Input and Output frequencies

X-axis display

Discard Changes

Load a mixer setup

Save a mixer setup

Apply mixer settings

Assign a source to mixer input or LO.

Read all assigned roles

Read the source assigned to a role.

### Set Input

Input start frequency

Input stop frequency

Input power level

Numerator - Input Frac.Mult

Denominator - Input Frac.Mult

Input to Swept or fixed

Input fixed frequency

### Set LO

Recall a previously-configured external source. This is the only external LO configuration command.
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LO freq sweep mode (fixed or swept)</td>
<td>LO fixed frequency</td>
</tr>
<tr>
<td></td>
<td>LO start frequency</td>
</tr>
<tr>
<td></td>
<td>LO stop frequency</td>
</tr>
<tr>
<td></td>
<td>LO power</td>
</tr>
<tr>
<td></td>
<td>LO power start</td>
</tr>
<tr>
<td></td>
<td>LO power stop</td>
</tr>
<tr>
<td></td>
<td>Numerator - LO Frac.Mult</td>
</tr>
<tr>
<td></td>
<td>Denominator - LO Frac.Mult</td>
</tr>
<tr>
<td></td>
<td>Input Greater / Less that LO</td>
</tr>
<tr>
<td><strong>Set IF</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Sideband (high or low)</td>
</tr>
<tr>
<td><strong>Set Output</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Sideband (high or low)</td>
</tr>
<tr>
<td></td>
<td>Output start frequency</td>
</tr>
<tr>
<td></td>
<td>Output stop frequency</td>
</tr>
<tr>
<td></td>
<td>Output to swept or fixed</td>
</tr>
<tr>
<td></td>
<td>Output fixed frequency</td>
</tr>
<tr>
<td><strong>IMDx Cal</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Enable LO Power cal</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.50**  See What's New
### Compression Marker

<table>
<thead>
<tr>
<th>Description</th>
<th>Command Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression Marker level found.</td>
<td>CALC:MARKer:COMPression:LEVel</td>
<td>CompressionLevel</td>
</tr>
<tr>
<td>Read Compression Marker Input power</td>
<td>CALC:MARKer:COMPression:PIN</td>
<td>CompressionPin</td>
</tr>
<tr>
<td>Read Compression Marker Output power</td>
<td>CALC:MARKer:COMPression:POUT</td>
<td>CompressionPout</td>
</tr>
<tr>
<td>New Search function</td>
<td>CALC:MARKer:FUNCTION:SEL</td>
<td>SearchCompressionPoint</td>
</tr>
<tr>
<td>New Execute function</td>
<td>CALC:MARKer:FUNCTION:EXEC</td>
<td>SearchCompressionPoint</td>
</tr>
</tbody>
</table>

### Port Extensions

<table>
<thead>
<tr>
<th>Description</th>
<th>Command Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Port Ext in distance</td>
<td>SENS:CORR:EXT:PORT:DIST</td>
<td>PortDistance</td>
</tr>
<tr>
<td>Set distance units</td>
<td>SENS:CORR:EXT:PORT:UNIT</td>
<td>PortDistanceUnit</td>
</tr>
<tr>
<td>Set Media per port</td>
<td>SENS:CORR:EXT:PORT:MEDium</td>
<td>PortMedium</td>
</tr>
<tr>
<td>Set waveguide cutoff freq per port</td>
<td>SENS:CORR:EXT:PORT:WGCutoff</td>
<td>PortWGCutoffFreq</td>
</tr>
<tr>
<td>Set Velocity Factor per port</td>
<td>SENS:CORR:EXT:PORT:VELF</td>
<td>PortVelocityFactor</td>
</tr>
<tr>
<td>Couple to system Velocity Factor</td>
<td>SENS:CORR:EXT:PORT:SYSV</td>
<td>PortCoupleToSystemVelocity</td>
</tr>
<tr>
<td>Couple to system Media type</td>
<td>SENS:CORR:EXT:PORT:SYSM</td>
<td>PortCoupleToSystemMedia</td>
</tr>
</tbody>
</table>

### Electrical Delay

<table>
<thead>
<tr>
<th>Description</th>
<th>Command Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delay in distance</td>
<td>CALC:CORR:EDElay:DISTance</td>
<td>ElecDistanceDelay</td>
</tr>
<tr>
<td>Set units for distance</td>
<td>CALC:CORR:EDElay:UNIT</td>
<td>ElecDistanceDelayUnit</td>
</tr>
</tbody>
</table>
### Receiver Leveling

<table>
<thead>
<tr>
<th>Command</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rx Level ON</td>
<td>OFF</td>
<td>SOUR:POW:ALC:MODE:REC</td>
</tr>
<tr>
<td>Select the reference receiver</td>
<td>SOUR:POW:ALC:MODE:REC:REF</td>
<td>ReferenceReceiver</td>
</tr>
<tr>
<td>Set maximum iterations</td>
<td>SOUR:POW:ALC:MODE:REC:ITER</td>
<td>IterationNumber</td>
</tr>
<tr>
<td>Set tolerance</td>
<td>SOUR:POW:ALC:MODE:REC:TOL</td>
<td>Tolerance</td>
</tr>
<tr>
<td>Set offset value.</td>
<td>SOUR:POW:ALC:MODE:REC:OFFS</td>
<td>PowerOffset</td>
</tr>
<tr>
<td>Separate IFBW</td>
<td>SOUR:POW:ALC:MODE:REC:FAST</td>
<td>FastMode</td>
</tr>
<tr>
<td>Set Rx IFBW</td>
<td>SOUR:POW:ALC:MODE:REC:IFBW</td>
<td>LevelingIFBW</td>
</tr>
<tr>
<td>Safe mode ON</td>
<td>OFF</td>
<td>SOUR:POW:ALC:MODE:REC:SAFE</td>
</tr>
<tr>
<td>Safe mode Max power</td>
<td>SOUR:POW:ALC:MODE:REC:SAFE:MAX</td>
<td>PowerMax</td>
</tr>
<tr>
<td>Safe mode Min power</td>
<td>SOUR:POW:ALC:MODE:REC:SAFE:MIN</td>
<td>PowerMin</td>
</tr>
</tbody>
</table>

### Phase Sweep

<table>
<thead>
<tr>
<th>Command</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable Phase sweep</td>
<td>CALC:FSIM:BAL:PHAS:SWE:STAT</td>
<td>PhaseSwpState</td>
</tr>
<tr>
<td>Start Phase port 1</td>
<td>CALC:FSIM:BAL:BPOR:SWE:PHAS:STAR BalPort1StartPhase</td>
<td></td>
</tr>
<tr>
<td>Stop Phase port 1</td>
<td>CALC:FSIM:BAL:BPOR:SWE:PHAS:STOP BalPort1StopPhase</td>
<td></td>
</tr>
<tr>
<td>Stop Phase port 2</td>
<td>CALC:FSIM:BAL:BPOR:SWE:PHAS:STOP BalPort2StopPhase</td>
<td></td>
</tr>
<tr>
<td>Enable as fixture offset</td>
<td>CALC:FSIM:BAL:FIXT:SWE:PHAS</td>
<td>PhaseSwpAsFixture</td>
</tr>
</tbody>
</table>

### Other

<table>
<thead>
<tr>
<th>Command</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns ECal orientation.</td>
<td></td>
<td>Not Applicable</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.35**  See What's New
### FIFO Data Buffer

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FIFO ON/OFF</td>
<td>SYST:FIFO[:STATe]</td>
<td>State</td>
</tr>
<tr>
<td>Read number of data points</td>
<td>SYST:FIFO:DATA:COUNT?</td>
<td>DataCount</td>
</tr>
<tr>
<td>Read data</td>
<td>SYST:FIFO:DATA?</td>
<td>Data</td>
</tr>
<tr>
<td>Read data compact form</td>
<td>None</td>
<td>DataInCompactForm</td>
</tr>
<tr>
<td>Clear data</td>
<td>SYST:FIFO:DATA:CLEar</td>
<td>Clear</td>
</tr>
</tbody>
</table>

### Other N5264A Commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FastCW</td>
<td>SENS:SWE:TYPE:FACW</td>
<td>FastCWPointCount</td>
</tr>
<tr>
<td>Enable Point Averaging</td>
<td>SENS:AVER:MODE</td>
<td>AverageMode</td>
</tr>
<tr>
<td>Enable Point Sweep</td>
<td>SENS:SWE:GEN:POINTsweep</td>
<td>PointSweepState</td>
</tr>
<tr>
<td>Set Trace Sweep</td>
<td>SENS:SWE:TRIG:MODE</td>
<td>Trigger Mode</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.33**  
See What's New

### Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set display format units</td>
<td>Calc:Format:Unit</td>
<td>FormatUnit</td>
</tr>
<tr>
<td>Perform trace max</td>
<td>Disp:TMAX</td>
<td>TraceMax</td>
</tr>
<tr>
<td>Fast sweep mode</td>
<td>SENS:SWE:SPE</td>
<td>SweepSpeedMode</td>
</tr>
<tr>
<td>Launch Cal Wizard for apps (new behavior)</td>
<td>SYST:CORR:WIZ</td>
<td>LaunchCalWizard</td>
</tr>
<tr>
<td>Queries the TCP/IP port number for a TCP/IP socket connection.</td>
<td>SYST:COMM:TCPPIP:CONT</td>
<td>None</td>
</tr>
<tr>
<td>Set CWFreq to Marker location</td>
<td>CALC:MARK:SET</td>
<td>SetCWFreq</td>
</tr>
</tbody>
</table>
Returns a list of channel numbers
SYST:CHAN:CAT?  Not new

Returns measurement numbers
SYST:MEAS:CAT?  Not new

Returns trace number
SYST:MEAS<n>:TRACe?  Not new

Returns Meas name
SYST:MEAS:NAME?  Not new

Return window number
SYST:MEAS<n>:WINDow?  Not new

Returns window numbers
SYST:WIND:CAT?  Not new

Same as calc:par:sel except takes a meas number
CALC:PAR:MNUM:[SEL]  Not new

Returns Limit line pass/fail status
CALC:LIMIT:FAIL?  Not new

Deletes the current limit line?
CALC:LIMIT:DATA:DELe te  Not new

---

**Swept IMD**

<table>
<thead>
<tr>
<th>Create a measurement</th>
<th>Calc:Custom:Define</th>
<th>CreateCustomMeasurementEx</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set sweep type</td>
<td>SENS:IMD:SWE:TYPE</td>
<td>SweepType</td>
</tr>
<tr>
<td>Set DeltaF</td>
<td>SENS:IMD:DFR</td>
<td>DeltaFrequency</td>
</tr>
<tr>
<td>Set center freq</td>
<td>SENS:IMD:FREQ:FCEN</td>
<td>FrequencyCenter</td>
</tr>
<tr>
<td>Start for center freq sweep</td>
<td>SENS:IMD:FREQ:FCEN:STAR</td>
<td>FrequencyCenterStart</td>
</tr>
<tr>
<td>Stop for center freq sweep</td>
<td>SENS:IMD:FREQ:FCEN:STOP</td>
<td>FrequencyCenterStop</td>
</tr>
<tr>
<td>Center for center freq sweep</td>
<td>SENS:IMD:FREQ:FCEN:CENT</td>
<td>FrequencyCenterCenter</td>
</tr>
<tr>
<td>Span for center freq sweep</td>
<td>SENS:IMD:FREQ:FCEN:SPAN</td>
<td>FrequencyCenterSpan</td>
</tr>
<tr>
<td>Start for DeltaF sweep</td>
<td>SENS:IMD:FREQ:DFR:STAR</td>
<td>DeltaFrequencyStart</td>
</tr>
<tr>
<td>Stop for DeltaF sweep</td>
<td>SENS:IMD:FREQ:DFR:STOP</td>
<td>DeltaFrequencyStop</td>
</tr>
<tr>
<td>Set F1 for CW and Power sweep</td>
<td>SENS:IMD:FREQ:F1</td>
<td>F1Frequency</td>
</tr>
<tr>
<td>Set F2 for CW and Power sweep</td>
<td>SENS:IMD:FREQ:F2</td>
<td>F2Frequency</td>
</tr>
<tr>
<td>Set main tone IFBW</td>
<td>SENS:IMD:IFBW:MAIN</td>
<td>MainToneIFBandwidth</td>
</tr>
<tr>
<td>Parameter</td>
<td>Command/Setting</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------------------------</td>
<td>---------------------------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>Set product tones IFBW</td>
<td>SENS:IMD:IFBW:IMT</td>
<td>IMToneIFBandwidth</td>
</tr>
<tr>
<td>Enables power coupling for F1 and F2</td>
<td>SENS:IMD:TPOW:COUP</td>
<td>CoupleTonePower</td>
</tr>
<tr>
<td>Set power level for F1 tone</td>
<td>SENS:IMD:TPOW:F1</td>
<td>TonePower</td>
</tr>
<tr>
<td>Set power level for F2 tone</td>
<td>SENS:IMD:TPOW:F2</td>
<td>TonePower</td>
</tr>
<tr>
<td>F1 start for power sweep</td>
<td>SENS:IMD:TPOW:F1:STAR</td>
<td>TonePowerStart</td>
</tr>
<tr>
<td>F1 stop for power sweep</td>
<td>SENS:IMD:TPOW:F1:STOP</td>
<td>TonePowerStop</td>
</tr>
<tr>
<td>F2 start for power sweep</td>
<td>SENS:IMD:TPOW:F2:STAR</td>
<td>TonePowerStart</td>
</tr>
<tr>
<td>F2 stop for power sweep</td>
<td>SENS:IMD:TPOW:F2:STOP</td>
<td>TonePowerStop</td>
</tr>
<tr>
<td>Read highest product allowed</td>
<td>None</td>
<td>HighestOrderProduct</td>
</tr>
<tr>
<td><strong>For CTB, CSO, and XMod parameters</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Normalization Mode</td>
<td>SENS:IMD:NORM:MODE</td>
<td>CompositeNormalizationMode</td>
</tr>
<tr>
<td>Normalized CSO power</td>
<td>SENS:IMD:CSO:NORM:POW</td>
<td>CompositeNormalizedCSOPower</td>
</tr>
<tr>
<td>CSO Offset</td>
<td>SENS:IMD:CSO:OFFS</td>
<td>CSOOffset</td>
</tr>
<tr>
<td>CSO Number of Distortion products</td>
<td>SENS:IMD:CSO:NDPR</td>
<td>CSONumDistortionProducts</td>
</tr>
<tr>
<td>Normalized CTB power</td>
<td>SENS:IMD:CTB:NORM:POW</td>
<td>CompositeNormalizedCTBPower</td>
</tr>
<tr>
<td>CTB and XMod Number of carriers</td>
<td>SENS:IMD:CTB:NCAR</td>
<td>CTBXMODNumCarriers</td>
</tr>
<tr>
<td>CTB Offset</td>
<td>SENS:IMD:CTB:OFFS</td>
<td>CTBOffset</td>
</tr>
</tbody>
</table>
**IMD Calibration**

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Cal frequencies</td>
<td>SENS:CORR:IMD:CAL:FREQ</td>
<td>Calibration Frequencies</td>
</tr>
<tr>
<td>Max Products</td>
<td>SENS:CORR:IMD:MPR</td>
<td>Max Product</td>
</tr>
<tr>
<td>Set power</td>
<td>SENS:CORR:IMD:POW</td>
<td>Power Level</td>
</tr>
<tr>
<td>Sensor Cal Kit</td>
<td>SENS:CORR:IMD:SENS:CKIT</td>
<td>Power Sensor Cal Kit Type</td>
</tr>
<tr>
<td>Sensor connector</td>
<td>SENS:CORR:IMD:SENS:CONN</td>
<td>Power Sensor Connector Type</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.2**  See What's New

**iTMSA**

<table>
<thead>
<tr>
<th>Command</th>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Stimulus Mode</td>
<td>CALC: FSIM: BAL: STIM: MOD</td>
<td>Mode</td>
</tr>
<tr>
<td>Set Phase Offset</td>
<td>CALC: FSIM: BAL: BPOR: OFFS: PHAS</td>
<td>Bal Port 1 Phase Offset</td>
</tr>
<tr>
<td>Set Phase Offset as fixture</td>
<td>CALC: FSIM: BAL: FIXT: OFFS: PHAS</td>
<td>Phase As Fixture</td>
</tr>
<tr>
<td>Set Power Offset as fixture</td>
<td>CALC: FSIM: BAL: FIXT: OFFS: POW</td>
<td>Power As Fixture</td>
</tr>
<tr>
<td>Set Source power for balanced ports</td>
<td>SOUR: POW</td>
<td>Test Port Power</td>
</tr>
</tbody>
</table>

Returns the number of source ports.

Returns the string names of source ports.

Returns the port number for the specified string port name.
### Uncoupled Power Sweep

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Start power for uncoupled power sweep</td>
<td>SOUR:POW:PORT:STARt</td>
<td>StartPowerEx</td>
</tr>
<tr>
<td>Set Start power for uncoupled power sweep</td>
<td>SOUR:POW:PORT:STOP</td>
<td>StopPowerEx</td>
</tr>
</tbody>
</table>

### Choose FCA ports

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Map PNA to DUT ports</td>
<td>SENS:MIX:PMAP</td>
<td>SetDutPorts</td>
</tr>
<tr>
<td>Read Input port number</td>
<td>SENS:MIX:PMAP:INP?</td>
<td>DeviceInputPort</td>
</tr>
<tr>
<td>Read Output port number</td>
<td>SENS:MIX:PMAP:OUTP?</td>
<td>DeviceOutputPort</td>
</tr>
</tbody>
</table>

### LXI

<table>
<thead>
<tr>
<th>Description</th>
<th>None</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Returns <strong>Structured</strong> status of the PNA networking configuration.</td>
<td></td>
<td>GetIPConfigurationStruct</td>
</tr>
<tr>
<td>Returns <strong>string</strong> status of the PNA networking configuration.</td>
<td></td>
<td>LANConfiguration</td>
</tr>
<tr>
<td>Resets the PNA LAN configuration.</td>
<td></td>
<td>LANConfigurationInitialize</td>
</tr>
<tr>
<td>Modifies settings of the PNA computer networking configuration.</td>
<td></td>
<td>SetIPConfiguration</td>
</tr>
<tr>
<td>Displays the LAN Status dialog with LAN Status Indicator showing IDENTIFY.</td>
<td></td>
<td>LXIDeviceIDState</td>
</tr>
</tbody>
</table>
## Miscellaneous

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYST:PREF:DEF cap.RestoreDefaults</td>
<td>Reset Preference Defaults</td>
</tr>
<tr>
<td>Sens:Class:Name? Get_MeasurementClass</td>
<td>Returns the Measurement Class name</td>
</tr>
<tr>
<td>CALC:GCDAT:ITER TotalIterations</td>
<td>GCA - Returns number of iterations required in a SMART Sweep</td>
</tr>
</tbody>
</table>

### Gain Compression Setup

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:GCS:SWE:FREQ:POIN NumberOfFrequencyPoints</td>
<td>Number of frequency points</td>
</tr>
<tr>
<td>SENS:GCS:SWE:POW:POIN NumberOfPowerPoints</td>
<td>Number of power points</td>
</tr>
<tr>
<td>None</td>
<td>Maximum number of points</td>
</tr>
<tr>
<td>None</td>
<td>Total number of points</td>
</tr>
<tr>
<td>SENS:GCS:AMOD AcquisitionMode</td>
<td>Acquisition mode</td>
</tr>
<tr>
<td>SENS:GCS:SMAR:TOL SmartSweepTolerance</td>
<td>Smart tolerance</td>
</tr>
<tr>
<td>SENS:GCS:SMAR:MIT SmartSweepMaximumIterations</td>
<td>Smart Iterations</td>
</tr>
<tr>
<td>SENS:GCS:SMAR:STIM SmartSweepSettlingTime</td>
<td>Smart settling time</td>
</tr>
<tr>
<td>SENS:GCS:SMAR:SIT SmartSweepShowIterations</td>
<td>Smart show iterations</td>
</tr>
<tr>
<td>SENS:GCS:SFA? SearchFailures</td>
<td>Read compression failures</td>
</tr>
<tr>
<td>SENS:GCS:PORTM SetPortMap</td>
<td>Write port map</td>
</tr>
<tr>
<td>SENS:GCS:PORT DeviceInputPort</td>
<td>Read Port Map (Input)</td>
</tr>
<tr>
<td>SENS:GCS:PORT DeviceOutputPort</td>
<td>Read Port Map (Output)</td>
</tr>
<tr>
<td>SENS:GCS:EOS EndOfSweepOperation</td>
<td>End of Sweep</td>
</tr>
<tr>
<td>SENS:GCS:POW:LIN:INP:LEV InputLinearPowerLevel</td>
<td>Linear input power</td>
</tr>
<tr>
<td>SENS:GCS:POW:REV:LEV ReverseLinearPowerLevel</td>
<td>Reverse Power</td>
</tr>
</tbody>
</table>

The following are new programming commands for **PNA release 8.0**  See What's New
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stop power</td>
<td>SENS:GCS:POW:STOP:LEV</td>
<td>chan.Stop Power</td>
</tr>
<tr>
<td>Compression algorithm</td>
<td>SENS:GCS:COMP:ALG</td>
<td>CompressionAlgorithm</td>
</tr>
<tr>
<td>Compression Level</td>
<td>SENS:GCS:COMP:LEV</td>
<td>CompressionLevel</td>
</tr>
<tr>
<td>Backoff Level</td>
<td>SENS:GCS:COMP:BACK:LEV</td>
<td>CompressionBackoff</td>
</tr>
<tr>
<td>X Delta</td>
<td>SENS:GCS:COMP:DELT:X</td>
<td>CompressionDeltaX</td>
</tr>
<tr>
<td>Y Delta</td>
<td>SENS:GCS:COMP:DELT:Y</td>
<td>CompressionDeltaY</td>
</tr>
<tr>
<td>Interpolation</td>
<td>SENS:GCS:COMP:INT</td>
<td>CompressionInterpolation</td>
</tr>
<tr>
<td>Safe Sweep enable</td>
<td>SENS:GCS:SAFE:ENAB</td>
<td>SafeSweepEnable</td>
</tr>
<tr>
<td>Safe Sweep coarse</td>
<td>SENS:GCS:SAFE:CPAD</td>
<td>SafeSweepCoarsePowerAdjustment</td>
</tr>
<tr>
<td>Safe Sweep fine</td>
<td>SENS:GCS:SAFE:FPAD</td>
<td>SafeSweepFinePowerAdjustment</td>
</tr>
<tr>
<td>Safe Sweep threshold</td>
<td>SENS:GCS:SAFE:FTHR</td>
<td>SafeSweepFineThreshold</td>
</tr>
<tr>
<td>Read all GCA data</td>
<td>CALC:GCData:DATA</td>
<td>GetRaw2DData</td>
</tr>
<tr>
<td>Read real GCA data</td>
<td>CALC:GCData:REAL</td>
<td>GetDataIm</td>
</tr>
<tr>
<td>Read imaginary GCA data</td>
<td>CALC:GCData:IMAG</td>
<td>GetDataRe</td>
</tr>
</tbody>
</table>

### Noise Figure Setup

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create Noise figure meas</td>
<td>Calc:Cust:Def</td>
<td>CreateCustomMeasurementEx</td>
</tr>
<tr>
<td>Sets the number of impedance</td>
<td>SENS:NOIS:IMP:COUN</td>
<td>ImpedanceStates</td>
</tr>
<tr>
<td>states to use</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Noise averaging ON and OFF</td>
<td>SENS:NOIS:AVER:STAT</td>
<td>NoiseAverageState</td>
</tr>
<tr>
<td>Set averaging of noise receiver.</td>
<td>SENS:NOIS:AVER</td>
<td>NoiseAverageFactor</td>
</tr>
<tr>
<td>Set bandwidth of noise receiver.</td>
<td>SENS:NOIS:BWID</td>
<td>NoiseBandwidth</td>
</tr>
<tr>
<td>Set gain state of noise receiver.</td>
<td>SENS:NOIS:GAIN</td>
<td>NoiseGain</td>
</tr>
<tr>
<td>Sets noise tuner identifier</td>
<td>SENS:NOIS:TUN:ID</td>
<td>NoiseTuner</td>
</tr>
</tbody>
</table>
Sets the port identifier of the ECal noise tuner that is connected to the PNA Source.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:NOIS:TUN:INP</td>
<td>NoiseTunerIn</td>
</tr>
<tr>
<td>SENS:NOIS:TUN:OUTP</td>
<td>NoiseTunerOut</td>
</tr>
</tbody>
</table>

Set the port identifier of the ECal noise tuner that is connected to the DUT.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:NOIS:TUN:INP</td>
<td>NoiseTunerIn</td>
</tr>
<tr>
<td>SENS:NOIS:TUN:OUTP</td>
<td>NoiseTunerOut</td>
</tr>
</tbody>
</table>

Set the excess noise source ON or OFF.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONTrol:NOIs:SOURce or</td>
<td>NoiseSourceState</td>
</tr>
<tr>
<td>OUTPut:MANual:NOIs:[:STAte]</td>
<td></td>
</tr>
</tbody>
</table>

Set mechanical switches.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SENS:PATH:CONF:ELEM</td>
<td>PathConfiguration</td>
</tr>
</tbody>
</table>

Sets the default setting for the Noise Tuner switch.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYST:PREF:ITEM:SWIT:DEF</td>
<td>Port1NoiseTunerSwitchPresetsToExternal</td>
</tr>
</tbody>
</table>

### Noise Figure Cal

<table>
<thead>
<tr>
<th>Description</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create Noise Cal object</td>
<td>N/A</td>
<td>CreateCustomCalEx</td>
</tr>
<tr>
<td>Set Noise Calibration method</td>
<td>SENS:NOIS:CAL:METH</td>
<td>CalMethod</td>
</tr>
<tr>
<td>Noise source ENR filename</td>
<td>SENS:NOIS:ENR:FIL</td>
<td>ENRFile</td>
</tr>
<tr>
<td>Set noise source Cal Kit type</td>
<td>SENS:NOIS:SOUR:CKIT</td>
<td>NoiseSourceCalKitType</td>
</tr>
<tr>
<td>Set ambient temperature</td>
<td>SENS:NOIS:TEMP:AMB</td>
<td>AmbientTemperature</td>
</tr>
<tr>
<td>Sets noise source connector type</td>
<td>SENS:NOIS:SOUR:CONN</td>
<td>NoiseSourceConnectorType</td>
</tr>
</tbody>
</table>
### Noise Figure ENR File Data Management

<table>
<thead>
<tr>
<th>Operation</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set ENR calibration data.</td>
<td>SENS:CORR:ENR:CAL:TABLE:DATA</td>
<td>PutENRData</td>
</tr>
<tr>
<td>Read ENR calibration data.</td>
<td>SENS:CORR:ENR:CAL:TABLE:DATA?</td>
<td>GetENRData</td>
</tr>
<tr>
<td>Get/set ID of ENR table.</td>
<td>SENS:CORR:ENR:CAL:TABLE:ID:DATA</td>
<td>ENRID</td>
</tr>
<tr>
<td>Get/set serial number of noise source.</td>
<td>SENS:CORR:ENR:CAL:TABLE:SERIAL:DATA</td>
<td>ENRSN</td>
</tr>
<tr>
<td>Load ENR table from file.</td>
<td>MMEMory:LOAD:ENR</td>
<td>LoadENRFile</td>
</tr>
<tr>
<td>Save ENR table to file.</td>
<td>MMEMory:STORE:ENR</td>
<td>SaveENRFile</td>
</tr>
</tbody>
</table>

### Custom Cal Window

<table>
<thead>
<tr>
<th>Operation</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turn ON</td>
<td>OFF Custom Cal window.</td>
<td>SENS:CORR:COLL:DISP:WIND</td>
</tr>
<tr>
<td>Show NO Custom Cal windows.</td>
<td>SENS:CORR:COLL:DISP:WIND:AOFF</td>
<td>DisplayOnlyCalWindowDuringCalAcquisition</td>
</tr>
<tr>
<td>Specify channel to sweep before Cal acquisition.</td>
<td>SENS:CORR:COLL:SWE:CHAN</td>
<td>AllowChannelToSweepDuringCalAcquisition</td>
</tr>
<tr>
<td>Sweep NO channel before Cal acquisition.</td>
<td>SENS:CORR:COLL:SWE:CHAN:AOFF</td>
<td>SweepOnlyCalChannelDuringCalAcquisition</td>
</tr>
<tr>
<td>Preview sweep before remote Cal acquisition.</td>
<td>SENS:CORR:COLL:GUID:PACQUIRE</td>
<td>SetupMeasurementsForStep</td>
</tr>
</tbody>
</table>

### Miscellaneous

<table>
<thead>
<tr>
<th>Operation</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Trigger sweep mode</td>
<td>SENS:SWE:TRIG:MODE</td>
<td>Trigger Mode</td>
</tr>
<tr>
<td>Copy a Cal Set</td>
<td>SENSE:CORR:CSET:COPY</td>
<td></td>
</tr>
</tbody>
</table>
COM versus SCPI

There are two methods you can use to remotely control the PNA: COM and SCPI. The following topics can help you choose the method that best meets your needs:

- **Software Connection**
- **Physical Connection**
- **Programming Languages**

### Other Topics about COM Concepts

**Software Connection**

**COM** uses a binary protocol, allowing you to directly invoke a PNA feature. This is more efficient than SCPI. For example, the following statement calls directly into the PNA, executing the routine GetIDString.

```python
PNA.GetIDString()
```

**SCPI** is a text based instrument language. To retrieve the ID string, you would send the following text string to the PNA:

```text
IbWrite("*IDN?")
```

The PNA SCPI parser would first decode this text string to determine that the user has asked for the PNA to identify itself. Then the parser would call the COM method GetIDString().

### The Physical Connection

**Internal Control**

With either COM or SCPI, the best throughput is attained by using the PNA’s internal PC to execute your test code. However, if your test code uses too much system resources (CPU cycles and/or memory), this will slow the PNA’s performance.

Using the SICL I/O Libraries, you can also connect to the PNA from a program running on the PNA.

**External Control**

You can control the PNA from a remote PC using either COM or SCPI.

**COM** - (Component Object Model) can be used to access any program like the PNA (835x.exe) or library (.dll) that exposes its features using a COM compliant object model. These programs or libraries are called "servers". Programs (like your remote program on your PC) that connect to and use the features of these servers are called “clients.”

With COM, the server and the client do not need to reside on the same machine. DCOM, or distributed COM, makes the location of the server transparent to the client. When you access the PNA from a remote computer, you are using DCOM. In this case, the mechanical transport is a LAN (local area network).

However, using COM can add additional complexity:

- There are some DCOM security issues that may be a problem for you. Learn more.
Using the default interface when compiling type libraries results in code that will only run with the latest firmware. Learn more.

**SCPI** - Using a GPIB interface card in a remote computer, you can connect to the instrument using a GPIB cable. There are some constraints on the length of this cable and the number of instruments that can be daisy-chained together.

Using the Agilent SICL I/O libraries, you can connect to the instrument over a LAN connection.

(LAN or INTERNAL) You can send SCPI commands using COM with the `ScpiStringParser` object.

If you have legacy code written in SCPI for another network analyzer, you may be able to leverage that code to control the PNA. However, the PNA uses a different platform than previous Agilent Network Analyzers. Therefore, not all commands have a direct replacement. See the PNA Code Translator Application.

**Programming Languages**

You can program the PNA with either COM or SCPI using several languages. The most common include:

**Agilent VEE** - With this language you can send text based SCPI commands and also use automation. VEE 6.0 or later is recommended.

**Visual Basic** - This language has great support for automation objects and can be used to drive SCPI commands. The use of VISA drivers for your GPIB hardware interface will make the task of sending SCPI commands easier.

**C++** - This language can do it all. It is not as easy to use as the above two, but more flexible.

Last Modified:

24-Mar-2009 Updated
Remotely Specifying a Source Port

In the 'not-too-distant past', it was a simple task to specify a PNA source port. It was either port 1 or port 2. Now, for the following reasons, it is not so simple:

- **Internal 2nd sources** are now offered on various PNA models. However, some source ports do not have a port number. One example is the second source on the PNA-X 2-port model (option 224). Learn more about Internal Second Sources.

- **External sources** can now be controlled by the PNA as though they are internal sources. External sources do not have a source port number, but use String names as identifiers.
  
  - **For FCA ONLY**: Once configured using the Configuration dialog, an external source can be selected remotely and controlled by the PNA by specifying the LOName using SCPI or COM.
  
  - **All other uses for External sources**: The external source must be configured and selected from the External Source dialog. You can then save an Instrument State file, then recall that state file remotely.

- **Multiport test sets**…choose between ports 1 through port N, where N is the number of ports on the test set. You still use a port number, but this port number refers to a logical port. The Port mapping feature maps the logical port to a physical port. Learn more about Multiport test sets.

- **iTMSA (Opt 460)** When this option is present, the string names for balanced source ports are returned with the appropriate COM and SCPI commands. For example, "SE Port 1" is used to access 'Single-ended Port 1".

**Source Port String Names**

The PNA User Interface (UI) makes it easy to configure and select the sources and ports. Remotely however, string names are used now, in addition to port numbers, to specify a Source port.

**COM** - The existing COM commands specify source ports as numbers and they are still used. It is necessary to learn the port number from the string using the GetPortNumber Method. Port numbers are assigned dynamically depending on whether external sources are selected and the number of ports of the PNA.

- **SourcePortNames Property**

- **GetPortNumber Method**

- **SourcePortCount Property**

An example:

```vbs
dim app
set app = CreateObject("Agilentpna835x.application")
dim channel
set channel = app.Channel
dim portnum
portnum = Channel.GetPortNumber("Src2 Out1")
app.CreateMeasurement 1,"A",portnum
```
SCPI - ALL of the existing SCPI commands that specify a source port are extended to also allow the source port to be specified using string names. For example, send the following command to set the power on Src2 Out1:

- **SOUR:POW 5, “Src2 Out1”**
- Use **Source:Cat?** to list the available source port string names.

---

Last Modified:

23-May-2008   Added iTMSA option
24-Apr-2008   Several edits
5-Nov-2007   MX New topic
Shut Down or Restart the PNA Remotely

To Shut down, send this command:

```plaintext
Shutdown -s -t: 00
```

To Restart, send this command:

```plaintext
Shutdown -r -t: 00
```

These will wait 00 seconds before executing the command.

1. Paste the command into a Notepad file.

2. Save the file as `shutdownPNA.bat` anywhere on the PNA, preferably in the C:/Windowns/system 32 directory.

3. To call the file using GPIB, issue this SCPI command:

```plaintext
Diag:batch 'shutdownPNA.bat'
```

If you have placed the file somewhere on the PNA other than the system32 directory, include the full path to the batch file; such as:

```plaintext
Diag:batch 'C:/Program Files/Agilent/Network analyzer/Documents/shutdownPNA.bat'
```

For variations on this command, on any Windows computer:

Click **Start**, then **Run**, then type **cmd**, then **OK**.

At the prompt, type **shutdown**.

Last Modified:

22-May-2009   New topic
Frequency Converter Application (Option 083)

FCA includes both Scalar (SMC) and Vector (VMC) measurements and calibrations.

In this topic:

- What's New in FCA
- FCA Options Explained
- Comparison of VMC and SMC
- Requirements and Limitations
- How to make SMC or VMC Measurements
  - Create a Measurement
  - Make Measurement Settings
    - Sweep Tab
      - Segment Sweep
    - Power Tab
    - Mixer Setup Tabs (separate topic)
    - Select X-axis Display
    - Save Trace Data
    - Avoid Spurs

See Also

- SMC Measurements and Calibrations
- VMC Measurements and Calibrations
- Configure an External LO Source
- SMC with a Booster Amp
- Characterize Adaptor Macro
- Measure a DUT with an Embedded LO

- For a detailed understanding of FCA, see our Mixer Measurements App Notes.
Examples

- How to make a VMC Measurement
- How to make an SMC Measurement

Other Application topics

What's new in FCA
Rev A.09.33

- New configuration dialogs
- Segment Sweep
- Power Sweep
- Save CSV Trace Data

FCA Options Explained

- Option 083 provides FCA which includes Scalar Mixer (SMC) and Vector Mixer (VMC) Measurements.

- Option 082 provides ONLY SMC measurements. This is the ONLY FCA option that is allowed on the N5230C.

- Option 084 provides Embedded LO measurements. This option requires one of the Converter Applications.

- See all PNA-X Options and Configurations.

Comparison of SMC and VMC
| Overview | Scalar Mixer Calibration  
See Hardware setup | Vector Mixer Calibration  
See Hardware setup |
|---|---|---|
| Provides highest Scalar accuracy for measurements of conversion loss/gain.  
**Optionally measures phase**  
Combines SOLT and power-meter calibration.  
Simpler setup than Vector Mixer Calibration. | Provides unparalleled accuracy for measurements of relative phase and absolute group delay.  
Uses combination of SOLT standards and a reciprocal mixer/filter pair during calibration.  
More complicated setup and calibration procedure than Scalar Mixer Calibration.  
After calibration, both reciprocal and non-reciprocal mixers and converters can easily be measured. |
| Measurements Offered | Both forward and reverse directions.  
DUT can be connected to any PNA ports. | Amplitude response VC21  
Phase response  
Group delay  
DUT input must be connected to PNA port 1.  
DUT output can be connected to any other PNA port. |
| Equipment Required | Power meter and sensor | Reference mixer  
Calibration mixer/filter combination (must be reciprocal  \( S_{21} = S_{12} \)) |
| Common equipment for both SMC and VMC | |  
• Mechanical cal kit or ECal module |


**Requirements and Limitations**

The following PNA-X features are **NOT** available with FCA:

- Analog Sweep ([Stepped sweep](#) mode only)
- [Log frequency](#) sweeps
- [Unratioed receiver measurements](#) (A, B, R)
- ECal User Characterization
- **Time Domain**
- **Balanced measurements**
How to make SMC or VMC Measurements

The following is an overview of how to make an FCA measurement:

1. DECIDE to use either a SMC or VMC measurement. See a comparison of these two measurement types.
2. CREATE an SMC or VMC Measurement.
3. SETUP the measurements.
4. CALIBRATE your SMC or VMC measurement.

Create an SMC or VMC Measurement

1. On the PNA-X front panel, press Meas then [Measurement Class]
2. Select SMC or VMC, then either:
   - OK delete the existing measurement, or
   - New Channel to create the measurement in a new channel.
3. The default SMC or VMC measurement is displayed.
4. See SMC measurements or VMC measurements to learn about the parameters that are offered in each.
**How to make SMC or VMC settings**

To provide quicker access, use the Setup softkey. [Learn how.](#)

### Using front-panel HARDKEY [softkey] buttons

1. Press **Freq**
2. then [Mixer Setup]

### Using a mouse with PNA Menus

1. Click **Stimulus**
2. then **Freq**
3. then **Mixer Setup**

---

**Valid Mixer Configuration / Sweep Type Combinations**

Configuring the SMC and VMC Setup dialog can be challenging at first. **RED** messages like this one appear at the bottom of the Setup dialog to notify you of an invalid setup.

Unsupported mixer configuration and sweep type

At least one range (Input, LO, or Output) MUST be Fixed.

The following are the **Valid Mixer Configurations**:

<table>
<thead>
<tr>
<th>Sweep Type</th>
<th>Input</th>
<th>LO</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>Swept</td>
<td>Swept</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Swept</td>
<td>Fixed</td>
<td>Swept</td>
</tr>
<tr>
<td></td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
<tr>
<td>CW Time</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
</tbody>
</table>

---

3228
**Tips**

Although you will soon become comfortable navigating these tabs, at first it may be best to complete the dialog in the following order:

1. For 2-stage mixers, select **Mixer Setup** settings.
2. Select **Sweep tab** settings.
3. Select **Mixer Frequency** settings.
4. Select **Power** settings.
5. Select **Mixer (LO) Power** settings.

The following FCA settings are common to VMC and SMC:

**Sweep Tab - SMC and VMC** dialog box help

**Sweep Type**

- **Linear** Sweep frequency. Measurements are displayed on a standard grid with ten equal horizontal divisions. Learn how to [select the range to display on the X-Axis](#).
- **CW Time** All ranges are set to a Fixed (CW) frequency, and the data is displayed versus time.
- **Segment Sweep** Sweep user-defined segments. [Learn more](#).
- **Power** Sweep Input or LO power.

**X-axis Point Spacing** (Available only with Segment Sweep) - [Learn about this feature](#)

**Avoid Spurs** - [Learn about this feature](#).

**Number of Points** [Learn about this feature](#).

**IF Bandwidth** [Learn about this feature](#).

**Phase Reference Point** (SMC ONLY) [Learn about this feature](#).
**Power Tab - SMC and VMC dialog box help**

**Note:** Set LO Power on the **Mixer (LO) Power tab**.

Configures Input and Output power settings for an FCA measurement. Use the **Mixer Power tab** to set LO power.

**Power ON (All channels)** Check to turn RF Power ON or clear to turn power OFF for all channels.

**Port Powers Coupled** Check to set the same power level at the DUT Input and Output ports. The LO power is NOT coupled. Clear to set power levels independently for each test port. Uncouple power, for example, to apply more power in the reverse direction than in the forward direction. Learn more about **Setting Independent Port Power**

**DUT Input / Output Port**

Select the PNA port that is connected to the DUT Input and Output. For **VMC**, the DUT input must always be connected to PNA port 1 because of the need for a reference mixer on port 1.

**Power Level** Set the power level to the DUT Input port. To set power at the Output port, clear the **Port Powers Coupled** checkbox.

**Source Attenuator Auto** Check to automatically select the correct attenuation to achieve the specified input power. Clear, then select attenuator setting that is used achieve the specified Power Level. Learn more about **Source Attenuation**.

All PNA channels in continuous sweep must have the same attenuation value. Learn more.

**Receiver Attenuator** Specifies the receiver attenuator setting for the DUT port.

**Source Leveling** Choose from: **Internal** (normal operation), **Open Loop** (used only for Wideband Pulse measurements), or **Receiver - R1** for **Receiver Leveling**.

**DUT Input and Output Port Power Sweep**

Available when Power (sweep) is selected on the **Sweep tab**.

**Input Start** and **Stop Power** To set Start and Stop power at the Output port, clear the **Port Powers Coupled** checkbox.
Note: If your DUT requires more input power than this setting allows below 3.2 GHz, use the PNA-X Hi-power mode, available from the RF Path Configuration dialog. The disadvantage to this is higher harmonic content.

**Power Points**  Number of power points to measure.

**Power Step (Size)**  Calculated value from current Start, Stop, and Points settings. This setting can NOT be changed directly.

**Path Configuration**  click to launch the RF Path Configuration dialog.

The following tabs are shared with all Mixer / Converter Applications:

- **Mixer Frequency tab**
- **Mixer (LO) Power tab**
- **Mixer Setup tab**

**Mixer Frequency tab - SMC and VMC Setup**

Learn about this dialog
FCA Segment Sweep
The following settings appear on the Mixer Frequency tab when Segment Sweep is selected on the Sweep tab.
How to configure a segment:

1. Click **Add**. Click **Delete** to remove a segment and renumber all subsequent segments.

2. State is **ON** by default. Click **OFF** and that segment will not be included in the sweep.

3. Configure **Frequency settings** for Input, LO, and Output ranges.

   - For each segment, the same **sweep requirements** apply as a standard (non-segment) sweep. For example, at least one range MUST be Fixed (Start = Stop frequencies).

   - The Input, Output, and LO frequencies of segments ARE allowed to overlap other segments.

   - All segments must sweep in either the forward (Start<Stop) or reverse (Start>Stop) directions. Mixed sweep directions are NOT allowed.

   - The following settings can be set independently for each segment:
     - **Number of Points** - Total number of points for all segments is limited to the **Max allowed by the PNA**.
     - **IF Bandwidth**
     - **Port Powers**: (Input, Output, LO 1, LO2). These settings override the settings on the **Power tab**.

   - The following settings apply to **ALL** segments:
     - Number of Converter/Mixer Stages (1 or 2).
     - LO Source Selections
     - **All Input and LO Multipliers and Dividers**
     - **Source and Receiver Attenuator Settings**
• **Source Leveling**

• **Avoid Spurs**

• **Nominal Incident Power** (SMC only)

• **X-Axis Display** (Input, LO1, LO2, Output) There must be at least two data points for this setting to be available.

• **X-Axis Point Spacing** (vs Normal point spacing).

• **SMC + Phase**

  • Mixer Segment sweep data can be saved to a *.S2PX file (NOT *.S2P).
  
  • Mixer Segment setup information is saved to a *.MXRX file. [Learn more](#).

---

### Apply and Interpolate FCA Cal Sets

In general, when a Cal Set covers a wider frequency range than the channel, the PNA will offer to interpolate the Cal Set when it is applied. [Learn more](#). However, with FCA measurements the LO frequency range may also be considered.

- **VMC measurements** ALWAYS CONSIDER the LO frequency range and performs interpolation if possible. If the LO frequency range of the measurement is NOT within the LO frequency range of the Cal Set, then the Cal Set can NOT be applied.

- **SMC measurements** ALWAYS IGNORE the LO frequency range. Therefore, if the Input and Output frequency ranges of the measurements are within those of the Cal Set, then the Cal Set is interpolated if necessary and applied. For example, this would allow you to perform ONE SMC calibration with Input range = the PNA frequency span, LO at 0 Hz, and Output range + the PNA frequency span. This Cal Set could be applied to ALL SMC measurements. [Learn more about applying SMC Cal Sets](#).

These same general concepts apply to **segment sweeps**. However, if ALL applicable frequency ranges (SMC: Input and Output and VMC: Input, Output, and LO) are NOT within those ranges of the measurement for ONE segment, then the Cal Set is NOT applied for ANY segment.

---

### Select X-axis Display for FCA Measurements

Click **Response**, then **Measure**, then **Select X-axis**, then **Input**, **LO**, or **Output**

When **Sweep Type = Linear**, you can choose to show the frequency range of any of the swept parameters on the X-axis.

For example, the following image shows an SMC Fixed Output response with the **Input frequency range** on the X-axis:

- Output: 100 MHz (data trace)
- **Input**: 2 GHz to 23 GHz (X-axis)
LO: 1.9 GHz to 22.9 GHz (not shown)
Marker annotation shows Output power at Input frequency.

Save Trace Data
You can save your FCA measurement data in several standard formats.
Click File, then Save Data As to save your FCA data.
The following shows how CSV and SNP files are saved.

Mixer Trace Data
When you select Mixer Trace Data, the FCA data is saved to a CSV file in the following format:

#MIXER TRACE FILE,A.01.00
SegIndex, InputFreq, OutputFreq, LO1Freq, InputPower, LO1Power, SC21 Mag (dB), SC21 Phase (Deg)

SNP Format
Each record contains 1 stimulus value and 4 parameters (total of 9 values) as follows:
Stim Real(p1) Imag(p1) Real(p2) Imag(p2) Real(p3) Imag(p3) Real(p4) Imag(p4)
where pX is the parameter depending on measurement type:

<table>
<thead>
<tr>
<th>Measurement Type</th>
<th>p1</th>
<th>p2</th>
<th>p3</th>
<th>p4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scalar</td>
<td>S11</td>
<td>SC21 (FWD)</td>
<td>SC12 (REV)</td>
<td>S22</td>
</tr>
<tr>
<td>Vector</td>
<td>S11</td>
<td>VC21</td>
<td>VC12</td>
<td>S22</td>
</tr>
<tr>
<td>Mixer Characterization</td>
<td>Directivity</td>
<td>Source Match</td>
<td>Reflection Tracking</td>
<td>M21</td>
</tr>
</tbody>
</table>
If correction is OFF, data is only saved for the active parameter. Zeros are saved for all other parameters.

If correction is ON, data is saved for all of the parameters.

All files contain the following Header Information: Brackets [ ] contain parameters.

!Agilent [Instrument Model Number]: [version]
!Mixer S2P File: [Mixer Measurement Type]
!Parameters: [Parameter List]
!Calibration State: [On/Off]

!# Begin Mixer Setup
![[Mixer Setup parameters listed here]
![[Mixer Parameter 1]
  .
  .
![[Mixer Parameter n]
!# End Mixer Setup

# [S2P data here]

Avoid Spurs

On the Mixer Setup dialog box, check Avoid Spurs

The Avoid Spurs feature of the Frequency Converter Application attempts to prevent unwanted mixing products from appearing on the PNA screen. The Avoid Spurs feature does not significantly impact measurement speed.

**Note:** The Avoid Spurs feature is OFF by default for FCA calibrations. For highest accuracy, make measurements with the Avoid Spurs feature at the same state (ON or OFF) as was used when calibrating.

Description

A spur, or spurious signal, is a term used to describe the unwanted product of two signals mixing together. When you configure the mixer setup dialog box for a desired Output, the PNA computes the frequencies of potential unwanted signals. By manipulating internal PNA hardware, these signals are avoided and do not appear on the PNA display. This means you do not need to use external filters to prevent spurious signals from appearing on the PNA display.

The time required for the PNA to compute the frequencies of unwanted spurious signals MAY be noticeable depending on the number of data points in your measurement. However, once computed, the time required for the PNA to avoid the spurs is usually insignificant.

Limitations

The Avoid Spurs utility cannot avoid every spur. However, when there is a choice of spurs to avoid, it will avoid the largest spur.

The Computation of Avoided Spurs

The Avoid Spur computer avoids the following spurs:

- LO, and its interaction with internal PNA components, and 16 of its harmonics.
- Input frequencies and 16 of its harmonics.
- Undesired Image frequencies (Sum or Difference) and 16 of its harmonics.

Last modified:

- **27-Apr-2011** Removed Copy Channels limitation
- **9-Dec-2010** Updated for A.09.33
- **26-Aug-2010** Add link to Cal 2-stage SMC+Phase
- **25-Mar-2010** Add SMC with phase, Data Save As, GAV
- **5-Feb-2010** Add Speed SMC
- **3-Sep-2008** Removed legacy content
- **1-May-2008** Updated for selectable ports
- **5-Oct-2007** Added link to embedded LO
Scalar Mixer/Converter Measurements (SMC)

SMC Setup and Calibration is very similar to VMC. See FCA Overview to learn about the features that are common to these two applications.

The following information is unique to SMC:

- SMC Hardware Setup
- Create an SMC Measurement
- SMC Parameters Offered
- The SMC Mixer Setup dialog
- SMC + Phase
- SMC Calibration

See Also

Embedded LO
How to make an SMC Fixed Output Measurement
SMC with a Booster Amp
Programming Commands

SMC Hardware Setup
SMC requires a **power meter/sensor**, **two sources**, and a **Cal Kit or ECal module**

- Your DUT can be connected to any PNA ports. [Learn more.](#)
- When using PNA-X with Internal Second Source, the external source is NOT necessary. [Learn which PNA ports can be used for the LO.](#)
- Connect **External Source** to the PNA GPIB Controller port. Learn how to [Configure an External LO Source](#)
- See the rear panels of the [PNA-X](#) or [PNA ’C’](#) models.
Use either a GPIB power meter or USB power sensor.

**How to configure two power sensors** to cover the SMC measurement frequency range.

Using a dual channel power meter, with both sensors connected:

1. At the SMC **Select DUT Connectors** dialog, click **Source Cal Settings**
2. At **Source Calibration Settings** dialog, click **Power Meter Config**
3. At **Power Meter Settings** dialog, click **Sensors**
4. At **Power Sensor Settings** dialog, clear the "Use this sensor only..." checkbox for both sensors.
5. Then enter the **Min** and **Max Frequencies** for both sensors.

During the SMC Cal, you will be prompted to connect each sensor at the appropriate time.

---

**Create an SMC Measurement**

1. On the PNA-X front panel, press **Meas** then **[Measurement Class]**
2. Select **SMC**, then either:
   - **OK** delete the existing measurement, or
   - **New Channel** to create the measurement in a new channel.

3. An SC21 measurement is displayed.

**SMC Parameters Offered**

To select additional parameters to display, click **Trace**, then **New Trace**, then select a parameter from the list.

---

**Important Note:** Connecting your DUT to the PNA:

**RF** and **IF** terminology is NOT used in FCA because the PNA does not know how the DUT is labeled or how it will be used. Instead, the general terms **INPUT** and **OUTPUT** are used.

- **INPUT** - The DUT port being stimulated with frequencies before conversion.
- **OUTPUT** - The DUT port outputting converted frequencies.

**INPUT** and **OUTPUT** Frequencies are specified using the **Mixer Setup dialog box**.
The DUT input and output can be connected to any PNA ports.

**Note**: Although there are MANY configuration possibilities, the following images and descriptions show ONLY a DUT connected to PNA ports 1 and 2.

Legend:
- **Black** are ratioed measurements (test port/reference receiver).
- **Green** are unratioed measurements (either a test port OR reference receiver).

<table>
<thead>
<tr>
<th>DUT <strong>Input</strong> to PNA port 1</th>
<th>DUT <strong>Output</strong> to PNA port 2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Ratioed</strong></td>
<td><strong>Ratioed</strong></td>
</tr>
<tr>
<td>- SC21 (Conversion Loss)</td>
<td>Stimulus at Input,</td>
</tr>
<tr>
<td></td>
<td>response at Output (B/R1).</td>
</tr>
<tr>
<td>- SC12 (Reverse Isolation)</td>
<td>Stimulus at Output,</td>
</tr>
<tr>
<td></td>
<td>response at Input (A/R2)</td>
</tr>
<tr>
<td>- S11 (Input match)</td>
<td>Stimulus and response at</td>
</tr>
<tr>
<td></td>
<td>Input (A/R1)</td>
</tr>
<tr>
<td>- S22 (Output match)</td>
<td>Stimulus and response at</td>
</tr>
<tr>
<td></td>
<td>Output (B/R2)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DUT <strong>Input</strong> to PNA port 2</th>
<th>DUT <strong>Output</strong> to PNA port 1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Unratioed</strong> Absolute test port receiver measurements. The receiver is automatically selected depending on the DUT configuration.</td>
<td></td>
</tr>
<tr>
<td>- <strong>IPwr</strong> (Incident Power) - stimulus and response at Input.</td>
<td></td>
</tr>
<tr>
<td>- <strong>RevIPwr</strong> (Reverse Incident Power) - stimulus and response at Output.</td>
<td></td>
</tr>
<tr>
<td>- <strong>OPwr</strong> (Output Power) - stimulus at Input, response at Output.</td>
<td></td>
</tr>
<tr>
<td>- <strong>RevOPwr</strong> (Reverse Output Power) - stimulus at Output, response at Input.</td>
<td></td>
</tr>
</tbody>
</table>
SMC Mixer Setup

How to start the SMC Mixer Setup dialog
To provide quicker access, use the Setup softkey. Learn how.

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press Freq</td>
<td>1. Click Stimulus</td>
</tr>
<tr>
<td>2. then [Mixer Setup]</td>
<td>2. then Freq</td>
</tr>
<tr>
<td></td>
<td>3. then Mixer Setup</td>
</tr>
</tbody>
</table>

The following SMC Mixer Setup dialog tabs are presented:

- **Sweep Tab** (shared with VMC)
- **Power Tab** (shared with VMC)
- **Mixer Freq Tab** (shared with all converter apps)
- **Mixer Power Tab** (shared with all converter apps)
- **Mixer Setup Tab** (shared with all converter apps)

SMC + Phase

Beginning with PNA Rev.A.09.20, you can optionally measure phase with SMC. This feature is available ONLY with Opt 083 on a PNA-X or N522x model.

A Reference Mixer is NOT required as it is with VMC. Instead, during an SMC calibration, the phase delay through a characterized Cal Mixer is measured and compared to the known delay. The difference between the measurement and the known delay is used to correct all subsequent SMC phase measurements.

Comparing SMC+Phase with VMC Phase Measurements

SMC phase measurements do NOT require a reference mixer, and are therefore easier to make than VMC phase measurements. Also the SMC calibration mixer is only required to have a known delay value, although an S2P characterization file provides more accurate results.

- When measuring converters with an embedded LO, SMC with phase can provide results that are as stable and free from sweep-to-sweep jitter as VMC.

- When measuring converters with an external LO that is shared with the reference mixer (as shown in the VMC Setup diagram), VMC provides results that are more stable than SMC+Phase.
How to enable Phase with SMC

On the **Sweep tab**, check **Enable Phase**.

To View Phase (using any parameter): click **Response**, then **Format**, then select the format of interest (phase, unwrapped phase, group delay, and so forth).

<table>
<thead>
<tr>
<th>Phase Settings dialog box help</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Phase Settings dialog box" /></td>
</tr>
</tbody>
</table>

**Enable Phase with SMC**  Check to perform phase measurements. You can also enable phase measurements during calibration.

**Phase Reference Point**

The SMC Phase measurement technique provides for a coherent phase relationship from one frequency to the next in each sweep. However, the phase measurement of the first data point is random from sweep to sweep. This initial phase offset does not impact measurements such as group delay or deviation from linear phase. However, in order to keep a phase trace from appearing random, all phase data in the sweep is normalized against a single point. This results in a stable, normalized phase trace.

For this normalization, select the measurement point that has the best signal-to-noise ratio. The phase at the selected point will always be zero. This selection applies for both the measurement and the calibration sweeps. The Reference Point is fixed at the middle point when **segment sweep** is selected.

**How to improve the stability of SMC+Phase measurements**

Stable phase measurements are attained by increasing **Sweep Averaging**, and sometimes lowering the IFBW, until you attain the desired compromise between sweep time and trace jitter (the amount of random phase change at a single data point). For SMC + phase, the default IFBW is 10 kHz, and 1 average. During calibration, the Averaging factor is temporarily multiplied by 4 to ensure an accurate phase calibration.

The following procedure shows how to view and improve phase jitter:

1. Create an SMC + Phase channel (Click Response, Measurement Class).
2. Enable Phase with SMC (See above) On the phase trace (to follow) notice that the only point that has NO jitter is the data point that you selected as the Phase Reference point.
3. Change the measurement to **IPWR**: (Click Response, Measure, IPWR)
4. Change Format to Phase. (Click Response, Format, Phase)
5. Normalize the trace. (Click Marker/Analysis, Memory, Normalize) Learn more about Normalization.
6. Autoscale the trace. (Click Response, Scale, Autoscale).
7. Optionally monitor the jitter with **Trace Statistics** (Std Dev)
8. Increase Averaging and possibly lower IFBW to improve jitter. (Click Response, Avg, then Average and IFBW).

9. After the adjustments are made, change the measurement back to your measurement of interest.

10. When measuring a new DUT, restart Averaging.

**How to Calibrate a 2-stage (LO) SMC+Phase Measurement**

When calibrating a dual-stage SMC+Phase measurement for group delay using a characterized mixer, the channel setup requires frequency values for two LOs, but the characterized-thru mixer uses only one LO. The frequencies of LO1 and LO2 are different. There are two ways to overcome this challenge:

1. Before the calibration, set the LO that is provided by an external source to uncontrolled. Then manually set the frequency of this external source to the LO frequency that gives the same input and output frequencies, and the same sweep direction, as the dual-stage setup. Perform the calibration under this condition. After the calibration, return the LO to controlled so that its frequency will be properly set during the measurement of the DUT.

2. Configure a 1-stage mixer setup, with the LO set to the frequency that gives the same input and output frequencies and the same sweep direction as the dual-stage setup. Perform the calibration under this condition. Save the calibration data as a user calset. Configure the dual-stage case, and apply the 1-stage calibration.

**Speed Up SMC Measurements**

Using default SMC settings, any calibrated SMC measurement requires four sweeps. However, you can reduce the number of sweeps required by selecting one or more of the following settings.

- **Use Nominal Incident Power**
- **Apply Cal Set or Cal Type**
- To speed up a Swept LO measurement when using an external source for the LO, use **Hardware List (BNC)** Trigger setting. Learn more.

**Use Nominal Incident Power**

Click **Response**, then **Measure**, then **Use Nominal Incident Power**

Each data sweep of a fully corrected SMC transmission measurement actually requires FOUR data sweeps. When you clear **Use Nominal Incident Power**, the reference receiver (R1 or R2) does not measure incident power. Instead, the incident power is assumed to be at the level that was set with the **Source Power Calibration** that is done as part of every SMC measurement. The degradation in accuracy is very negligible if the input or output of your test device is well-matched.

This selection eliminates sweeps ONLY when both **Include Input Match** AND **Include Output Match** is cleared on the Cal Type dialog. Learn more.

**Apply a Cal Set or SMC Cal Type**

You can create an FCA measurement and apply an existing Cal Set as you can with any PNA measurement. Learn
about Cal Sets. In addition, from a Cal Set, you can apply a specific SMC Cal Type to an existing SMC measurement.

**How to apply an SMC Cal Type**

1. Create an SMC measurement
2. Calibrate or apply an existing SMC Cal Set, then...

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then [Manage Cals]
3. then [Correction Methods]

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Cal**
3. then **Manage Cals**
4. then **Correction Methods**

**Correction Method** dialog box help

By default, each SMC calibration requires FOUR sweeps. Clearing boxes will eliminate sweeps and speed up your SMC measurements. The difference in speed is most noticeable when making fixed input or fixed output measurements with an external LO source.

**Include input match correction** Check to perform a sweep to measure and correct for INPUT match. Clear this box if the input of your mixer is well-matched to the PNA, or if your setup does not permit a valid S11 measurement.

**Include output match correction** Check to perform a sweep to measure and correct for OUTPUT match. Clear this box if the output of your mixer is well-matched to the PNA, or if your setup does not permit a valid...
Include SC12 Sweep  Check to perform a reverse sweep to measure SC12.

- When checked (default setting), a calibrated SMC measurement sweeps in both forward (SC21) and reverse (SC12) directions.
- Clear this checkbox to eliminate sweeps in the reverse direction. This means that the following measurements will NOT be corrected: SC12, RevOPwr, RevIPwr.

Corrected Parameters  Lists the parameters that can be corrected given the boxes that are currently checked. These parameters may not be currently measured.

Calibration Type  Shows the type of SMC Cal that will be applied given the boxes that are currently checked.

Learn about Use Nominal Incident Power

How many sweeps can be eliminated?

<table>
<thead>
<tr>
<th>Setting</th>
<th>Parameters</th>
<th>Learn about parameter abbreviations</th>
<th># of sweeps</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALL checked and clear Use Nominal Incident Power</td>
<td>IPwr, OPwr, RevIPwr, RevOPwr, SC12, SC11, S11, S22</td>
<td>Total</td>
<td></td>
</tr>
</tbody>
</table>

Perform this action... to REMOVE these parameters... and these sweeps

<table>
<thead>
<tr>
<th>Setting</th>
<th>Parameters</th>
<th>Learn about parameter abbreviations</th>
<th># of sweeps</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clear &quot;Include SC12“</td>
<td>Remove RevIPwr, RevOPwr, SC12</td>
<td></td>
<td>Removes 1</td>
</tr>
<tr>
<td>Clear &quot;Include OUTPUT match“</td>
<td>Remove S22</td>
<td></td>
<td>Removes 1 when Nominal is checked*</td>
</tr>
<tr>
<td>Clear &quot;Include INPUT match“</td>
<td>Remove S11</td>
<td></td>
<td>Removes 1 when Nominal is checked*</td>
</tr>
<tr>
<td>Check &quot;Use Nominal Incident Power“</td>
<td>Remove IPwr, RevIPwr</td>
<td></td>
<td>May remove up to 2*</td>
</tr>
</tbody>
</table>

ALL cleared and check Nominal Incident Power | OPwr, SC11 | 1 Total |

*S11 shares a sweep with IPwr and S22 shares a sweep with RevIPwr. Therefore, when Include Input Match or Include Output Match is checked, then checking Nominal incident power does nothing.

*SMC measurement* sweeps can NOT be eliminated.

SMC Calibration Overview

The SMC Calibration Wizard guides you through this process.

1. Connect a power meter / sensor to PNA Port 1. At each step of the input and output frequency, the PNA measures:
• input match of the power sensor
• source power of the PNA

2. Perform two 2-port SOLT calibrations: one over the INPUT frequencies and one over the OUTPUT frequencies of the DUT. (If your DUT is a linear device, the calibration uses only the INPUT frequency range.) Use either a mechanical calibration kit or an ECal module.

SMC Cal Wizard

The following dialog boxes are presented during SMC Calibration.

Indented steps are optional.

- Calibration Setup
  - Waveguide/In-fixture/On-Wafer Setup
- Select DUT Connectors and Cal Kits
  - Modify Frequency Cal
  - Specify how the ECal module is connected
- Power Cal Settings
- SMC Cal Steps
- Calibration Completed
- Specify Adapter Delay

How to Perform a SMC Calibration

1. Create an SMC measurement, then...

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>CAL</strong></td>
<td>1. Click <strong>Response</strong></td>
</tr>
<tr>
<td>2. then <strong>Start Cal</strong></td>
<td>2. then <strong>Cal Wizard</strong></td>
</tr>
<tr>
<td>3. then <strong>Cal Wizard</strong></td>
<td></td>
</tr>
</tbody>
</table>
SMC Calibration Setup dialog box help

Allows you to review and change the settings for your SMC calibration.

Waveguide/In-fixture/On-Wafer Setup  Click Next to launch the following Setup dialog box.

Independent power cal for input and output ports (no thru)  Check if a Thru standard is NOT available. During the power cal, you will be prompted to connect the power sensor to the Input, then the Output port.

Additional Power Cal Steps

Enable LO1 / LO2 Power Cal  Check when LO1 / LO2 is controlled (on the Mixer Setup tab) to perform a Power Cal on the LO source(s).

Phase Correction

Enable Phase Correction  Check to enable Phase measurements.

During an SMC calibration, the phase delay through a characterized Cal Mixer is measured and compared to the known delay. The difference between the measurement and the known delay is used to correct the SMC phase measurements.

Using an *.S2P file (which characterizes the delay across the measurement frequency range) is more accurate than using a single known delay value.

Use an *.S2PX file when making segmented SMC + Phase measurements. Learn more.

The following are two methods to characterize the Cal Mixer over the SMC measurement frequency range:

1. Use the Mixer Characterization Wizard. The Cal Mixer has the same requirements as the VMC Cal Mixer. Learn more.

2. In a calibrated VMC channel, measure the group delay of the calibration mixer, then save to an *.S2P or *.S2PX file. However, a characterized mixer is required to calibrate the VMC channel.

To start the Mixer Characterization Wizard, click Response, then Cal, then Mixer Characterization Wizard. Learn more about Phase Correction for SMC measurements.
Waveguide/In-fixture/On-Wafer Setup dialog box help

This dialog box appears ONLY if you checked the Waveguide/In-fixture/On-Wafer Setup box in the previous Cal Setup dialog.

Allows you to embed or de-embed circuit networks on the input and output of your mixer under test.

For Network1 (Input) and Network2 (Output) select Embed, De-embed, or None.

Browse  Click to navigate to the .S2P file that models the network to embed or de-embed.

Reverse port positions for input/output  Check to cause the Fixture/Adapter to be configured with Port 2 connected to the PNA and Port 1 to be connected to the DUT. The image in the dialog is updated to reflect that change.

Enable Extrapolation  Check (default setting) to apply a simple extrapolation when the S2P file has a narrower frequency range than the channel. The values for the first and last data points are extended in either direction to cover the frequency range of the measurement. A warning message is also displayed when extrapolation is necessary.

To Embed or De-embed

- When you have a 2 port network that needs to be connected between the Cal reference plane and the DUT during the measurement, but it is NOT present during the calibration, then that network has to be De-Embedded from the port in question during the calibration. In other words, De-Embedding in FCA calibration extends the calibration reference plane to include the two port network.

- When you have a 2 port network that is included as part of the calibration reference plane but has to be disconnected during the measurement, then that 2-port network has to be Embedded for the port in question during the calibration. In other words, Embedding in FCA calibration retracts the calibration reference plane to exclude the two port network during the measurement.

Notes

- Characterize Adaptor Macro can be used to create the S2P file.

- Interpolation is performed when more frequencies are included in the file than in the channel, and the data points do not exactly match those of the measurement.
Select DUT Connectors and Cal Kits dialog box help

Allows you to specify the connector type and Cal Kit for each DUT port.

**Port n** For each listed PNA port, specify the DUT connector type and gender, and the Cal Kit to use.

**Note:** If your DUT connectors are:

- Waveguide  Change the system impedance to 1 ohm before performing a calibration. See Setting System Impedance.

- Not listed (male and female)  Select **Type A** as the connector type. Type A requires a calibration kit file containing the electrical properties of the standards used for calibration (see Calibration kits).

- Unspecified (like a packaged device)  Select **Type B** as the connector type. Type B requires a calibration kit file containing the electrical properties of the standards used for calibration (see Calibration kits).

**Modify Cal**  Check, then click **Next**, to start the Modify Frequency Cal dialog.

**Source Cal Settings**  Click to start the Source Cal Settings dialog.

Modify Frequency Cal dialog box help

**Thru Cal Method**  For each Thru connection, choose the Thru method. Learn more about these choices.

**Cal Type/Stds**  Click to start the Modify Calibration Selections dialog box.

The following selections are available ONLY if using an ECal module.

**Do orientation**  When this box is checked (default) the PNA senses the ECal model and direction in which the ECal module port is connected to the PNA ports. If power to the ECal module is too low, it will appear as if there is no ECal module connected. If you use low power and are having this problem, clear this check box.
to provide the orientation manually.

Orientation occurs first at the middle of the frequency range that you are calibrating. If a signal is not detected, it tries again at the lowest frequency in the range. If you have an **E8361A** or **E836xB** PNA and do an ECal completely within 10 - 20 MHz OR 60 - 67 GHz, you may need to do orientation manually. There may not be sufficient power to orient the ECal module at those frequencies.

**View/Detect ECal Characterizations**  Appears only if an ECal module is selected for use. Click to invoke the [View ECal Modules and Characterizations](#) dialog box. Displays a list of ECal modules that are connected to the PNA.

### Power Cal Settings dialog box help

![Power Cal Settings dialog box](image)

**Note:** A **Use Power Table** checkbox (not shown) is available when a mmWave SMC measurement is active. [Learn more](#).

**Power Cal at:** Select the source port for which a Power Calibration will be performed. The source and receiver correction will be transferred to all other sources and receivers involved in the S-parameter measurements.

**Use Multiple Sensors**  (NOT available with mmWave SMC measurements.) Check this box when you want to use more than ONE power sensor to cover the measurement frequency range. The dialog is replaced with the **Multiple Sensors** dialog (see following image). When "Use Multiple Sensors" is cleared (default setting), connect only ONE sensor to the PNA.

**Power Meter Settings**  Click to start the standard [Power Meter Settings dialog](#).

**De-embed (power sensor) adapter**  When the power sensor connector is NOT the same type and gender as the DUT connector for the specified port, then for optimum accuracy, extra cal steps are required to measure and correct for the adapter that is used to connect the power sensor to the reference plane.

- **Clear**  this box to NOT compensate for the added adapter.
- **Check**  this box to perform extra calibration steps to measure and correct for the adapter.

Then select the **Power Sensor Connector** type and gender of the power sensor. "Ignored" does NOT compensate for the added adapter, just as if the checkbox were cleared.

When this connector matches the DUT connector for the same port, then the PNA assumes that there is no adapter. Extra cal steps are NOT required and the Cal Kit selection is not available.

Otherwise, select the **Cal Kit** to be used to calibrate at the adapter.

See [Accuracy Settings](#) below.
Specify how the ECal module is connected dialog box help

This dialog box appears when the Do orientation checkbox in the previous Modify Frequency dialog box is cleared.

Click the ECal Port that is connected to each PNA port.

SMC Calibration Steps dialog box help

Power Level at which to perform the Power Cal.

It is usually best to set power level to 0 dBm at the power sensor because the power sensor is calibrated at that level. Lower power levels will yield a slower and noisier calibration.

If an external component is used between the PNA-X test port and the calibration reference plane, then adjust the power level so that the power at the sensor is about 0 dBm if possible.

The current source attenuation value is shown on the dialog.

LO Power Cal (Optional) When enabled, perform a Source Power Cal at the DUT LO connector. An LO must already be selected. Learn how. The power level of the LO source calibration is set on the (LO) Power Tab.
**Calibration Completed** dialog box help

**Finish**  
Save to the channel's calibration register.

**Save As User Cal Set**  
Starts the **Save as User Cal Set** dialog box AND save to the channel's calibration register.

**Cancel**  
Calibration is NOT applied or saved.

Learn about **Calibration Registers**.

Learn about **User Cal Sets**.

---

**Specify delay** dialog box help

This dialog appears ONLY when **Adapter Removal** or **Unknown Thru** calibrations are performed.

The following values were estimated from the measurement. Most of the time, they are adequate. However, for CW sweep or frequency sweep with large step sizes, the accuracy of the values may be improved.

**Adapter delay**  
To improve this value, measure and record the delay of the adapter with a dense step size. Enter that value here. The required precision value is the accuracy that is required to characterize the delay value.

**Nominal phase offset**  
(Waveguide ONLY). To improve this value, measure and record the phase offset of the Waveguide adapter with dense step size. Enter that value here.

When one connector is coax and the other connector is waveguide, the phase offset has an ambiguity of 180 degrees. For consistency, the estimate provided here is always between 0 and 180 degrees. You can change this estimate to any value between -180 degrees and +180 degrees.

**For SMC calibrations**, this dialog box appears twice: once for the input frequencies and once for the output frequencies. The values can be slightly different.
8-Jun-2011  Clarify 'stable phase measurements'
1-Jun-2011  Add *.S2Px
25-Oct-2010  Combined SMC info to one topic and added fixturing features
26-Aug-2010  Add link to Cal 2-stage SMC+Phase
25-Mar-2010  Add SMC with phase, Data Save As, GAV
5-Feb-2010  Add Speed SMC
3-Sep-2008  Removed legacy content
1-May-2008  Updated for selectable ports
5-Oct-2007  Added link to embedded LO
VMC Measurements

VMC Setup and Calibration is very similar to SMC. See FCA Overview to learn about the features that are common to these two applications.

The following information is unique to VMC:

- VMC Hardware Setup
- Create a VMC Measurement
- VMC Parameters Offered
- The VMC Mixer Setup dialog
- VMC Calibrations

See Also

- Embedded LO
- How to make a VMC Fixed Out measurement

VMC Hardware Setup
DUT Input (RF) must be connected to PNA port 1.
DUT Output (IF) can be connected to any other PNA port.

Notes:

- When using a PNA-X with Internal Second Source, the external source is NOT necessary.
- See note regarding LO power out both second source ports
- Learn which PNA ports can be used for the LO.
- Measure a DUT with an Embedded LO

Reference Mixer

The Reference mixer provides a phase reference for the measurements. The reference mixer is connected in the reference receiver path of the network analyzer, between the source out and receiver R1 in ports, as shown below.

The reference mixer is considered part of the test system setup like the test cables. It remains in place during the entire calibration and measurement process. The reference mixer is switched in and out of the measurement path by the PNA as needed. See how to manually switch the reference mixer.
The reference mixer does not need to be reciprocal and does not have to match the calibration mixer or the mixer-under-test in performance. The only requirement of the reference mixer is that it cover the same frequency range as the mixer under-test. In general, it is valuable to select a reference mixer that can be used with a variety of different setups. For example, a broadband mixer can be used in place of several narrow-band alternatives.

A low pass filter on the output of the reference mixer can be used to suppress the LO leakage signal that comes out of the reference mixer output. It is not strictly needed, but ensures that the PNA will not have any source unlock or unlevel errors due to the LO leakage.

- Connect the Reference Mixer INPUT to PNA Ref 1 Source out
- Connect the Reference Mixer OUTPUT to PNA Rcvr R1 In

**Calibration Mixer/Filter**

The Calibration mixer/filter is characterized either before or during a VMC calibration. It is used during the VMC calibration as the THRU standard. The calibration mixer/filter combination must meet the following requirements:

- The mixer must be reciprocal over the frequency range of the mixer under test. This means that it has the same magnitude and phase response in the up-converting and down-converting directions (C21 = C12) as shown in the following diagram.

![Diagram](image)

- If the Input and Output frequency ranges are overlapping, the mixer must have Input to Output Isolation greater than 10 dB more than the conversion loss in the overlapping range.

- The filter must reject the undesired mixing product, and pass the desired mixing product, at the output of the cal mixer. This requirement can be made easier by characterizing the mixer/filter as a downconverter. Learn more.

**Note:** With a corrected VC21Swept LO measurement, the phase data is displayed relative to the phase of the calibration mixer that was used during the VMC calibration. In addition, Group delay display format is NOT valid.

See an example of a Fixed Output VMC Measurement

**Important note: Orientation of Reference mixer and Calibration mixer/filter**

The reference mixer is ALWAYS connected in the same orientation as the DUT, since the output frequency of the
reference mixer has to match that of the DUT. The same applies to the calibration mixer/filter if it is characterized as part of a full VMC cal.

If you characterize the calibration mixer/filter separately, you can characterize it as either an upconverter or downconverter. Learn more.

**LO Source**

**Note:** When using a PNA-X with Internal Second Source, the external source is NOT necessary.

- See note regarding LO power out both second source ports
- Learn which PNA ports can be used for the LO.
- Connect External Sources to the PNA GPIB Controller port.
- See the rear panels of the PNA-X or PNA 'C' models.
- Learn how to Configure an External LO Source

**Create a VMC Measurement**

1. On the PNA-X front panel, press Meas then **[Measurement Class]**

2. Select **VMC**, then either:
   - **OK** delete the existing measurement, or
   - **New Channel** to create the measurement in a new channel.

3. A VC21 measurement is displayed.

4. To select additional parameters to display, click **Trace**, then **New Trace**, then select a parameter from the list.

**VMC Parameters Offered**

**Important Note:** Connecting your DUT to the PNA:

RF and IF terminology is NOT used in FCA because the PNA does not know how the DUT is labeled or how it will be used. Instead, the general terms INPUT and OUTPUT are used.

- **INPUT** - The DUT port being stimulated with frequencies before conversion.
- **OUTPUT** - The DUT port outputting converted frequencies.

INPUT and OUTPUT Frequencies are specified using the Mixer Setup dialog box.

The DUT input is always connected to PNA port 1. However, the DUT output can be connected to any other PNA
port.

- VC21, VC31, or VC41 Conversion Loss/Gain (default) - stimulus at Input, response at Output
- S11 - stimulus and response at Input
- S22, S33, or S44 - stimulus and response at Output
- R1 (or R) - stimulus at Input, measures absolute power at the R1 receiver (uncorrected)
- B, C, or D - stimulus at Input, measures absolute power at the output receiver (uncorrected)
- Reverse conversion loss is NOT offered because of the reference mixer.

See Also

Measure a DUT with an Embedded LO

VMC Mixer Setup

How to start the VMC Mixer Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

Using front-panel HARDKEY [softkey] buttons

1. Press Freq
2. then [Mixer Setup]

Using a mouse with PNA Menus

1. Click Stimulus
2. then Freq
3. then Mixer Setup

The following VMC Mixer Setup dialog tabs are presented:

- **Sweep Tab** (shared with SMC)
- **Power Tab** (shared with SMC)
- **Mixer Freq Tab** (shared with all converter apps)
- **Mixer Power Tab** (shared with all converter apps)
- **Mixer Setup Tab** (shared with all converter apps)

VMC Calibration Overview
The Calibration Wizard guides you through this process. The first three steps characterize the calibration mixer that is used as the THRU standard during the calibration process.

1. Perform a **2-port SOLT calibration** over the INPUT frequency range of the DUT, and another **2-port SOLT calibration** over the OUTPUT frequency range. Use either a mechanical calibration kit or an ECal module.

2. Characterize the input and output match of the **calibration mixer/filter combination** with the external LO connected and the output terminated with an open, short, and load. **Learn how to connect the calibration mixer/filter.** Once characterized, an S2P file is saved and can be recalled for use in subsequent VMC calibrations using the same stimulus settings. **Note:** Use an *.S2PX file for SEGMENTED VMC measurements. [Learn more](#).

3. Connect the **reference mixer** between the Source Out and Rcvr R1 front-panel connectors. Connect the output port of the calibration mixer/filter combination to PNA Port 2 (or at the end of the cable attached to the port).

4. Measure the calibration mixer/filter combination as the THRU calibration standard.

5. The PNA calculates the error terms necessary to make corrected phase measurements of your mixer/ converter under test.

**VMC Cal Wizard**

The following dialog boxes are presented during VMC Calibration and VMC [Mixer Characterization](#).

- **Calibration Setup**
  - Waveguide/In-fixture/On-Wafer Setup
- **Calibration Mixer Characterization**
- **Measurement Direction**
- **Select DUT Connectors and Cal Kits**
  - Modify Frequency Cal
  - Specify how the ECal module is connected
  - Modify Mixer Cal
  - Select the ECal Port to be connected to the Output of the Calibration Mixer
- **Vector Mixer Cal Steps**
- **Measure Calibration Standards**
  - Save Mixer Characterization
- **Calibration Completed**
How to Perform a VMC Calibration

1. Create an FCA measurement, then...

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press CAL</td>
<td>1. Click Response</td>
</tr>
<tr>
<td>2. then [Start Cal]</td>
<td>2. then Cal Wizard</td>
</tr>
<tr>
<td>3. then [Cal Wizard]</td>
<td>3. then Cal Wizard</td>
</tr>
</tbody>
</table>

To perform Mixer Characterization ONLY

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Not Available</td>
<td>1. Click response</td>
</tr>
<tr>
<td>2. then Cal</td>
<td>2. then Cal</td>
</tr>
<tr>
<td>3. then Start Cal</td>
<td>3. then Mixer Characterization Wizard</td>
</tr>
<tr>
<td>4. then Mixer Characterization Wizard</td>
<td></td>
</tr>
</tbody>
</table>

Calibration Setup dialog box help

Waveguide/In-fixture/On-Wafer Setup Check to embed or de-embed circuit networks on the input and output of your mixer under test. Starts the following dialog box.

Additional Power Cal Steps - Used ONLY for SMC Cals
Waveguide/In-fixture/On-Wafer Setup dialog box help

This dialog box appears ONLY if you checked the Waveguide/In-fixture/On-Wafer Setup box in the previous Cal Setup dialog.

Allows you to embed or de-embed circuit networks on the input and output of your mixer under test.

For Network1 (Input) and Network2 (Output) select Embed, De-embed, or None.

Browse  Click to navigate to the .S2P file that models the network to embed or de-embed.

Reverse port positions for input/output  Check to cause the Fixture/Adapter to be configured with Port 2 connected to the PNA and Port 1 to be connected to the DUT. The image in the dialog is updated to reflect that change.

Enable Extrapolation  Check (default setting) to apply a simple extrapolation when the S2P file has a narrower frequency range than the channel. The values for the first and last data points are extended in either direction to cover the frequency range of the measurement. A warning message is also displayed when extrapolation is necessary.

To Embed or De-embed

- When you have a 2 port network that needs to be connected between the Cal reference plane and the DUT during the measurement, but it is NOT present during the calibration, then that network has to be De-Embedded from the port in question during the calibration. In other words, De-Embedding in FCA calibration extends the calibration reference plane to include the two port network.

- When you have a 2 port network that is included as part of the calibration reference plane but has to be disconnected during the measurement, then that 2-port network has to be Embedded for the port in question during the calibration. In other words, Embedding in FCA calibration retracts the calibration reference plane to exclude the two port network during the measurement.

Notes

- Characterize Adaptor Macro can be used to create the S2P file.

- Interpolation is performed when more frequencies are included in the file than in the channel, and the data points do not exactly match those of the measurement.
What is Calibration Mixer Characterization? For a brief explanation, see Calibration Mixer.

Select Mixer Characterization Method

**Perform Characterization (requires a reference mixer)** Performs a Mixer characterization in addition to the VMC calibration. The mixer characterization file will be saved at the end for use in subsequent VMC calibrations. Choose this selection if you do NOT already have a mixer characterization file to load.

**Load characterization from file** Loads an S2P calibration mixer characterization file. Click **Browse** to locate the file.

**Note:** Load an *.S2PX file for SEGMENTED VMC measurements. [Learn more.]

- The frequency range of the S2P file MUST be the same, or larger than, the frequency range of the FCA measurement. If the S2P file frequency range is larger, or the data points do not exactly match those of the measurement, interpolation will be performed.

- The VMC calibration requires that the calibration mixer be connected in the same orientation as that in which it was characterized. The direction in which it was characterized is not part of the file that is recalled. You have to remember and connect it appropriately.

"Invalid Mixer Characterization File" is displayed if the frequency range of the S2P file is smaller than those of the measurement.

**Note:** A Mixer Characterization Cal can be performed separately. [Learn how.]
**Measurement Direction dialog box help**

This dialog box appears ONLY if your settings in the Mixer Setup dialog box indicate that your DUT is being tested as an upconverter (input < output). It allows you to characterize the Calibration Mixer / Filter as a downconverter (input > output) or an upconverter.

The following example shows why you would choose to characterize the calibration mixer as a downconverter. Consider a DUT being used as an upconverter. The input frequency is 70 MHz, the LO is 20 GHz, and the selected (+) output frequency is 20.07 GHz. If we chose (-) in the mixer setup dialog, the output frequency would be 19.93 GHz.

- **Characterize as upconverter**  A very sharp cutoff filter is required to reject the undesired output of 19.93 GHz and pass the desired 20.07 GHz.

- **Characterize as downconverter**  The input frequency is 20.07 GHz; the LO is 20 GHz. The sum (+) output is 40.07 GHz and the diff (-) output is 70 MHz. These are very easy to separate with a low-pass filter. The original frequencies are always used in the downconversion process, so be sure to choose a filter that will pass 70 MHz and reject 40.07 GHz.

See connection diagrams.

**Select DUT Connectors and Cal Kits dialog box help**

Allows you to specify the connector type and Cal Kit for each DUT port.

**Port n**  For each listed PNA port, specify the DUT connector type and gender, and the Cal Kit to use.

**Mixer Out Port**  Output port of the image filter that is connected to the calibration mixer / filter combination. Specify the Cal Kit / standards to use for the measurement of the calibration mixer / filter combination.

**Note:**  When selecting a cal kit for the Mixer Out Port, be sure that the kit has standards with connectors that can mate to the mixer output port. If you choose an ECal, the ECal must have at least one port that can mate to the mixer output port.

**Note:**  If your DUT connectors are:

- **Waveguide**  Change the system impedance to 1 ohm before performing a calibration. See Setting System Impedance.

- **Not listed** (male and female)  Select Type A as the connector type. Type A requires a calibration kit file containing the electrical properties of the standards used for calibration (see Calibration kits).
- **Unspecified** (like a packaged device) Select **Type B** as the connector type. Type B requires a calibration kit file containing the electrical properties of the standards used for calibration (see Calibration kits).

**Modify Cal** Check to start the Modify Cal dialog. If performing a Mixer Characterization Cal at the same time as VMC Cal, two Modify Cal dialogs will be presented, one after the other.

---

**Modify Frequency Cal** dialog box help

For VMC calibrations - NOT for Mixer Characterization.

**Thru Cal Method** For each Thru connection, choose the Thru method. Learn more about these choices.

**Cal Type/Stds** Click to start the Modify Calibration Selections dialog box.

The following selections are available ONLY if using an ECal module.

- **Do orientation** When this box is checked (default) the PNA senses the ECal model and direction in which the ECal module port is connected to the PNA ports. If power to the ECal module is too low, it will appear as if there is no ECal module connected. If you use low power and are having this problem, clear this check box to provide the orientation manually.

  Orientation occurs first at the middle of the frequency range that you are calibrating. If a signal is not detected, it tries again at the lowest frequency in the range. If you have an **E8361A** or **E836xB** PNA and do an ECal completely within 10 - 20 MHz OR 60 - 67 GHz, you may need to do orientation manually. There may not be sufficient power to orient the ECal module at those frequencies.

- **View/Detect ECal Characterizations** Appears only if an ECal module is selected for use. Click to invoke the View ECal Modules and Characterizations dialog box. Displays a list of ECal modules that are connected to the PNA.
Specify how the ECal module is connected dialog box help

This dialog box appears when the Do orientation checkbox in the previous Modify Frequency dialog box is cleared.
Click the ECal Port that is connected to each PNA port.

Modify Mixer Cal dialog box help

Mixer Characterization ONLY. The Thru standard is not measured. Therefore, the Thru Cal Method choices are not available.

View / Detect ECal Characterizations Available ONLY if using an ECal module. Invokes the Select ECal Module and Characterization dialog box.

Select the ECal Port to be connected to the Output of the Calibration Mixer dialog box help

Select the ECal Port to be connected to the output of the image filter of the Calibration Mixer / Filter combination. See connection diagram of Calibration Mixer / Filter combination.
**Measure Calibration Standards** dialog box help

Prompts for standards to be measured. Connect the standard, then click **Measure**.

**Measure**  Measures the mechanical standard and continue to the next calibration step.

**[ReMeasure]**  Replaces Measure after standard has been measured. Allows you to remeasure a standard.

**Done**  Click to proceed to the **Calibration Complete** dialog. Available only after all measurements for the calibration are complete.

**Back**  Returns to the previous dialog box.

**Next**  Does NOT make a measurement. Proceeds to the next required step.

**Cancel**  Exits the Calibration Wizard.

---

**Vector Mixer Cal Steps** dialog box help

Connect the Open, Short, and Load standards to the image filter output, then click **Measure**.

This portion of the calibration characterizes the calibration mixer.

The connection is different depending on if the calibration mixer is an upconverter being characterized as a down converter.

**Note:**

The following are simplified connection diagrams - the reference mixer and LO signals must also be connected. These images assume that the DUT output is connected to PNA port 2.

As a **Downconverter**, (The PNA automatically switches to make the S22 measurement on the device.)

As an **Upconverter**
Done  Click to proceed to the Calibration Complete dialog. Available only after all measurements for the calibration are complete.

Save Mixer Characterization dialog box help

Allows you to save the characterization data of your calibration mixer. When performing another VMC calibration using the same calibration mixer, this S2P file can then be recalled.

Browse  Navigate to the location where you want to save the characterization data of your calibration mixer. Either use the default file name or enter a custom file name.

Next  Saves the mixer characterization file and continues with the next step in the full system calibration routine.

Finish  Replaces Next if you are only characterizing the calibration mixer instead of performing a full system calibration. Saves the mixer characterization file and exits the mixer characterization routine.
**Calibration Completed** dialog box help

**Finish**  
Save to the channel's calibration register.

**Save As User Cal Set**  
Starts the [Save as User Cal Set dialog box](#) AND save to the channel's calibration register.

**Cancel**  
Calibration is NOT applied or saved.

Learn about [Calibration Registers](#).

Learn about [User Cal Sets](#).

---

**Specify delay** dialog box help

This dialog appears ONLY when [Adapter Removal](#) or [Unknown Thru](#) calibrations are performed.

The following values were estimated from the measurement. Most of the time, they are adequate. However, for CW sweep or frequency sweep with large step sizes, the accuracy of the values may be improved.

**Adapter delay**  
To improve this value, measure and record the delay of the adapter with a dense step size. Enter that value here. The required precision value is the accuracy that is required to characterize the delay value.

**Nominal phase offset** (Waveguide ONLY). To improve this value, measure and record the phase offset of the Waveguide adapter with dense step size. Enter that value here.

When one connector is coax and the other connector is waveguide, the phase offset has an ambiguity of 180 degrees. For consistency, the estimate provided here is always between 0 and 180 degrees. You can change this estimate to any value between -180 degrees and +180 degrees.

**For FCA calibrations**, this dialog box appears twice: once for the input frequencies and once for the output frequencies. The values can be slightly different.

---

Last Modified:

21-Oct-2010    MX New topic
How to make a VMC Fixed Output Measurement

The following is a step-by-step example illustrating how to measure a mixer in swept LO mode using FCA Vector Mixer Calibration.

There are fewer components required for SMC as compared to VMC, and fewer measurement steps. You can now make relative phase measurements with SMC. Also, ONLY SMC (not VMC) can measure the reverse conversion loss of the mixer.

This procedure can also be used for making fixed LO measurements, which is quite similar. Although the LO source is still required, the physical triggering cables that connect the PNA and External Source are not required.

Required Equipment

- PNA-X or PNA 'C' models
  - with option 083 (FCA)
  - with PNA Rev A.09.33 or above

**Note:** This example has been updated to show keystrokes for the FCA User Interface available with PNA Rev. A.09.33 and above.

- GPIB External Source (Agilent ESG or PSG works best). This source is NOT necessary when using PNA-X with Internal Second source.
- Reference Mixer (see requirements)
- Calibration Mixer/Filter (see requirements)
- Power splitter - Not necessary when using PNA-X with Internal Second source.
- ECal module with connectors that match the Input and Output connectors of the DUT. You can use adapters to make the ECal module match the DUT connectors, but first perform an ECal user-characterization with the adapters attached. ECal makes the FCA calibration much easier.

Cables and adapters

- **Optional** GPIB Power meter and sensor (for LO power calibration)

The example mixer

The example device is a mixer with the following characteristics:

- LO and Input Frequency Range: 2 GHz to 4.2 GHz
- Output Frequency Range: DC to 1.3 GHz

We will measure:

- Fwd Conversion Loss (VC21)
- Input match (S11)
- Output match (S22)
- Rev Conversion Loss is NOT possible because of the reference mixer.

**VMC Setup**

Connect the devices as shown in the following diagram:

**Note:** This setup can also be used for SMC measurements, allowing you to make VMC and SMC measurements simultaneously on separate channels. The Reference Mixer is automatically switched during SMC measurements. The Cal Mixer/Filter is not used.

- When using a PNA-X with an Internal Second Source, the external source is NOT necessary.
  - See note regarding LO power out both second source ports
  - Learn which PNA ports can be used for the LO.
- The low-pass filter on the output of the Reference Mixer is recommended, but NOT required. [Learn more.]
- When using 4-port models, the mixer input must be connected to PNA port 1. The mixer output can be connected to any other PNA port.

**Make Connections on the Instrument rear panels:**

1. Connect the Source to the PNA GPIB Controller port using a GPIB cable.
2. Using two BNC cables, connect the Source and PNA Trigger connectors as shown in the following image. This is not necessary when making fixed LO measurements, or using a PNA-X with Internal Second Source.
3. Using a BNC cable, connect the PNA **10 MHz Reference Output** to the Ext. Source **10 MHz Reference Input**.

![Rear Panel Connectors Diagram]

**Create the Measurement**

**For this document:**

- Front-panel hardkeys are formatted as "Press TRACE"
- Front-panel softkeys are formatted as "Press [S11]
- Menus are formatted as "Click System"

1. Connect the DUT.
2. Press **P RESET** to make sure you are starting with a known state.
3. Press **Meas** then **[Measurement Class]**, then **Vector Mixer/Converter** then **OK**. At the **Confirm...** dialog, click **OK**. An S11 trace is created.
4. Press **[VC21]** to replace the S11 trace.
Configure the Mixer settings

1. Press **Freq** then [Mixer Setup] to start the Mixer Setup dialog.
2. On the **Sweep tab**, no changes from the default settings are required. The Avoid Spurs feature is useful for eliminating spurs in test setups with excessive LO leakage.
3. On the **Power tab**, change the DUT Input Port Power Level to -17 dBm.
4. On the **Mixer Frequency tab**, enter the frequency values as shown in the following image:

   a. You can enter the Input and the Output frequencies, then click **Calc LO**.
   b. If **Input > LO** is NOT checked, the PNA assumes you want the Input < LO frequencies, and higher LO frequencies are calculated as a result.

External Source Configuration

When using a PNA-X with Internal Second Source, the external source is NOT necessary.

- See note regarding LO power out both source ports
- Learn which PNA ports can be used for the LO.

1. On the **Mixer Setup tab**, click **Add Source** to start the External Source Configuration dialog. Learn how to configure an external source.
2. On the External Source Configuration dialog, click **Device Properties**.
   a. Click **Hardware List (BNC)**, which is the fastest measurement method. This method requires the BNC Trigger cables that connect the PNA and source. If not available, **Software CW** can be used, but measurements are much slower.
   b. If necessary, select the Interface (usually **GPIB**), then **Refresh**, then select the **Available IO Configuration**.
   c. Click **OK** to close the Device Properties dialog, then **OK** to close the External Source Configuration dialog.
3. Next to LO1, select the configured external source name.

Complete Mixer Setup
1. On the **Mixer Power tab**, change LO1 Power to 0.00 dBm. This setting specifies the power out of the external source (not at the DUT) unless an LO power cal is performed.

2. When the settings are valid, the **Save**, **Apply** and **OK** buttons are available to click. To save the mixer settings in a file so you can recall them easily, click **Save...**, then type a descriptive filename. Then click **OK**.

3. Click **OK** to close the VMC Setup dialog.

4. To change the X-axis values from the default Output selection (800 MHz), click **Response**, then **Measure**, then **Input**.

5. The trace is updated as the external source steps in frequency. It should look something like the following image. Because of the reference mixer, the uncorrected VMC measurement can look like it has gain.

![Image of trace](image.png)

### Problems?

**Not sweeping:**

- On the PNA, press **TRIGGER**, then **Continuous** to start the PNA sweeping. Watch for error messages on the PNA and source.

**Problems communicating with the source:**

- Press **SYSTEM** then **Configure** then **External Device**. On the External Device Configuration dialog, select the external source, then click Device Properties. Next to Trigger Mode: select **Software CW trigger**, then close the dialog. Again, press **TRIGGER**, then **Continuous** to start the PNA sweeping. If this works, then something is wrong with **Hardware (BNC)**. Check the trigger cables on the rear panel.

- As a last resort, try rebooting the PNA. First, save the entire setup to a `.csa` file. When the PNA preset measurement appears, recall the `.csa` file to resume at this step.

If the source is sweeping, and the PNA Input is sweeping, but there is still no output.
- Check power levels at the LO and Input.
- Check the DUT by making a fixed LO measurement which is much easier.

**Perform a VMC Calibration**

1. Disconnect the DUT.
2. Connect the ECal module to a PNA USB port.
3. Press **Cal**, then **[Start Cal]**, then **[Cal Wizard]**. Because the VC21 measurement is active, the Cal Wizard automatically begins a VMC Calibration.
4. At the **Calibration Setup** dialog, click **Next**. Or check **Enable LO1 Power Cal** to perform a **Source Power Cal** to specify the LO Power at the DUT. This requires a power meter or USB power sensor be connected.
5. At the **Calibration Mixer Characterization** dialog, click **Next**. We will perform characterization of the Calibration mixer as part of the VMC cal. Later we will save the Calibration mixer characterization so that, in future VMC calibrations that use this same frequency range, we can recall the Calibration mixer characterization by clicking **Load Characterization from file**.
6. At the **Select DUT Connectors and Cal Kits** dialog, for **Port 1** select the connector type and gender of your DUT INPUT. For **Port 2** select the connector type and gender of your DUT OUTPUT. Then select ECal as the Cal Kit to use for each connector. Click **Next**.
7. At the **Select the ECal Port to be Connected** dialog, ensure that **Port A** is selected for **Port 1**, then click **Next**.
8. At the **Vector Mixer Calibration Step 1 of 3** dialog, connect the ECal module Port A to the Port 1 cable, and Port B to the Port 2 cable. Then click **Measure**. This portion of the calibration gathers the linear (non-frequency-translating) error terms of the test setup at the input and output frequencies.
9. At the **Vector Mixer Calibration Step 2 of 3** dialog, connect the following, then click **Measure**. This portion of the calibration will connect reflection standards to characterize the S-parameters of the calibration mixer/filter.
   - Port 1 cable to the Input of the calibration mixer.
   - LO cable to the LO port of the calibration mixer.
   - ECal module to the Output of the calibration mixer/filter.
10. At the **Vector Mixer Calibration Step 3 of 3** dialog, disconnect the ECal module and connect the Port 2 cable to the output of the calibration mixer/filter, then click **Measure**. This step completes the calibration using the characterized mixer/filter as a Thru standard.
11. At the **Save Mixer Characterization** dialog, click **Browse**, then type a unique filename and click **OK**. Then click **Next**. This saves the Calibration Mixer characterization to an S2P file. This file can be recalled for subsequent VMC calibrations.
12. At the **Calibration completed** dialog, you can choose to save the VMC calibration as a User Cal Set. Otherwise, click **Finish** to complete the VMC calibration. Correction is turned ON and applied to the VMC trace that we set up earlier.

**What is happening?**

When an external source is sweeping, the measurements are much slower. When correction is ON, you will see that there are times when nothing is happening on the screen. This is because there are background measurements being made but not displayed.

This is exactly the same as when full 2-port correction is applied to an S-parameter. All four parameters are measured, then correction is applied, then all four measurements are updated. This occurs much faster when there is no external source. With a VMC measurement, there is no VC12 (reverse transmission measurement), so there are only three background measurements. With correction OFF, the traces are updated as the data is measured. You can see this taking place by creating the following measurements.

**Create S11 Input and S22 Output Match**

1. Press **M**EAS then [S11] and [S22] to add these measurements to the same channel.

2. While the source is sweeping, watch the source port indicator on the front of the PNA. First, the port 1 indicator will light for two sweeps, then the port 2 indicator will light for 1 sweep while all 3 traces update.

3. Press **CAL**, then [Correction OFF]. Notice that the relevant traces will update as the sweep is occurring.

The following image shows the corrected Conversion Loss (VC21), Input Match (S11), Output Match (S22) and the uncorrected Conversion Loss (VC21), which is a memory trace.
<table>
<thead>
<tr>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4-Apr-2011</td>
<td>Updated for FCA2 (A.09.33)</td>
</tr>
<tr>
<td>16-Apr-2009</td>
<td>Updated for PNA-X</td>
</tr>
<tr>
<td>6-Apr-2009</td>
<td>Replaced N5242a with PNA-X</td>
</tr>
<tr>
<td>5-May-2008</td>
<td>Added selectable output note.</td>
</tr>
<tr>
<td>17-Apr-2008</td>
<td>Added 10 MHz Ref image</td>
</tr>
</tbody>
</table>
How to make an SMC Fixed Output Measurement

The following is a step-by-step example illustrating how to measure a 1-stage mixer in swept LO mode using FCA Scalar Mixer Calibration.

There are fewer components required for SMC as compared to VMC, and fewer measurement steps. You can now make relative phase measurements with SMC. Also, ONLY SMC (not VMC) can measure the reverse conversion loss of the mixer.

This procedure can also be used for making fixed LO measurements, which is quite similar. Although a second source is still required, when using an external source, the physical triggering cables between the PNA and External Source are not required.

Required Equipment

- PNA-X or PNA 'C' models
  - with option 083 (FCA) or option 082 (SMC)
  - with PNA Rev. A.09.33 or above.

Note: This example has been updated to show keystrokes for the FCA User Interface available with PNA Rev. A.09.33 and above.

- GPIB External Source. Not necessary when using PNA-X with Internal Second source.
- ECal module with connectors that match the Input and Output connectors of the DUT. You can use adapters to make the ECal module match the DUT connectors, but first perform an ECal user-characterization with the adapters attached. ECal makes the FCA calibration much easier.
- GPIB or USB power meter / sensor
- Cables and adapters

Note: This procedure refers to an External Source to control the LO. If using a PNA-X with an Internal second source, an external source is not necessary. Connect the LO directly to the second source output.

The example mixer

The example device is a down-converter mixer with the following characteristics:

- LO and Input Frequency Range: 2 GHz to 4.2 GHz
- Output Frequency Range: DC to 1.3 GHz

We will measure:

- Fwd Conversion Loss (SC21)
- Input Match (S11)
- Output Match (S22)
- Reverse Conversion Loss (SC12)

**SMC Setup**

Connect the devices as shown in the following diagram:

![Diagram of SMC Setup](image)

The DUT can be connected to any PNA ports. [Learn more.](#)

This procedure uses DUT input to PNA port 1 and output to PNA port 2.

This procedure refers to an External Source to control the LO. If using a PNA-X with an [Internal second source](#), an external source is not necessary. Connect the LO directly to the second source output (port 3 or port 4 on a PNA-X).

**Make Connections on the rear panels:**

1. If using a GPIB power meter, connect the power meter to the external source GPIB connector. If using a USB power meter, connect it to any unused USB port.

2. If using a PNA-X with an [Internal second source](#), the following three steps are not necessary. Connect the LO directly to the second source output.
3. Using a GPIB cable, connect the PNA GPIB controller port to the external source GPIB connector.

4. Using two BNC cables, connect the Source and PNA Trigger connectors as shown in the following image. This is not necessary when making fixed LO measurements.

5. Using a BNC cable, connect the PNA **10 MHz Reference Output** to the Ext. Source **10 MHz Reference Input**.

![Rear Panel Connectors](image)

**Create the Measurement**

**For this document:**

- Front-panel hardkeys are formatted as “Press **TRACE**”
- Front-panel softkeys are formatted as "Press [S11]"
- Menus are formatted as "Click **System**"

1. Connect the DUT.

2. On the PNA, press System, then [Configure], then [Power Meter Settings].

3. Under Interface, select **GPIB**, then enter the power meter address. Or select **USB**, then select the USB power meter that is connected to the PNA.

4. Press Preset to make sure you are starting with a known state.

5. Press Meas then [Measurement Class], then **Scalar Mixer/Converter**, then **OK**. At the **Confirm...** dialog, click **OK**. An S11 trace is created.


**Configure the Mixer settings**

1. Press Freq then **Input** to start the Mixer Setup dialog.
2. On the Mixer Frequency tab, enter the Mixer setup values as shown in the image below.

![Mixer Frequency Tab](image)

Notes:

- To provide quicker access to the Mixer Setup dialog, use the Setup softkey. [Learn how.](#)
- Rather then enter ALL of the frequency settings, you can enter the Input and the Output frequencies, then click **Calculate LO**.
- If **Input>LO** is NOT checked, the PNA assumes you want the Input < LO frequencies, and higher LO frequencies are calculated as a result.
- The LO power level setting specifies the power out of the external source; not at the DUT) unless an LO power cal is performed.
- When the settings are valid, the background color around the **Apply** button is available.

**Configure the LO Source**

1. On the Mixer Setup tab:

2. Change LO1 to either an internal PNA source or a pre-configured external source.

3. To configure an external source:
   
a. Click **Add Source** in the upper-right corner of the Mixer Setup dialog.
   
b. Complete the **External Source Configuration** dialog.
   
c. If there is a problem communicating with the source, the PNA will display an error. [See Problems?](#)
   
d. Click **OK** to return to the Mixer Setup dialog.
   
e. The new external source is now available as an LO1 selection.

4. Save the mixer settings in a file so you can recall them easily. Click **Save...**, then type a descriptive filename, such as “FixedOutputMixer”.

5. The trace should begin to sweep as the external source steps in frequency. It should look something like this:
Problems?

Not sweeping:

- On the PNA, press Trigger, then **Continuous** to start the PNA sweeping. Watch for error messages on the PNA and source.

Problems communicating with the source:

- Press System then **Configure** then External Source. On the Select Sources dialog, click Configure. Select your LO source. Click **Software CW trigger**, then close the dialog. Again, press Trigger, then **Continuous** to start the PNA sweeping. If this works, then something is wrong with **Hardware (BNC)**. Check the trigger cables on the rear panel.

- Can the PNA communicate with the power meter? If not, there is something wrong with the GPIB or USB communication.

- As a last resort, try rebooting the PNA. First, **save the entire setup to a .csa file**. When the PNA preset measurement appears, recall this .csa file and continue at this step.

If the source is sweeping, and the PNA Input is sweeping, but there is still no output.

- Check power levels at the LO and Input.

- Check the DUT by making a fixed LO measurement - much easier.

Tip: You can optionally calibrate the LO Power level at the DUT using a standard **Source Power Calibration**. Select the source port in the Source Power Cal dialog.

Perform an SMC calibration

1. Disconnect the DUT.

2. Connect the ECal module to a PNA USB port.
3. Press Cal, then [Start Cal], then [Cal Wizard]. Because the SC21 measurement is active, the Cal Wizard automatically begins an SMC calibration.

4. At the Calibration Setup dialog, click Next.

5. At the Select DUT Connectors and Cal Kits dialog, for DUT Port 1 select the connector type and gender of your DUT INPUT. For DUT Port 2 select the connector type and gender of your DUT OUTPUT. Then select ECal as the Cal Kit to use for each connector. Click Next.

6. At the Scalar Mixer Calibration Step 1 of 2 dialog, connect the power sensor to the Port 1 test cable, then click Measure. The data will be used to correct for input mismatch errors.

7. At the Scalar Mixer Calibration Step 2 of 2 dialog, connect the ECal module Port A to the Port 1 cable, and Port B to the Port 2 cable. Then click Measure. This portion of the calibration gathers the linear (non-frequency-translating) error terms of the test setup at the input and output frequencies.

8. At the Calibration completed dialog, you can choose to save the SMC calibration as a User Cal Set. Otherwise, click Finish to complete the SMC calibration. Correction is turned ON and applied to the SMC trace.

What is happening?
When an external source is sweeping, the measurements are much slower. When correction is ON, you will see that there are times when nothing is happening on the screen. This is because there are background measurements being made but not displayed.

This is exactly the same as when full 2-port correction is applied to an S-parameter. All four parameters are measured, then correction is applied, then all four measurements are updated. This occurs much faster when there is no external source. With correction OFF, the traces are updated as the data is measured. You can see this taking place by creating the following measurements.

Create S12 Upconverter, S11 Input and S22 Output Match

1. Press Meas then [S11], [SC12], and [S22] to add these measurements to the same channel.

2. While the source is sweeping, watch the source port indicator on the front of the PNA. First, the port 1 indicator will light for two sweeps, then the port 2 indicator will light for 2 sweeps. During the last sweep, all 4 traces update.

3. Press Cal, then [Correction OFF]. Notice that the relevant traces update as the sweep is occurring.

With the SC12 measurement you can see the reciprocity of the mixer.

Note: With the recent improvements to FCA, this step is MUCH easier than before. SMC forward and reverse measurements can now reside in the same channel and are calibrated automatically at the same time.
Last Modified:

- 4-Apr-2011  Updated for A.09.33
- 14-Apr-2010  Added N5230A/C and Opt 082
- 6-Apr-2009  Updated for PNA-X
- 5-May-2008  Modified for selectable ports
- 17-Apr-2008  Added 10 MHz ref image
Embedded LO Measurements

The Embedded LO feature allows you to make VMC, SMC, IMDx, IMx Spectrum, GCX, and NFX measurements of mixers that have a FIXED LO inside the DUT.

**Note:** This feature is available as Opt 084, and must be enabled.

Measurements of these devices are challenging for a couple of reasons:

1. The VMC measurement process requires the use of a reference mixer that has the same LO frequency as the DUT. A separate internal or external source must be used for the reference mixer LO. This LO must be controlled by the PNA. A PNA with an internal second source is much faster.

2. All Embedded LO measurements require the PNA receivers to be tuned to the correct frequency to measure the mixer output, which is highly dependent on the exact LO frequency.

How the PNA measures the embedded LO

The nominal frequency of the embedded LO is input into the Mixer Setup dialog. This is used as a starting point for the measurement.

Before each DUT measurement sweep, background sweeps are made to determine the frequency of the embedded LO to a configurable degree of accuracy.

Background sweeps...

- **Broadband** Sweep - rough measurement of the embedded LO frequency, made around a selectable data point over a selectable frequency span. The input signal to the DUT is tuned to a selectable CW frequency. The B receiver is swept across a selectable span around the anticipated output frequency. The difference between the frequency of the found signal and the desired output frequency is then applied as an adjustment.

- **Precise** Sweep The B receiver is measured at the selectable data point. Measurements of phase versus time are made, from which the exact offset frequency is computed, until either the tolerance value or maximum iterations are met.

- For VMC measurements, the reference mixer frequency is updated as the embedded LO frequency is determined.

How to measure a DUT with an Embedded LO

1. Create a VMC, SMC, IMDx, IMx Spectrum, GCX, or NFX measurement.

2. In the mixer setup dialog, enter the nominal frequency of the embedded LO as the LO frequency.

3. Perform a calibration as usual.

4. Launch and complete the Embedded LO Mode dialog box (below)

For VMC (FCA App) ONLY:
The LO source for the Reference Mixer can be either:

- An **Internal** source when using a **PNA-X that has two sources**.
- An **External** source:
  
  - Must be controlled by the PNA. [Learn how.](#)
  - Must be locked to the PNA using the **10 MHz reference**.

**During Calibration** - The LO source is shared between the Reference Mixer and the Calibration Mixer/Filter. This requires a splitter when using an external source, as shown in the following image.

**During the Measurement** - Only the Reference Mixer uses the LO source. Terminate the LO source port that is no longer used by the Calibration Mixer/Filter to ensure that the match seen by the Reference Mixer LO port does not change after the calibration, as shown in the following image. This precaution is not necessary when using the internal second source (ports 3 and 4) of the PNA-X.
For SMC, IMDx, IMxSpectrum, and NFX measurements:
No unique setup is required for embedded LO measurements.

How to Launch the Embedded LO Mode dialog box

Using front-panel HARDKEY [softkey] buttons

1. Not Available

Using a mouse with PNA Menus

1. Click Response
2. then Measure
3. then Embedded LO
The Tuning Settings balance LO measurement speed versus accuracy. You can see that accuracy is becoming compromised when noise starts to appear on the measurement trace.

Scroll up to learn more about the Embedded LO measurement process.

**Embedded LO Mode On** Check to enable measurement of the Embedded LO.

**Tuning Point** Select, or specify, the data point in the mixer sweep that will be used to find the embedded LO frequency. If a marker is enabled, that data point can be used. For broadband and Precise sweeps, choose a point in the mixer sweep where noise is least likely to be found. This is generally the center of a sweep or the center of a filter if used.

**LO Frequency Delta** The absolute difference between the measured embedded LO frequency and the LO setting that is entered in the Mixer Setup dialog. This value is updated each time the embedded LO frequency is measured. Entering a value is a way to change the LO frequency on the mixer setup without invalidating the calibration.

- **Reset** Set the LO Frequency Delta back to 0 Hz.
- **Find Now** The PNA finds and measures the actual LO frequency using the current dialog settings. This data is displayed in the Status box.

**Tuning Settings** These settings determine the amount of time spent versus the degree of accuracy to which the LO Frequency is measured. You can see that accuracy is becoming compromised when noise starts to appear on the measurement trace.

- **Reset** Set all Tuning Settings back to the defaults.
- **Broadband and Precise** Do the entire tuning process for each background sweep.
**Precise only**  Does NOT perform broadband tuning on each sweep. Use this setting when the embedded LO is stable.

**Disable tuning**  Only the previously measured LO Frequency Delta is applied to the reference mixer LO and PNA receivers.

**Sweep Span**  Narrowing the sweep span limits the number of data points that are measured in the broadband sweep and makes the measurement faster.

**Max Iterations**  The maximum number of Precise sweeps to make. When this number is reached, the final measurement is used.

**Tolerance**  When two consecutive Precise measurements are made within this value, the final measurement is used. If this is not achieved within the Max Iterations value, then the last measurement is used. This is the best of the ‘Tunings settings’ to change to improve accuracy.

**Tuning IFBW**  IF Bandwidth used for Broadband and Precise tuning sweeps. The larger the IFBW, the faster the sweep, but the signal may not be found.

**Tune every**  Set the interval at which tuning is performed before a measurement sweep. ‘Tune every 3 sweeps’ means that every third measurement sweep is preceded by tuning sweeps. If the embedded LO drifts, or if regularly changing DUTs, use ‘Tune every 1 sweep’.

**Status**  Allows textual and graphical representation of the Embedded LO measurement sweeps.

**Clear**  Removes the text information currently being displayed.

**Graph**  Launches the following graphical (spectrum analyzer type) display sweeps of the latest embedded LO measurement.
Embedded LO Diagnostic dialog box help

Presents a graphical (spectrum analyzer type) display of the latest embedded LO measurement. Click Previous and Next to view available Broadband and Precise sweeps. The LO Frequency is displayed in the Marker annotation.

Last Modified:

- 12-Aug-2009  Added SMC and IMD measurements
- 3-Sep-2008  Removed legacy content
- 5-Oct-2007  Added config image and text
- 5-Jul-2007  Update access point
- 6-Apr-2007  MX New topic
SMC with a Booster Amp

If your mixer measurement requires more source power on the input than the PNA can provide, a booster amplifier can be used to provide the additional power. This topic describes how to configure and make a calibrated SMC measurement using a booster amplifier.

Connect

Connect the booster amplifier between the Source-Out and Coupler-Thru connectors on the front-panel as shown in the following diagram. Your PNA block diagram may not look like this.

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>SOURCE OUT</td>
<td>h</td>
<td>RCVR B IN</td>
</tr>
<tr>
<td>b</td>
<td>RCVR R1 IN</td>
<td>i</td>
<td>CPLR ARM</td>
</tr>
<tr>
<td>c</td>
<td>SOURCE OUT</td>
<td>j</td>
<td>PORT 2</td>
</tr>
<tr>
<td>d</td>
<td>CPLR THRU</td>
<td>k</td>
<td>CPLR THRU</td>
</tr>
<tr>
<td>e</td>
<td>PORT 1</td>
<td>l</td>
<td>SOURCE OUT</td>
</tr>
<tr>
<td>f</td>
<td>CPLR ARM</td>
<td>m</td>
<td>RCVR R2 IN</td>
</tr>
<tr>
<td>g</td>
<td>RCVR A IN</td>
<td>n</td>
<td>SOURCE OUT</td>
</tr>
</tbody>
</table>

Measurement and Calibration Setup

In the following procedure:
Test Port power is the power level out of the source.

Corrected power is the power level you require at the mixer input and output.

This procedure assumes you will apply stimulus power to the mixer input to make SC11 and SC21 measurements, and to the output of the mixer to make SC22 and SC12 measurements.

1. Determine the gain of the booster amplifier. If the gain has significant slope across the input and output range of the mixer, see Booster Amp with a Gain Slope.

2. Determine the corrected power for both the input (port 1) and output (port 2) of the mixer.

3. Calculate the Test Port power for both ports by subtracting the gain of the amplifier from both the input and output corrected power levels.

For example, the following values assume a 25 dB booster amp on port 1 as in the diagram above.

<table>
<thead>
<tr>
<th>Corrected Power</th>
<th>Amp Gain</th>
<th>Test Port Power</th>
</tr>
</thead>
<tbody>
<tr>
<td>Port 1 (input)</td>
<td>0 dBm</td>
<td>- 25 dB</td>
</tr>
<tr>
<td>Port 2 (output)</td>
<td>-20 dBm</td>
<td>- 25 dB</td>
</tr>
</tbody>
</table>

4. On the PNA Power dialog, clear the Port Power Coupled checkbox, which allows different power levels for each port.

5. Enter the calculated Test Port Power values for each port.

6. During the SMC Cal Wizard Select DUT Connectors and Cal Kits dialog, click View/Modify Source Cal Settings to invoke the Source Calibration Settings dialog.

7. In Power Offset, enter the booster amplifier gain.

Booster Amp with a Gain Slope

SMC calibration takes place over the entire input and output range of the mixer. Therefore, the booster amplifier will also be subjected to the entire input and output frequency range of the mixer.

To compensate for a gain slope, you might have to experiment with the source attenuator setting, power-offset value, and initial power value to get a combination that will not cause the PNA source to go unleveled during or after the cal.

For example, assume the booster amp gain is 30 dB at the low end, and 20 dB at the high end. If you enter 30 dB for the power offset value, the PNA might run out of ALC range when the actual gain drops to 20 dB. The PNA will try to increase its source power to account for the 10 dB gain drop. Therefore, pick a power offset value that is in the middle of the amplifier gain band (25 dB).

If possible, select a PNA attenuator setting that puts the ALC approximately in the middle of its range at the desired corrected power with the mid-band gain. This condition means the ALC can set the power higher and lower to account for the gain slope, without unleveling.
If the gain slope is too large, then there may not be a setting that prevents a source unlevel. In this case, a flatter booster amp must be used.

Last Modified:

19-May-2008    Edited for generic image
Frequency Offset Mode

Frequency Offset Mode (FOM) provides the capability to have the PNA Sources tune to frequencies that are different (offset) from the PNA Receivers.

PNA Option 080 provides you with the hardware and basic software capability to make Frequency Offset Measurements. This topic discusses the PNA settings that are relevant to making these types of measurements. See Frequency Converting Device Measurements for more information on making specific device measurements.

- Frequency Offset Dialog Box
- Setup Examples
- Test Set (Reference Switch) Dialog Box

Other Frequency Offset topics

How to make Frequency Offset settings

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>PNA Menu using a mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press STIMULUS</td>
<td>1. Click Stimulus</td>
</tr>
<tr>
<td>2. then [Frequency Offset]</td>
<td>2. then Frequency</td>
</tr>
<tr>
<td></td>
<td>3. then Frequency Offset</td>
</tr>
</tbody>
</table>

Frequency Offset dialog box help
The following are major changes to FOM:

- Stimulus and Response are now called Sources and Receivers.
- Sources and Receivers settings can be made in two ways:
  1. By **Coupling** to the Primary (Channel) settings. This is the only method used in previous releases.
  2. By **Uncoupling** and setting Sources and Receivers values independently. This is the new, simplified method.

- External sources appear here and can be controlled from this dialog. Learn more.

**Note:** Source2 supplies power for ports 3 and 4. **Turn Source2 power ON** using the Power and Attenuators dialog. This (Frequency Offset) is the only dialog for controlling the frequency of Source 2. Learn more about Source2.

**Frequency Offset (ON/OFF)** Enables Frequency Offset Mode on ALL measurements that are present in the active channel.

When FOM is NOT enabled, all frequencies are the same as the active channel.

**Tip:** First make other settings on this dialog box, then click **Frequency Offset** ON.

**Primary** The current Active Channel settings. When a Source or Receiver is coupled to the Primary settings, its Sweep Type is the same as that of the Primary. The frequency settings of the coupled range are mathematically derived from the Primary settings using the Multiplier, Divisor, and Offset values. With this approach, only the Primary settings need to be changed in order to affect change in the coupled Sources and Receivers. Changes to the Primary channel settings occur when Frequency Offset is checked ON. **See example using Primary and Coupled setting.**

**Tip:** Primary settings are ONLY used when Sources and Receivers are Coupled. It is often easier to Uncouple, then set Sources and Receivers independently.

Source and Source2 if available. Learn more about Internal Second Source.

**Receivers** All receivers that are used in the channel, including Reference receivers, are tuned to the specified frequency settings.

**Mode**

- **Coupled** Source and Receiver settings are mathematically derived from the Primary settings using Multiplier, Divisor, and Offset values. Learn more.
- **Uncoupled** Source and Receiver settings are entered independently, without reference to Primary settings. When Uncoupled, Source and Receiver Ranges can use separate sweep types.

**Sweep Type** Click to change the type of sweep for each range. Only available for Primary and Uncoupled Sources and Receivers.
Unsupported Sweep Type combinations

- Power Sweep and Segment Sweep can NOT be used together.
- Uncoupled Log Sweep yields invalid data whenever the sources are offset from the receivers.
- Coupled Log Sweep is allowed only for the following two conditions:
  1. The offset = 0, the multiplier = 1, and the divisor = 1.
  2. The multiplier = 0

Settings  To change settings, click in the appropriate Settings cell, then click Edit.

- If coupled, invokes the Coupled dialog.
- If uncoupled or Primary invokes the Uncoupled settings dialog.

X-Axis  Select the settings to be displayed on the X-Axis.

X-Axis Point Spacing  Only available when a Segment Sweep Type is selected as the X-Axis display. Learn more.

Note:  When Frequency Offset is enabled, ALL receivers on the channel, including the reference receivers, tune to the new offset frequencies. Therefore the source and reference receiver will be at different frequencies. Therefore, FOM measurements that include a reference receiver, which includes all S-parameters, display invalid data.

To measure and display measurements at both the source and receiver frequencies, you must use two channels. Use Equation Editor to calculate the conversion loss. See a calibrated FOM conversion loss example.

Learn how to calibrate frequency offset measurements.
Coupled settings dialog box help

Coupled Formulas:

Range Start = \[\text{Primary Start} \times (\text{Multiplier} / \text{Divisor})\] + Offset
Range Stop = \[\text{Primary Stop} \times (\text{Multiplier} / \text{Divisor})\] + Offset

Where:

Offset  Specifies an absolute offset frequency in Hz. For mixer measurements, this would be the LO frequency. Range is +/- 1000 GHz. Offsets can be positive or negative.

Multiplier  Specifies (along with the divisor) the value to multiply by the stimulus. Range is +/- 1000.

- Negative multipliers cause the stimulus to sweep in decreasing direction. For downconverter mixer measurements, this would be for setups requiring the Input frequency to be less than LO frequency. See an example.
- 0 (zero) as the multiplier nulls the Primary setting. Then the Offset value adds to zero.

Divisor  Specifies (along with the multiplier) the value to multiply the stimulus. Range is 1 to 1000.

Primary (Linear Frequency)

Primary and Uncoupled settings dialog box help

This dialog will vary depending on the sweep type:

Linear and Log frequency

Uncoupled Log sweep yields invalid data whenever the sources are offset from the receivers.

Select Start/Stop or Center/Span

Frequency  Enter values

Points  (Primary only) Enter number of data points for the sweep.

Power

CW Freq  Enter frequency in Hz.
Points  (Primary only) Enter number of data points for the power sweep.
**CW Time**
- **CW Freq**  Enter frequency in Hz.
- **Sweep Time**  Enter time to complete one sweep. Enter 0 for the fastest sweep.

**Segment Sweep**  Edits are made exactly like the standard segment table.

**For Advanced Users:** Uncoupled Segment Sweep offers great flexibility in configuring measurements. In segment sweep mode:

- The **OK** button is NOT available until the total number of data points for all segments matches the number of Primary data points.
- **Independent IF Bandwidth** and **Independent Sweep Time** are available ONLY on the Primary (channel) and the Uncoupled **Receivers** - NOT Sources.
- **Independent Power** is available ONLY on the Primary (channel) and the Uncoupled **Sources** - NOT Receivers.

**Setup Examples**

Although the Frequency Offset settings can be used with many types of devices, these examples include mixer terminology.

See a Mixer Compression and Phase (AM-PM) Measurement using FOM.

See a calibrated FOM conversion loss example.

1. **Fixed LO - Upconverter**

- **Swept Stimulus (Mixer Input):** 1000 MHz - 1200 MHz
- **Fixed LO:** 1500
- **Swept Response (Mixer Output):** 2500 MHz to 2700 MHz

Make the following settings on the FOM dialog

**Source:** Uncoupled
- **Sweep Type:** Linear
- Click Settings, then Edit. In the Source dialog:
  - Start Frequency = 1000 MHz
  - Stop Frequency = 1200 MHz

**Receiver:** Uncoupled
- **Sweep Type:** Linear
- Click Settings, then Edit. In the Receiver dialog:
  - Start Frequency = 2500 MHz
  - Stop Frequency = 2700 MHz
LO Settings
Set external source to CW - 1500 MHz.

Source2: Uncoupled (Only with Second PNA Internal Source)
- Sweep Type: CW Time
- Click Settings, then Edit. In the Source2 dialog:
  - CW Frequency = 1500 MHz

2. Fixed LO - Downconverter (Input < LO)

- Swept DECREASING Stimulus (Mixer Input): 1100 MHz to 1000 MHz
- Fixed LO: 2500 MHz
- Swept INCREASING Response (Mixer Output) 1400 MHz to 1500 MHz

Make the following settings on the FOM dialog
- **Primary**: Not used
- **Source** (Input): Uncoupled
  - Sweep Type: Linear
  - Click Settings, then Edit. In the Source dialog:
    - Start Frequency = 1100 MHz
    - Stop Frequency = 1000 MHz
- **Receiver** (Output): Uncoupled
  - Sweep Type: Linear
  - Click Settings, then Edit. In the Receiver dialog:
    - Start Frequency = 1400 MHz
    - Stop Frequency = 1500 MHz

LO Settings
Set external source to CW - 2500 MHz.

Source2: Uncoupled (Only with Second PNA Internal Source)
- Sweep Type: CW Time
- Click Settings, then Edit. In the Source2 dialog:
  - CW Frequency = 2500 MHz

3. Swept LO - Fixed Output - Upconverter

Swept External LO measurements in Frequency Offset Mode can be very difficult. The external LO source must be synchronized with the swept output or input (as in this case). See Synchronizing and External Source Control to see how this is done. The Frequency Converter Application Opt 083 performs makes these measurements easily.
- **Swept Stimulus (Mixer Input)**: 1000 MHz to 1100 MHz
- **Swept LO**: 1500 MHz to 1400 MHz
- **Fixed Response (Mixer Output)**: 2500 MHz

Make the following settings on the FOM dialog

**Source**: Uncoupled
- Sweep Type: Linear
- Click Settings, then Edit. In the Source dialog:
  - Start Frequency = 1000 MHz
  - Stop Frequency = 1100 MHz

**Receiver**: Uncoupled
- Sweep Type: CW Time
- Click Settings, then Edit. In the Receiver dialog:
  - CW Frequency = 2500 MHz

**LO Settings**

- If using external source, set to sweep from 1500 - 1400 MHz.
- If using **Source2 (Second Internal Source)**: set to Uncoupled, then:
  - Sweep Type: Linear
  - Click Settings, then Edit. In the Source2 dialog:
  - Start Frequency = 1500 MHz
  - Stop Frequency = 1400 MHz

---

**4. Power Sweep for Mixers**

To measure the gain compression of a mixer, the input power to the mixer is swept. The input and output frequencies are fixed but offset from one another.

This is a good use of Coupled settings because the same compression test can be performed at several different frequencies. With coupled Source and Receiver ranges, the Primary (channel) frequency can be easily changed from the front panel. The coupled source and receiver frequencies will update accordingly.

- **Swept Input Power**: -10 dBm to 0 dBm
- **Fixed Input Frequency**: 1500 MHz
- **Fixed LO**: 500 MHz
- **Fixed Output**: 2000 MHz

Make the following settings on the FOM dialog
Primary:
Sweep Type: Power Sweep
Click Settings, then Edit. In the Primary dialog:
CW Frequency = 1500 MHz

Source: Coupled
Default settings make CW Frequency: 1500 MHz (same as Primary)

Receiver: Coupled
Default settings make Sweep Type: CW Time
Click Settings, then Edit. In the Receiver dialog:
Offset = 500 MHz

LO Settings

- If using external source, set to CW: 500 MHz.
- If using Source2 (Internal Second Source), set to Coupled, then:
  Sweep Type: Power Sweep
  Click Settings, then Edit. In the Source2 dialog:
  CW Frequency = 500 MHz

---

Test Set Reference Switch

PNA models with option 081 and all PNA-X models have a switch in the test set that allows you to bypass the port 1 reference receiver through the front panel Reference 1 connectors. This switch lets you easily switch between standard S-Parameter measurements and measurements using a reference mixer. You could use this feature to make standard S11 measurements and converter transmission measurements relative to a reference ("golden") mixer.

Note: The Frequency Converter Application Option 083 simplifies the task of making extremely accurate phase measurements on MOST frequency converting devices.
How to access the Test Set dialog box

Using front-panel HARDKEY [softkey] buttons

1. Press TRACE/CHAN
2. then [Channel]
3. then [Hardware Setup]
4. then [Path Config]

PNA Menu using a mouse

1. Click Channel
2. then Hardware Setup
3. then Path Config

Test Set dialog box help

Note: This feature is available on PNA 'C' models with Option 081 (external reference switch) and all PNA-X models.

R1 Input Path

Internal: bypass R1 Loop  Connects the port 1 source directly to the R1 receiver.

External: flow through R1 Loop  Allows direct access to the R1 receiver through the Reference 1 front-panel connectors.

See specifications which include a block diagram of reference switch.

Last modified:

30-Apr-2009  Added PNA-X models for test set switch
3-Sep-2008  Removed legacy content
25-Feb-2008  Added link to AM-PM procedure
16-Oct-2007  Minor edits
11/21/06  MQQ Modified for new dialog
Frequency Converting Device Measurements

Many frequency offset measurements can be made using the PNA with option 080. The following is a list of some of those measurements and how they are made.

- Conversion Loss
- Conversion Compression
- Return Loss and VSWR
- Isolation
- Harmonic Distortion

See Also: Frequency Offset Measurement Accuracy
Frequency Offset Measurement Accuracy

This topic discuss methods that can be used to make accurate frequency offset measurements.

- **Calibrations**
- **Mismatch Errors**
- **Accurate and Stable LO**

<table>
<thead>
<tr>
<th>See other Mixer Measurement topics</th>
</tr>
</thead>
</table>

**Calibrations**

With Frequency Offset measurements, the stimulus and response frequencies are different. Standard calibration error terms are calculated using reference measurements. Therefore, traditional calibration methods such as full 2-port SOLT cannot be used with frequency offset.

*Source and Receiver Power calibrations* can be used to calibrate your Frequency Offset measurements.

*Frequency Converter Application* (option 083) offers fully calibrated scalar and vector frequency offset measurements.

**Source Power calibration:**

- Sets accurate power level at stimulus frequencies regardless of the receiver that will be used in the measurement.
- Can be copied to other channels with copy channels feature.
- Can be interpolated.

**Receiver Power Cal:**

- Requires a source cal to have already been performed and applied.
- Cannot be copied to other channels.

**Therefore:**

- Start by performing a source power cal over the combined stimulus and response frequencies.
- **Copy the channel** to other needed channels and the source power cal is copied.
- Change the frequency range of the copied channel to response frequencies.
- Perform a receiver cal at the response frequencies on individual channels.
- Change the frequency range to stimulus frequency and switch frequency offset ON.
- On Status Bar, ensure that source and receiver cals are ON (source cal will be interpolated).

See Frequency Offset Conversion Loss Measurements to see a step-by-step example.

Mismatch Errors
Mismatch errors result when there is a connection between two ports that have different impedances. With S-parameter measurements, these mismatches are measured and mathematically removed during a full 2-port calibration. This is much more difficult with frequency offset measurements. A much easier solution is to use high-quality attenuators on the input and output of the mixer.

By adding a high-quality attenuator to a port, the effective port match can be improved by up to twice the value of the attenuation. For example, a 10-dB attenuator, with a port match of 32 dB, can transform an original port match of 10 dB into an effective match of 25 dB. However, as the match of the attenuator approaches the match of the original source, the improvement diminishes.

**Note:** The Frequency Converter Application (option 083) uses calibration techniques that correct for mismatch errors.

The larger the attenuation, the more nearly the resulting match approaches that of the attenuator, as shown in the following graphic. However, excessive attenuation is not desired because that will decrease the dynamic range of the measurement system.

Accurate and Stable LO
When using frequency offset mode, if the LO signal is not accurate and stable, the output signal will not be at the expected response frequency. As a result, the output signal can fall on the skirts of the PNA receiver IF filter, or fall completely outside of the receiver filter passband.

Also, the LO power level is critical in mixer measurements. Be sure to monitor these power levels closely.
Conversion Loss (or Gain)

- What is Conversion Loss?
- Why Measure Conversion Loss?
- How to Measure Conversion Loss

**What is Conversion Loss?**
Conversion loss is defined as the ratio of the power at the output frequency to the power at the input frequency with a given LO (local oscillator) power. This is illustrated in the graphic below. A specified LO power is necessary because conversion loss varies with the level of the LO, as the impedance of the mixer diode changes.

![Conversion Loss Diagram](image)

**Why Measure Conversion Loss?**
Conversion loss (or gain in the case of many converters and tuners) is a measure of how efficiently a mixer converts energy from the input frequency to the output frequency. If the conversion loss response of a mixer or converter is not flat over the frequency span of intended operation, valuable information may be lost from the resulting output signal.

**How to Measure Conversion Loss**
Conversion loss is a transmission measurement. It is measured by applying an input signal (stimulus) and an LO signal at specific known power levels, and measuring the resulting output signal level. Because the output frequency is different from the input frequency, frequency offset mode (option 080) must be used for this measurement.
Note: This measurement is made much easier if your PNA has the Frequency Converter Application.

Equipment Setup

Example: A calibrated Conversion Loss (Down-converter) measurement

Swept Input with Fixed LO = Swept Output

- RF Input: 3.1 - 3.3 GHz
- LO: 2.2 GHz
- IF Output: 900 - 1100 MHz

PNA setup and calibrate on channel 1

1. On channel 1 create an unratioed R measurement over the ENTIRE input and output frequency span (.9 - 3.3 GHz). This will be the base source power cal that will be copied to the R and B channel measurements.
2. Perform a source calibration using a power meter. This makes the power level at the input of the mixer very accurate.

Setup Reference measurement on channel 2

1. Copy channel 1 to channel 2 which will display the reference input to the mixer. The channel 1 source power cal is copied with the other channel settings.
2. Change measurement to R1 unratioed.
3. Change RF Input frequency to 3.1 - 3.3 GHz. The source power cal becomes interpolated.
4. Perform receiver power cal. Do not need to make physical connections. The PNA source is internally connected to the R1 receiver. Makes the R receiver read the source power level.
**Setup B measurement on channel 3**

1. Copy channel 1 to channel 3. This channel will display the output of the mixer. The channel 1 source power cal is copied with the other channel settings.

2. Change measurement to B unratioed.

3. Change IF Output frequency to .9 - 1.1 GHz. This causes the source power cal becomes interpolated.

4. Connect thru line from port 1 to port 2.

5. Perform receiver power cal. This makes the B receiver read the source power at the IF Output frequencies.

6. **Turn OFF receiver power cal.** This prevents an error when changing to input frequencies (next step).

7. Change RF Input frequency to 3.1 - 3.3 GHz. This changes the channel back to the mixer RF Input frequencies.

8. **Enable Frequency Offset.**

9. Change Offset to (-2.2 GHz). This tunes the B receiver to the IF Output frequencies .9 to 1.1 GHz. **Note:** The minus sign indicates a down-converter measurement.

10. Turn ON receiver power cal.

**Measure the Mixer**

1. Connect the mixer.

2. Adjust **scale** to suit your needs.

3. Enable **markers** to read power levels for each trace.

The display below shows:

- Ch3 B receiver (bottom trace) absolute output power.

- Ch2 R1 receiver measurement (top trace) absolute input power to the mixer.

With this method, the conversion loss math (B/R1) can be performed with **Equation Editor** (not shown). The B/R1 ratio measurement is not supported with receiver power Cal turned on. However, conversion loss (C21) measurements can be made directly and are much easier using the Frequency Converter Application, FCA (Opt 083).
**Conversion Compression**

- What is Conversion Compression?
- Why Measure Conversion Compression?
- How to Measure Conversion Compression
- Measurement Accuracy Considerations

---

**What is Conversion Compression?**

Conversion compression is a measure of the maximum input signal level for which a mixer will produce linear operation. It is very similar to the gain compression experienced in amplifiers.

To understand conversion compression, you must first understand conversion loss. This is the ratio of the mixer output level to the mixer input level. This value remains constant over a specified input power range. When the input power level exceeds a certain maximum level, the constant ratio between input and output power levels begins to change. The point at which the ratio has decreased 1 dB is called the 1-dB compression point. This is illustrated in the graphic below.

![Diagram of IF Conversion Loss Ratio (dB) vs. Input Signal RF showing 1 dB compression point and ratioing shows compression point.]

**Why Measure Conversion Compression?**

Conversion compression is an indicator of the dynamic range of a device. Dynamic range is generally defined as the difference between the noise floor and the 1-dB compression point.

**How to Measure Conversion Compression**

The equipment and setup used to measure conversion compression are essentially the same as for measuring...
conversion loss and is illustrated in the following graphic.

The PNA performs a power sweep using frequency-offset mode and the resulting display shows the mixer's output power as a function of its input power. The 1-dB compression point (or others such as 3-dB) can be determined using markers.

---

**Measurement Accuracy Considerations**

**Equipment Setup Considerations**

- The couplers in the PNA have very good directivity. If the return loss of the DUT is bad, the reflected signal gets sampled by the PNA and can result in errors. This relates to error in DUT gain. To increase the accuracy, an attenuator can be added between the PNA's source port and the DUT's input port. Normally a 6- to 10-dB attenuator is sufficient. Addition of this attenuator, however, decreases the available drive to the DUT.

- With high drive levels the PNA can be driven into compression resulting in measurement error. With excessive drive levels, the PNA can be damaged. Add an attenuator between the output of the DUT and the receiver input of the PNA to avoid these problems.

**Calibration Considerations**

- Source power calibration can be used to provide a high level of accuracy for this measurement.
Isolation Measurements of Frequency Converting Devices

- What is Isolation?
- Why Measure Isolation?
- How to Measure Isolation

See other Frequency Converting Device Measurements

What is Isolation?
Isolation is a measure of the leakage, or feedthrough, from one port to another. The more isolation a mixer provides, the lower the amount of feedthrough. Isolation is measured at the same frequency as the stimulus, not the converted or shifted frequency. Therefore, Frequency Offset capability is not necessary for these measurements.

Three main isolation terms are of interest for mixer measurements:

- LO-to-OUT isolation ($V_{LO}$)
- LO-to-IN isolation ($V_{LO}$)
- IN-to-OUT feedthrough ($V_{IN}$)

Why Measure Isolation?
Any unwanted signal "leaking" through the device will mix with the desired output signal creating intermodulation products, adding to intermodulation distortion. These unwanted signals may be difficult to filter out.

How to Measure Isolation
Use the following setups to measure the isolation of a mixer:
Note the following:

- The Input to Output isolation is very dependent on the LO power level. Isolation should be measured with the LO power at its normal operating level.

- Each of the ports not being tested should be terminated with an impedance typical of actual operation. This may not always be the characteristic impedance, Z₀ (usually 50 or 75 ohms). For example, if the OUT port of a mixer is intended to be directly connected to a filter, then this filter should be used when measuring the LO-to-IN feedthrough.
Measuring Converters vs. Mixers

Measuring IN-to-OUT feedthrough of a converter is identical to that of a mixer. The IN-to-OUT feedthrough is generally very small for a converter due to the inclusion of an IF filter in the device. Because of this, the measurement may require the PNA to have increased *dynamic range*.

Measuring LO leakage (LO-to-OUT and LO-to-IN) of a converter requires a different technique because the LO port is typically not accessible:

- The PNA can be tuned to the frequency of the LO signal and either the OUT or IN port connected to the PNA receiver port. The PNA source port is not connected.

- A spectrum analyzer can be connected to either the OUT or IN port and tuned to the frequency of the LO signal.
Harmonic Distortion

- What is Harmonic Distortion?
- Why Measure Harmonic Distortion?
- How to Measure Harmonic Distortion
- Measurement and Accuracy Considerations

What is Harmonic Distortion?

Harmonics are multiples of any signal appearing at the mixer input and also multiples of the LO input. The distortion of the mixer's output characteristics caused by these harmonics is referred to as harmonic distortion. Harmonic distortion is caused by non-linearities in the device.

Harmonics are NOT signals created by two or more signals interacting (mixing); these signals are known as intermodulation products, which result in intermodulation distortion.

Why Measure Harmonic Distortion?

- It can degrade the performance of devices connected to the output of the mixer.
- The harmonics can also mix with other signals present in the mixer, adding to the intermodulation distortion of the mixer.

How to measure Harmonic Distortion

The harmonics can be measured using the PNA with Frequency Offset (option 80). The frequency of the LO to the mixer is set to zero and multiplier of the RF input is used to set the IF frequency (the harmonic). The equipment setup is shown below.

Since harmonics are specified in dBc, the fundamental RF and both the second and third harmonics are measured and the differences calculated. Multiple channels can be used to do this.

1. Connect the equipment.
2. Setup the measurement for calibration. See also Measurement and Accuracy Considerations.

   Use three channels and frequency offset mode:
   - Channel 1 = F1 to F2
   - Channel 2 = F1 to 2F2 (frequency offset mode, multiplier = 1)
   - Channel 3 = F1 to 3F2 (frequency offset mode, multiplier = 1)
Perform a source power calibration and receiver power calibration over the entire frequency range. See Measurement and Accuracy Considerations.

- Reduce the frequency span and increase the frequency offset multiplier on Channels 2 and 3:
  - Channel 2 = F1 to F2 (frequency offset mode, multiplier = 2)
  - Channel 3 = F1 to F2 (frequency offset mode, multiplier = 3)
  **Note:** Because the frequency span has been changed from that used for calibration, the source and receiver calibrations will be interpolated.

- Connect the DUT, make the measurement, and calculate the harmonic response:
  - Set up markers on Channels 1, 2 and 3, and determine the difference between the marker values to get the dBc value of each harmonic.
    - Channel 1 - Channel 2 = 2nd harmonic (dBc)
    - Channel 1 - Channel 3 = 3rd harmonic (dBc)
  **Note:** Be sure to set the markers to the appropriate stimulus. Channel 2 markers should be set to twice the frequency of Channel 1 markers. Channel 3 markers should be set to three times the frequency of Channel 1 markers.

**Measurement and Accuracy Considerations**

**Equipment Setup Considerations**

- A filter must be used at the input of the mixer to remove the PNA source harmonics.
Return Loss and VSWR

- What are Return Loss and VSWR?
- Why Measure Return Loss and VSWR?
- How to Measure Return Loss and VSWR

See other Frequency Converting Device Measurements

What is Return Loss and VSWR?

Return loss and VSWR are both linear reflection measurements, even when testing frequency conversion devices, because the reflected frequency is not converted. These measurements are essentially the same as for filters and amplifiers. Learn more about Reflection Measurements.

Why Measure Return Loss and VSWR?

Devices which have poor return loss and VSWR result in loss of signal power or degradation of signal information.

How to Measure Return Loss and VSWR

Setup the PNA measure return loss and VSWR as you would any two-port device. Connect your frequency converting device as shown in the following diagrams:

RETURN LOSS AND VSWR OF MIXER INPUT PORT

RETURN LOSS AND VSWR OF MIXER OUTPUT PORT
RETURN LOSS AND VSWR OF MIXER LO PORT
**FIFO (Opt 118) and other Antenna Features**

The following features, used with the PNA-X models and the N5264A, were designed specifically for Antenna applications.

- **Fast Sweep Features (Opt 118)**
  - FIFO Buffer and Fast CW
  - FIFO Buffer and Fast Groups
  - FIFO Buffer and Fast Segments

- **Other Useful Antenna Features**
  - Point Averaging
  - Point Sweep
  - Trace Triggering

- **See Also**
  - N5264A
  - Pulsed Measurements
  - Frequency (Security) Blanking
  - External Triggering

---

**Fast Sweep Features (Opt 118)**

The following features allow you to **very quickly** measure and download data to a remote computer.

- **Fast CW, Fast Groups**, and **Fast CW Segments** all work ONLY with the FIFO Data Buffer.

- These features can be used ONLY with **SCPI or COM commands**. COM is faster than SCPI when using **DataInCompactForm**. Otherwise, SCPI is faster than COM.

- These features are all demonstrated in the following example program:

  Download and install the **FIFO Tester** - a C# program that uses either the COM or SCPI interface to download data to, and read data from, the FIFO buffer. Source code is also provided. See [http://na.tm.agilent.com/pna/apps/applications.htm](http://na.tm.agilent.com/pna/apps/applications.htm)
The FIFO Data Buffer

The FIFO (First-IN, First-OUT) data buffer is a circular buffer that allows very fast Read-Write access.

- When enabled, all the data gathered is placed into a 4 GB FIFO buffer.
- You can write to, and simultaneously read from, the FIFO buffer.
- A maximum of 1 million data points can be read for each query.
- REAL / IMAGINARY pairs is the ONLY supported format for the FIFO buffer.
- A preset or instrument state recall will turn off the FIFO buffer collection.
- When more than one measurement is present, data from each measurement is stored in the FIFO buffer in the following order. These measurements are separated into lines for easier reading.
  - R, A, B, C, D,
  - R/R, A/R, B/R, C/R, D/R,
  - R/A, A/A, B/A, C/A, D/A,
  - R/B, A/B, B/B, C/B, D/B,
  - R/C, A/C, B/C, C/C, D/C,
  - R/D, A/D, B/D, C/D, D/D

Fast CW

In Fast CW mode the PNA display is not updated. There is no background computation or other ‘interference’ from the PNA computer. Therefore, data is acquired real-time.

The following requirements must be met before sending the Fast CW command.

- FIFO is ON
- A single channel is being measured. Other channels can be in Hold.
- All measurements are acquired in a single sweep.

**IMPORTANT - Fast CW and IF Bandwidth setting**

- IF Bandwidth of **10 kHz and lower** - Data is transferred immediately to the FIFO after every acquisition.
- IF Bandwidths **greater than 10 kHz** - Data is transferred to the FIFO in groups. A triggered acquisition is NOT placed into the FIFO buffer until either the total number of points is completed, or an intermediate group of points is finished. The number of points within a group differs for each IF Bandwidth setting.

Notes:
• See example programs in SCPI and COM.
• Fast CW sets the number of data points, overwriting the standard channel setting.
• When exiting Fast CW, the FIFO data buffer is cleared.
• External trigger signals are allowed only through the rear-panel Trig In connector - NOT the Aux1 and Aux2 In connectors.
• An error message appears if triggering is sent to the PNA faster than it can respond.

**Fast Groups with FIFO Data Buffer**

With this speed optimization feature, interaction with Windows or other PNA 'overhead' calls are suspended, allowing very fast and predictable measurement timing.

Fast Groups is automatically enabled when the following **requirements** are met:

• FIFO is ON
• A single channel is being measured. Other channels can be in Hold.
• All measurements are acquired in a single sweep.
• Group trigger is enabled with count > 1.

**Notes:**

• Fast CW can **NOT** be used with Fast Groups.
• Fast Groups and Fast CW Segments were designed to be used together, but not required.
• The FIFO Tester example program demonstrates this feature.

**Fast CW Segments with FIFO Buffer**

In this optimization feature, each CW segment (where the start and stop frequency is identical) within a channel is measured at speeds as fast as the Fast CW mode sweep.

Fast CW Segments is automatically enabled when the following **requirements** are met:

• FIFO is ON
• Start and stop frequency of a segment is identical.
• External trigger signals are allowed only through the rear-panel Trig In connector - NOT the Aux1 and Aux2 In connectors.

**Notes:**

• Fast CW can **NOT** be used with Fast CW Segments.
• The sweep can include non-CW segments, but these are not acquired in Fast mode.
Fast Groups and Fast CW Segments were designed to be used together, but not required.

The FIFO Tester example program demonstrates this feature.

In Fast CW Segments, when data is not being acquired in real-time, the following message appears: 
**Caution: Sweep time jitter. Try reducing the number of segments.** To avoid this error, reduce the number of segments in the channel.

### Other Antenna Features

**Point Averaging**

This feature is selected on the Average dialog.

When selected, each data point is measured the specified number of averages before stepping to the next data point. When point trigger is selected, only one trigger is required for each data point regardless of the number of averages.

**Point Sweep**

This feature is selected on the Sweep Setup dialog.

In Point Sweep mode, the PNA measures both the forward and reverse parameters at each frequency point before stepping to the next frequency. The display trace is updated as each data point is measured. Point sweep is the same as stepped sweep mode of the 8510 and 8530.

**Trace Triggering**

This feature is selected under Trigger Mode on the Trigger dialog.

Available ONLY when Point Sweep is selected. Each trigger signal causes two identical measurements to be triggered separately - one trigger signal is required for each measurement. Other trigger mode settings cause two identical parameters to be measured simultaneously.

Trace triggering is NOT permitted when a channel is using a 2 port (or more) S-Parameter calibration.

### Other Useful Antenna Features

- **Pulsed Measurements**
- **Frequency (Security) Blanking**
- **External Triggering**
Gain Compression for Amplifiers  GCA (Opt 086)

- What's New
- Features, Requirements, and Limitations
- Gain Compression Concepts
- Understanding the GCA Displayed Traces
- Gain Compression Parameters
- Compression Methods
- Acquisition Modes
- Using Gain Compression App
  - Frequency tab
  - Power tab
  - Compression tab
  - Safe Sweep Mode dialog
- Compression Analysis
- Saving GCA Data
- GCA Measurement Tips
- Macros

See Also

- Gain Compression for Converters
- GCA Calibration
- Programming commands
- App Note Amplifier Linear and Gain Compression Measurements

Other PNA Applications

Features, Requirements, and Limitations
Features

- Fast, easy, and complete Gain Compression measurements for amplifiers.
- Many compression parameters to choose from, including gain, input power at compression, output power at compression, input match, and compression level.
- Several compression methods to choose from, including deviation from linear gain, deviation from max gain, back-off, and X/Y, and compression from saturation.
- Three acquisition methods to choose from: Power per Freq, Freq per Power, and SMART Sweep
- SMARTCal Calibration Wizard to guide you through Full 2-Port or Enhanced Response calibration, plus Source Power calibration.
- Compression Analysis allows traditional power sweep view at a selected frequency.
- Receiver Leveling provides continuous source power accuracy.
- Supports Wideband and Narrowband Pulse measurements using the new integrated Pulse setup dialogs.

Requirements

- PNA-X or N522x with Opt 086 (software option only) must be enabled.
- When performing an optional calibration:
  - ECal module or Calibration Kit
  - Power meter/sensor

Limitations with GCA

- Number of points limited to 20,001 for two-dimensional acquisitions, 10,000 points for SMART Sweep.
- Standard CW power sweep is NOT supported in a Gain Compression channel.
- No Independent IFBW, Independent power levels in segment table.
- Stepped sweep mode only.
- Linear, Log, and Segment frequency sweep modes only.

The following PNA Features are NOT Available in a Gain Compression channel:

- Unratioed receiver measurements (A, B, R)
- ECal User Characterization
- Some Fixturing Features
• FOM or FCA
• External Test Set Control (Option 551)
• Interface Control
• IF Path Configuration
• Time Domain
• Balanced measurements
• Point and Sweep trigger
• Save Auto Formatted Citifile data.

Gain Compression Application Concepts

What is Gain Compression
An amplifier has a region of linear gain, where the gain is independent of the input power level. This gain is commonly referred to as small signal gain. As the input power is increased to a level that causes the amplifier to approach saturation, the gain will decrease. The 1 dB gain compression is defined as the input power level that causes amplifier gain to drop 1 dB relative to the linear gain.

You can quickly measure the gain compression using a compression marker on a power sweep trace.

Terms used in GCA

Linear Power Level The specified input power that yields linear gain (also known as 'small-signal gain') in the amplifier.

Reference gain The measured gain that is used as a reference for determining compression level. The Compression Method that is used could cause this value to be different.

Compression level The specified amount of gain reduction from the reference gain.

Target gain The gain at the specified compression level. Although this term does not appear in GCA, it is important to understand when discussing the various compression parameters.

For example, when using Compression from Linear Gain method with the following settings:

• Linear gain (measured at Linear Input power) = 10.2 dB
• Compression level (specified) = 1 dB
• Target gain = 9.2 dB

This is called ‘Target’ gain because GCA will search for the closest measured gain to 9.2000 dB. It may not measure this gain exactly.

**Compression point** The operating point at which the measured gain is closest to the Target Gain. All [compression parameters](#) report data for this operating point.

**Understanding the GCA Displayed Traces**

One of the most important concepts to remember with GCA is that, each frequency data point represents many measurements using different input power levels.

Some things to notice about how GCA displays **compression** data:

1. The X-axis values are ALWAYS frequency. Imagine behind each frequency data point, a traditional power sweep curve with corresponding measurements and calculations to find the specified compression point.

2. The Y-axis values are always reported at the **compression point**. The value that is displayed depends on the **compression** parameter that you choose. The S-parameters that are displayed in a GCA channel are always measured at the **linear and reverse** power level.

**Example:** Five of the six GCA compression parameters are displayed in the above image. The missing trace, DeltaGain21 is discussed below.

• Markers are placed at 4.549 GHz for all of the parameters.

• **Tr 3 Compln21** (Input power at the compression point) shows the marker value to be **-5.4117 dBm**. This is the power into the DUT that was required to achieve the compression point. Notice that this is about the same input power required to achieve the specified compression at ALL frequencies.
• Tr 5 CompGain21 (Gain at the compression point) shows the marker value 9.6443 dB. This is the measured gain at the compression point.

• To see the gain at a different input power at this frequency, use the Compression Analysis feature.

Gain Compression Parameters
There are several Gain Compression parameters, as well as standard S-parameters and ADC parameters, that can be measured in a GCA channel.

How to add GCA Parameters
First, create a GCA channel. Learn how. The default parameter is S21.

Using front-panel HARDKEY [softkey] buttons
1. Press Traces
2. then [New Trace]
3. then select a parameter

Using a mouse with PNA Menus
1. Click Trace/Chan
2. then New Trace
3. then select a parameter

Linear S-Parameters
For convenience, the standard S-parameters are offered in a GCA channel. S11 and S21 are measured at the specified Linear Input level. S22 and S12 are measured at the specified Reverse power level.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>When Measured</th>
</tr>
</thead>
<tbody>
<tr>
<td>S11</td>
<td>Input Match</td>
<td>Always</td>
</tr>
<tr>
<td>S21</td>
<td>Gain</td>
<td>Always</td>
</tr>
<tr>
<td>S22</td>
<td>Output Match</td>
<td>See Reverse</td>
</tr>
<tr>
<td>S12</td>
<td>Reverse Isolation</td>
<td>See Reverse</td>
</tr>
</tbody>
</table>

ADC Parameters
Four ADC analog-to-digital converter measurements are offered in a GCA channel:

• AI1 and AI2 are measured at the specified Linear Input level.

• CompAI1 and CompAI2 are measured at the specified compression point.

These DC measurements, along with Equation Editor, allow you to make PAE measurements at the Linear Input
level and compression point.

These measurements are made at pins 7 and 8 of the PNA Power IO connector. Learn more about ADC measurements.

Compression Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ComplIn21</td>
<td>Input power at the compression point.</td>
</tr>
<tr>
<td>CompOut21</td>
<td>Output power at the compression point.</td>
</tr>
<tr>
<td>CompGain21</td>
<td>Gain at the compression point.</td>
</tr>
<tr>
<td>CompS11</td>
<td>Input Match at the compression point.</td>
</tr>
<tr>
<td>RefS21</td>
<td>Linear Gain value used to calculate the compression level.</td>
</tr>
<tr>
<td>DeltaGain21</td>
<td>CompGain21 MINUS Linear Gain (in Log Mag format). This trace can be used to learn a lot about the DUT compression point. Learn more.</td>
</tr>
</tbody>
</table>

Note: The following table assumes: DUT Input = PNA port 1 and DUT Output = PNA port 2. When the Port mapping is different, the parameters in GCA are updated accordingly. For example, with Input = port 2 and Output = port 1, then "ComplIn12" would be displayed.

The raw data for these parameters are always measured.

Compression Methods

GCA offers the following methods to find the compression point of an amplifier using GCA:

Compression from Linear Gain

The Reference Gain is measured using the specified Linear (Input) Power Level. The Target Gain is calculated as the Linear Gain minus the specified Compression Level. For example 8.3 dB - 1 dB = 7.3 dB.

Compression from Max Gain

The linear region of an amplifier gain may not be perfectly linear. The highest gain value that is found at each frequency is used as the Reference (S21) Gain. The Target Gain is found in the same way as Compression from Linear Gain.

Compression from Saturation

This method is used to better find the compression point when measuring amplifiers with non-linear gain as shown in the following image:
The Max power out value * is found at each frequency. Then input power is lowered until the output power decreases by the specified 'From Max Pout' value. This is the compression point.

**Backoff and X/Y method**

These two compression methods are very similar.

- Both methods specify a difference in input power (X axis) between the linear region and compression point.
- For the Y-axis difference:
  - **Backoff method** specifies Compression Level which is a difference in Gain.
  - **X/Y method** specifies Delta Y which is a difference in Output Power.

GCA searches for these points differently for **2D sweeps** and **SMART sweep**.

The following images show how Backoff and X/Y method is calculated at ONE frequency.
The compression point (yellow circle) is where 10 dB more input power yields only 9 dB more output power than at the reference point (blue circle).

**Acquisition Modes**

The GCA offers three modes for data acquisition: Two 2D sweep modes, and SMART sweep.

To see a traditional power sweep at a single frequency, use the Compression Analysis feature. Learn more.

**2D (two-dimensional) Sweeps**

This is the easiest method to understand, and the least efficient for finding the compression point. Both 2D sweep modes work as follows:

1. All GCA measurements begin by measuring S-parameters at the specified Linear Power level. Reverse parameters are measured ONLY if Full 2-port calibration is applied or if a reverse parameter is displayed. Learn more about Cal choices.

2. Gain measurements are then made at ALL of the specified frequency and power values. Although these are conceptually 2-Dimensional sweeps, a single sweep is constructed in firmware. See Data Points Limit.

3. After data has been measured, a search is performed to find the compression point. You can choose to interpolate between the two measured points closest to the target gain. Learn more.

As each sweep is performed, dots are plotted next to the Ch indicator in the lower left corner of the display to indicate progress for the current sweep.

**Note:** For Backoff and X/Y compression method, GCA does not verify that the specified Start - Stop power range is at least the size of the specified Backoff or X value. The closest compression point is always reported.

**2D Sweep Modes**
- **2D Sweep Power per Frequency** - Input power is stepped from Start to Stop at each specified frequency. From the following example you can see that the device is exposed to the highest power level (p3) at the first frequency (f1). This could heat the device early in the measurement and affect compression results.

The following examples show (frequency, power) values for three frequency points and three power points, resulting in a total of 9 measurements:

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>f1</td>
<td>p1</td>
<td>f2</td>
<td>p1</td>
<td>f3</td>
<td>p1</td>
<td>f4</td>
<td>p1</td>
<td>f5</td>
<td>p2</td>
</tr>
</tbody>
</table>

- **2D Sweep Frequency per Power** - Frequency is swept from start to stop at each specified power level as follows:

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>f1</td>
<td>p1</td>
<td>f2</td>
<td>p1</td>
<td>f3</td>
<td>p1</td>
<td>f4</td>
<td>p1</td>
<td>f5</td>
<td>p2</td>
</tr>
</tbody>
</table>

**Viewing and Saving 2D Data**

It is NOT possible to plot ALL of the 2D measurement data on the PNA display. However, it can be saved to a *.csv file and then read into an Excel spreadsheet. The initial S-parameter measurement data is not saved to this file. Learn more.

You can also view on the PNA all power sweep information at a selected frequency using the Compression Analysis feature.

**SMART Sweep**

SMART Sweep is usually the fastest and most accurate method to measure Gain Compression. Unlike the 2D acquisition modes which measure all of the specified frequency / power points, SMART Sweep performs a series of power search iterations. At each frequency, an 'intelligent guess' of input power is made to find the compression level that is within tolerance. This guess is further refined with each successive power search iteration sweep.

SMART Sweep continues to iterate until one of the following conditions occur:

1. **ALL data points are within tolerance.** When the compression level for a data point achieves the specified tolerance, it continues to be measured and input power changed to improve the measurement within tolerance.

2. The specified compression level can NOT be achieved for the remaining frequencies that are not in tolerance. Either the Start power is too high or the Stop power is too low.

3. **Maximum iterations have been achieved.** If a measured gain is not within the specified tolerance before the specified Max number of Iterations has been reached, then the last power reading is used as the compression point.
The Iteration Counter, Dots, and Bangs(!)

Next to the Ch indicator, in the lower left corner of a GCA window, the following annotation appears:

- An **iteration counter** is incremented each time input power is adjusted.
- A **dot** appears when another 10% of the frequency points are within tolerance.
- ! (bangs) are displayed after the last iteration. Each bang represents 10% of the data points that are NOT within tolerance.

---

SMART Sweep and Compression Method

The intelligent guess process works differently depending on the compression method. This is important because Backoff and X/Y compression methods subject the DUT to significant changes in input power during an iteration sweep. This can affect the DUT and the measurement results.

Learn more about Backoff and X/Y compression methods.

ALL GCA measurements begin by measuring S-parameters at the specified Linear Power level. Reverse parameters are measured ONLY if Full 2-port calibration is applied or if a reverse parameter is displayed. Learn more about Cal choices.

- **Backoff and XY** Because both compression methods specify the separation between the "linear" region and the "compressed" region, each iteration requires a single sweep at two **dramatically** different power levels over the same frequency range. The first half of the sweep measures the DUT at the Backoff or X power level. The second half of the sweep measures the DUT at the compressed power level, specified by the **Start and Stop** power range. At the beginning of the second half, the power level rises by the Backoff or X value. The specified **Settling Time** is applied at this point to allow the DUT time to react to this significant change in power level. **Safe Sweep** does NOT minimize this change in input power. However, Safe Sweep with Backoff and XY methods **DOES** prevent the DUT from being exposed to too much input power.

- **Compression From Linear Gain** After the reference gain is measured at the linear input power, the next iteration measures the DUT at a higher power level which attempts to push the DUT well into compression. Subsequent sweeps, depending upon the compression level of the DUT, either increases or decreases the power in order to reach the desired compression level. Usually, by the third iteration sweep, a curve-fit algorithm is utilized to precisely find the compression point.

**Note:** The DUT can be subject to significant changes in power from one iteration sweep to the next. This can be minimized by the use of **SAFE Sweep** and careful selection of the corresponding settings.

- **Compression from Max Gain** The maximum gain that is found at each frequency is stored and used to calculate the compression point. SMART Sweep does NOT perform extra iterations to search for the maximum possible gain of the amplifier at each frequency.

- **Compression from Saturation** The maximum power out that is found at each frequency is stored and used to calculate the compression point. SMART Sweep does NOT perform extra iterations to search for the maximum possible power out of the amplifier at each frequency.
Using the Gain Compression Application

The following is a general procedure for performing a GCA measurement. The challenge with GCA is configuring a measurement that yields the true compression performance of YOUR DUT. This requires knowledge of the Gain Compression settings and knowledge of the DUT.

See specific dialog boxes below.

1. Disconnect the DUT if preset or default power levels may damage the PNA or DUT.
2. **Preset** the PNA, or configure a suitable **User Preset** that will be safe in case the DUT is connected.
3. Create a GCA channel. **Learn how.** The default trace is S21.
4. Start **GCA Setup dialog** and configure the measurement settings based on the DUT, adapters, attenuators, booster amplifiers, and fixtures to be used in the measurement.
5. Save the **instrument state** (optional).
6. Connect DUT and apply bias and RF power as appropriate. The default measurement for a GCA channel is S21 (amplifier gain). Inspect the gain measurement to ensure the DUT is operating as expected.
7. Add GCA compression parameter traces. **Learn how.**
8. Adjust the measurement settings to yield satisfactory compression parameters. See **GCA Measurement Tips.**
9. Start and complete the **GCA Calibration wizard.**

**How to start the Gain Compression Setup dialog**

To provide quicker access, use the Setup softkey. **Learn how.**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **FREQ**
2. then **[Gain Compression Setup]**

**Using a mouse with PNA Menus**

1. Click **Stimulus**
2. then **Frequency**
3. then **Gain Compression Setup**

**Frequency tab - Gain Compression** dialog box help
Configures the frequency settings over which Gain compression is to be measured, as well as the measurement method.

**Sweep Type**

Choose a method in which to sweep frequency: Linear, Log, and Segment Sweeps. This setting applies to all data acquisition modes.

**Notes**

- Log and Segment Sweep are NOT available on GCX.
- CW Sweep is NOT available in GCA. However, to see a traditional power sweep at a single frequency, use the Compression Analysis feature.
- To use CW Sweep in GCX, set all ranges to Fixed on the Mixer Frequency tab. This has the same effect as setting all ranges to Start = Stop Frequency.

**Segment Sweep Notes (GCA ONLY)**

- The segment table shown on the dialog is ‘READ-ONLY’.
- Learn how to Create and edit the Segment Sweep table.
- Independent IFBW and Power are NOT available.
- X-axis point spacing is available beginning with A.09.10.

**Data Acquisition Mode**

Specifies HOW the gain compression data is collected.

**SMART Sweep**

- At each frequency, input power is ‘intelligently’ adjusted to find a measured gain equal to the target gain.
- Faster and more accurate than 2D sweeps to measure Gain Compression point at a number of frequencies.
- Learn ALL about SMART Sweep
2D (two-dimensional) Sweeps

- **Sweep Power per Frequency**  Performs a series of power sweeps at each successive frequency.
- **Sweep Frequency per Power**  Performs a series of frequency sweeps at each successive power level.
- **Learn ALL about 2D sweeps**

**Sweep Settings**

Click each to learn more about these settings.

- **Number of points**  Number of frequency points to measure. The Frequency points may be limited due to the number of specified Power points. See Data Points Limit.

- **IF Bandwidth**  Set this value to yield acceptable trace noise when measuring gain at the linear power level. This level of noise contributes directly to the accuracy of compression point. A lower value (narrower IFBW) allows for more accurate, but slower, measurements. See GCA Measurement Tips to see how to best set IFBW.

- **Start / Stop, Center / Span**  frequencies. Set the frequency range over which to measure Gain compression.

**Data Points Limit**

The maximum number of measurement data points depends on Acquisition method and Compression method as follows:

<table>
<thead>
<tr>
<th></th>
<th>SMART sweep</th>
<th>2D sweep</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Compression method</strong></td>
<td>Number of frequency points is reduced to ensure the total number of data points does not exceed the specified limit.</td>
<td>Number of power points is reduced to ensure the total number of data points does not exceed the specified limit.</td>
</tr>
</tbody>
</table>
| Compressed from linear gain | Data points = freq points  
Max = 20,001 | Data points = (freq. points) * (power points)  
Max power points = 2,001  
Max data points = 20,000 |
| Compressed from max gain | Data points = freq points  
Max = 20,001 | |
| X/Y and Backoff | Data points = 2 * freq points  
Max = 20,001 | |

**Note:** Although the dialog box will allow you to enter any number of frequency or power points, the values are checked when OK or Apply is pressed. If a limit is exceeded, the relevant data points are reduced to the maximum allowable number without warning.
Power tab - Gain Compression dialog box help

Configures RF power and Power Sweep settings for Gain Compression measurement.

**Power ON (All channels)** Check to turn RF Power ON or clear to turn power OFF for all channels.

**Input Port**
- **Select** the PNA port that is connected to the DUT Input.
- **Linear Power Level** The input power that yields the linear gain of the DUT. The linear gain is used as the reference gain when calculating the **Compression from Linear Gain**. Input match is also measured at this power level.
- **Source Attenuator** Specifies the attenuator setting associated with the port connected to the input of the DUT. This attenuator will affect the range of available power into the DUT. All PNA channels in continuous sweep must have the same attenuation value. Learn more about Source Attenuation.

**Output Port**
- **Select** the PNA port that is connected to the DUT Output.
- **Reverse Output Power** Sets power level into the output of the DUT for reverse sweeps. Port power is automatically uncoupled.
  
  Reverse power is applied to the DUT ONLY under the following conditions. Otherwise, this setting is ignored.
  
  - When Linear Output Match or Linear Reverse Isolation parameters are requested.
When Full 2-port correction is used, you can perform a full 2-port cal and downgrade to an Enhanced Response Cal to prevent reverse power from being applied to the DUT. Learn more.

**Source Attenuator** Specifies the attenuator setting for the port connected to the DUT output. This setting will affect the range of available power at the DUT output port.

**Receiver Attenuator** Specifies the attenuator setting for the receiver associated with the DUT output port.

**Source Leveling** Specifies the leveling mode. Choose from: Internal (normal operation) or Open Loop (used only for [Wideband Pulse measurements](#)).

### Power Sweep

**Power Points** Number of power points to measure for 2D acquisition modes. The Power Points may be limited due to the number of frequency data points. See Data Points Limit. This setting is NOT available in SMART Sweep, which uses only enough power points to find the specified compression level.

**Start** and **Stop Power**

- **2D sweep** In Backoff, X/Y, and Compression from Max Gain methods, sets the range of power levels that are applied to the DUT to find BOTH the Reference Gain and Compression point. Make sure this range is wide enough to include both. For example, if the Backoff level is 10 dB, then the power range must be greater than 10 dB. Otherwise, GCA will report a compression value using the closest reference gain and compression point, which may be inaccurate. In Compression from Linear Gain, the reference gain is measured at the Linear Power Level, so the Start and Stop power levels are used to find the compression point.

- **SMART sweep** Sets the range of power over which GCA will search for the compression point. The reference gain is found using the Linear Power Level, Backoff, and X values, depending on the Compression Method. To reduce the number of iterations that are required to find the compression point, limit the Start / Stop power range to the input levels that will achieve compression. Do not include the linear region.

**Note:** If your DUT requires more input power to achieve compression below 3.2 GHz, use the PNA-X Hi-power mode, available from the [RF Path Configuration](#) dialog. The disadvantage to this is higher harmonic content.

**Power Step (Size)** Calculated value from current Start, Stop, and Points settings. This setting can NOT be changed directly.

**Path Configuration** click to launch the [RF Path Configuration](#) dialog.

---

**Compression tab - Gain Compression** dialog box help
Compression Method
Learn ALL about these Compression Methods

- **Compression from Linear Gain**  The specified compression level is measured from the linear gain. The linear gain is measured using the **Linear Power Level** that is specified on the Power tab.

- **Compression from Max Gain**  The specified compression level is measured from the maximum gain level. In SMART sweep, the Max Gain value is updated as each iteration occurs. To increase the chances of measuring the actual maximum gain of the amplifier, Safe Sweep should be invoked using low Coarse and Fine increments.

- **Compression from Back Off**  This compression method uses the Compression Level and Back Off values for finding the compression point.

- **X/Y Compression**  This compression method uses the specified parameters (X and Y) as the criterion for finding the compression point.

- **Compression from Saturation**  Similar to Compression from Max Gain, except the specified compression level is measured from the maximum power out level. Use this method to better find the compression point when measuring amplifiers with non-monotonic gain. In SMART sweep, the Max power out value is updated as each iteration occurs. To increase the chances of measuring the actual maximum power out of the amplifier, Safe Sweep should be invoked using low Coarse and Fine increments.

2D Sweep - Compression Point Interpolation
When a 2D Sweep is selected (on the Frequency tab), check this box to calculate and display interpolated compression traces.

The **Target gain** is calculated using a complex linear ratio between the two closest measured values. All compression parameters are then interpolated using this same ratio.

Clear the box to display compression parameters for the closest compression point, either high or low, to the level specified in the Compression Method setting.

SMART Sweep
Learn ALL about Smart Sweep.

**Tolerance**  Specifies an acceptable range for measuring the compression level. Reducing this value can
significantly increase the number of iterations that are required to find the compression point.

**Maximum Iterations** Specifies the maximum number of power search iterations SMART Sweep is allowed. Reducing this value can cause SMART sweep to terminate before all compression levels are found to within the specified tolerance.

**Show Iterations** When checked, the compression parameter traces are updated at the completion of each power search iteration. When cleared, compression parameter traces are updated when SMART Sweep completes the power search iteration process.

**End of Sweep Condition** Specifies the power level applied to the DUT at the completion of a GCA measurement.

GCA performs numerous power and frequency sweeps on the DUT during the overall measurement process. This setting has no affect on these intermediate sweeps. This setting only applies at the end of the very last sweep in the GCA channel.

In addition, this setting applies ONLY to the GCA channel. All other channels operate independently of this setting. Therefore, the power applied to the DUT after all channels have been measured may be different from this setting.

Choose from:

- **Default** Use the default PNA method. [Learn more.](#)
- **RF OFF** RF power is turned off when GCA completes a measurement cycle.
- **Start Power** RF power is set to the start power level.
- **Stop Power** RF power stays at the stop power level.

**Settling Time**

Used ONLY in SMART Sweep when Back Off or X/Y compression algorithms are selected.

This setting allows additional dwell time when the input power changes from the back-off level to the compression level. [Learn more.](#)
Safe Sweep Mode dialog box help

For use with SMART Sweep ONLY.

When enabled, Safe Sweep increases the input power to the DUT by the specified amounts, allowing the compression point to be achieved gradually. While this will increase the number of iterations required to achieve compression, it also minimizes the possibility of driving the DUT too far into compression.

**Note:** Safe Sweep does NOT minimize the dramatic change in input power with Backoff and XY method. However, Safe Sweep with Backoff and XY methods DOES prevent the DUT from being exposed to too much input power. Learn more.

**Safe Mode (Enable)** Check to enable Safe Sweep.

**Coarse Increment** Sets the maximum change in input power, up or down, which will be applied to the DUT from one iteration to the next. Default = 3.0 dB.

Without Safe Sweep, the maximum change in input power can be the entire Backoff or X value when using these compression methods.

**Fine Increment** Once the Fine Threshold has been achieved, this becomes the maximum change in input power, up or down, which will be applied to the DUT. Default = 1.00 dB

**Fine Threshold** Specifies the compression level in which Safe Sweep changes from the COARSE to the FINE increment. Default = 0.5 dB. This means that, by default, the PNA uses the Fine Increment adjustment when compression reaches 0.5 dB.

**Max Output Power** To protect the PNA from damage, when the PNA port that is connected to the DUT Output measures the specified value, the input power to the DUT is no longer incremented at that frequency. In these cases, the compression point would probably not be achieved.

Compression Analysis

Compression Analysis changes the current trace into a power sweep trace at a specified CW frequency. The current parameter and acquisition method is unchanged. For example, with a CompGain21 trace displayed and SMART Sweep selected, enable Compression Analysis. The trace becomes a power sweep trace at the specified CW frequency. The Y-axis displays S21 Gain at each X-axis power point.

When Smart sweep is used, a complete power sweep is not performed, but only the data points that are required to find the compression point. To see a traditional power IN vs power OUT compression sweep, use one of the 2-D acquisition methods.

You can create PNOP or PSAT markers on a CompOut trace with Compression Analysis mode ON. Learn more.
How to perform Compression Analysis

With any compression parameter (such as CompGainS21) displayed:

Using front-panel HARDKEY [softkey] buttons

1. Press Analysis
2. then [Compression Analysis]

Using a mouse with PNA Menus

1. Click Marker/Analysis
2. then Compression Analysis

Programming Commands

Compression Analysis dialog box help

Notes: When an S21 or S11 trace is active, any compression parameter (such as CompGainS21) must also be displayed.

Compression Analysis is NOT allowed for S12 or S22 traces.

Scroll up to learn more about Compression Analysis.

Analysis Frequency: CW  Enter a frequency to use for the compression analysis trace.

Compression Analysis Check to perform compression analysis. A compression trace is displayed at the Analysis (CW) Frequency.

Use Discrete Frequencies  Check to allow Analysis Frequencies at only the discrete points where data is measured. Clear to allow Analysis CW Frequencies that are interpolated from the data points. Then select ANY CW frequency between the start and stop frequencies of the GCA channel.

X-Axis
**Use Measured Pin**  The X-axis displays the actual power that is applied to the DUT after match correction and R-channel drift correction.

**Use Source Pwr Settings**  The X-axis displays the power level of the stimulus.

---

**Saving GCA Data**

Beginning with PNA release A.08.20, GCA data can be saved to a *.csv file in both 2D and SMART Sweep modes (previously only 2D modes). Also, a Delta Gain column has been added to the data.

---

**How to save GCA data**

With a GCA [Compression] trace active:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SAVE</strong></td>
<td>1. Click <strong>File</strong></td>
</tr>
<tr>
<td>2. then <strong>[Save Data As]</strong></td>
<td>2. then <strong>Save Data As</strong></td>
</tr>
<tr>
<td>3. *<em>File Type = GCA Sweep <em>.csv</em></em></td>
<td>3. *<em>File Type = GCA Sweep <em>.csv</em></em></td>
</tr>
</tbody>
</table>

---

**Notes**

- This data type can be read by spreadsheet programs, such as Microsoft Excel.
- Data from the last **complete** sweep is saved to the specified *.csv file.
- If calibration is turned **ON** when the file is saved, then all data is calibrated. Otherwise, raw data is saved.
- All *.csv data saves include a reference power level sweep at the beginning of each frequency data.
The data are organized by frequency / Power IN (as shown above) regardless of the method used to acquire the data. The above image shows SMART Sweep data with 5 iterations and 3 frequency points. The yellow highlight is added here for readability.

**GCA Measurement Tips**

There are many settings in the Gain Compression Application. Here are a few tips when using GCA to learn as much as possible about the compression characteristics of your DUT in the most efficient manner.

**DUT Compression Characteristics and GCA**

Although GCA provides excellent results with a wide variety of amplifiers, it works best with amplifiers which have a monotonic compression curve. In some cases where the compression curve is not monotonic, for example if the amplifier gain expands before it compresses, the correct compression level may not be found.

To help a SMART sweep find the correct compression point, limit the Start and Stop power levels around the anticipated compression point. [Learn more.](#)

The following two power-sweep traces are examples of non-monotonic gain:
**DeltaGain**

A DeltaGain trace is the best way to see how closely GCA is actually measuring to the desired compression level. In addition, you can view the phase of DeltaGain to see the phase deviation between the compressed gain and the reference gain. DeltaGain is calculated as:

- DeltaGain = Measured Gain (watts) / Ref Gain (watts)
- In LogMag format: DeltaGain = (Measured Gain) - (Ref Gain)

With SMART Sweep, DeltaGain (in LogMag format) shows how soon certain frequencies achieve the specified tolerance. Learn more.

Some other settings which may be helpful:

- Trigger source: Manual allows you to analyze data and make adjustments while allowing the device to cool.
- Construct Limit Lines around the compression point at the tolerance level.

The following image shows a DeltaGain21 trace using SMART Sweep. The Limit Lines were added manually.

In the above image:
| Relevant Settings | Method = Compression From Linear Gain  
|                  | Compression level = 1  
|                  | Iteration Tolerance = 0.05 dB.  
|                  | Maximum Iterations = 10  |
| Displayed Results | A data point on -1.00 indicates that, at that frequency, the exact compression level (1 dB) was measured.  
|                  | Several frequencies did not achieve the specified tolerance (0.05 dB) before the Max Iterations (10) was reached.  
|                  |   - FAIL and red data points outside the limit lines.  
|                  |   - Nine dots (....) indicate that 90% of the data points achieved the specified compression level.  
|                  |   - one ! indicates that 10% of the data points did not achieve compression.  
|                  |   - Learn more about the Iteration Counter and annotation.  |

**SMART Sweep Tips**

- Compression from Linear Gain is the easiest compression method to understand and control in SMART Sweep. [Learn more.](#)
- If SMART Sweep requires more than twenty iterations, this is an indication that something is wrong. Try changing the Tolerance setting, Frequency Range, Start / Stop power range, IF bandwidth, or [Dwell Time](#).
- If the number of iterations required to achieve the desired compression level changes significantly from one set of measurements to the next, this could be due to other effects, such as heating. Try increasing the dwell time or using a [wideband pulse](#) measurement configuration.
- If the DUT should not be significantly overdriven into compression, or the changes in the input power should be limited, use [Safe Sweep](#) mode with Deviation from Linear Gain compression method.

**Single Frequency Macros**

**Note:** Beginning with PNA rev. A.09.00, the Compression Analysis feature provides an easier method of viewing a traditional power sweep at a single frequency than the GCA macros. However, the Macros are still maintained on the PNA hard drive.

The macros perform a single power sweep on the DUT using a standard channel with corresponding stimulus settings. The macro can show measurement differences from the compression analysis traces due to bias/thermal/settling effects of the DUT. So, the macro can help confirm a DUT is exhibiting some type of settling behavior which will need to be handled in some way.

Also, the macro is a great GCA programming example.

With a 2D sweep (NOT SMART Sweep) a script that is stored on the PNA hard drive automatically creates a traditional power sweep measurement in a standard channel using the same stimulus setting as the GCA channel. Use a marker in the GCA channel to specify the frequency for the measurement.
The script has two modes of operation:

1. **View Mode** displays all of the previous 2D sweep data at that frequency.
2. **Measure Mode** performs a new measurement at that frequency.

Both modes create a new S-Parameter channel using the same stimulus settings as the GCA channel, including port power, attenuator, IF Bandwidth, and dwell settings. The new channel does not support calibration or pulse characteristics.

To see noise on a measurement, use the **Measure** macro in continuous sweep. Adjust the IFBW and averaging until the noise versus sweep speed meets your needs.

To see other effects of your DUT at a specific frequency, use the **View** macro and the **Measure** macro with 2D sweep mode. Both macros present data using a standard channel. The View macro shows 2D data at a specific frequency, while the Measure macro shows freshly-measured data at the same frequency. Ideally, the data from these two would be identical. However, changes in your DUT behavior due to heating or other effects can cause these to be different. If significant differences exist, try:

- Using the 2D Frequency per Power setting rather than Power per Frequency
- Adjusting the dwell time
- Adjusting IFBW
- Use a [wideband pulse configuration](#)

**How to setup the Macros**

Each macro must be setup separately.

1. Press **Macro**, then **Macro Setup**.
2. Select a blank line, then click **Edit**.
3. In **Macro Title**, type a short description such as Meas GCA or View GCA.
4. Click **Browse**, then navigate to C:/Program Files/Agilent/Network Analyzer/Applications/GCA/GCA.vbs
5. In Macro run string parameters:
   1. Type **M** for the Measure macro or **V** for View macro.
   2. Optional: Supply the following additional parameters in any order:
      - To run the program from a remote computer, specify the full computer name of the PNA.
      - Channel in which to create the measurement. If not specified, Measure is created in Ch30 and View is created in Ch31.
      - Example: Run string parameters for the Measure macro run from a remote computer in Channel 5.—-M MyPNA 5.
6. Click **OK**.

**How to run the Macros**

On a GCA channel:

1. Create a 2D sweep. Either Power per Freq or Freq per Power. Both macros always create a power sweep at the frequency of interest.

2. Create a CompIn trace.

3. On the CompIn trace, right-click and select **Add Marker**. Drag the marker to the frequency of interest.

4. Press **Macro**, then select either by the short description you provided in Step 3.

---

Last Modified:

- **27-Apr-2011**  Removed Copy Channels limitation
- **1-Apr-2011**   Removed acquisition limitations
- **17-Mar-2010**  Added Max power and pulse support - removed some fixturing limitations
- **19-Nov-2009**  Added X-axis point spacing (9.1)
- **13-Aug-2009**  Added Compression Analysis
- **30-Mar-2009**  Added ADC measurements
- **23-Feb-2009**  Added link to compression marker
- **1-Dec-2008**   Clarified SMART - Linear compression
- **9-May-2008**   Edited for data save, backoff, and XY method
- **23-Aug-2007**  New topic
Gain Compression for Converters (GCX)

Gain Compression is measured on Converters in the same manner as it is measured in Amplifiers. Also, the Mixer/Converter setup is very similar to that of SMC (Scalar Mixer Converter application).

In this topic (unique for GCX):

- **Requirements and Limitations**
- **Using GCX**
- **Create a GCX Measurement**
- **Valid Mixer Configuration / Sweep Type Combinations**
- **Measurement Parameters offered in GCX**
- **GCX Calibration**

The following **Gain Compression for Amplifiers** information is relevant for learning about GCX:

- **Gain Compression Concepts**
- **Understanding the GCA Displayed Traces**
- **Compression Methods**
- **Acquisition Modes**
- **Compression Analysis**
- **Saving GCA Data**
- **GCA Measurement Tips**

The following dialog setup tabs are shared with other applications:

- **Frequency tab** (GCA topic)
- **Power tab** (GCA topic)
- **Compression tab** (GCA topic)
  - **Safe Sweep Mode dialog** (GCA topic)
- **Mixer Frequency tab** (separate topic)
- **Mixer Setup tab** (separate topic)
- **Mixer (LO) Power tab** (separate topic)
Requirements and Limitations
GCX requires Option 086 (Gain Compression) and FCA (either Option 082 or 083).

Limitations:

- Number of points limited to 20,001 for two-dimensional acquisitions, 10,000 points for SMART Sweep.
- Linear and CW sweep ONLY - No Power, Log, or Segment sweep. Learn more.
- GCX does NOT provide any built-in image rejection techniques. You should provide image rejection hardware if necessary.
- Stepped sweep mode only.

The following PNA features are NOT available with Gain Compression on Converters:

- ECal User Characterization
- Time Domain
- Balanced measurements
- Save Formatted Citifile data.
- Save SnP data.
- Interface Control
- Port extensions
- Some Fixturing Features
- External Test Set Control (Option 551)
- Integrated Narrowband or Narrowband Pulse App

Using GCX
The following is a general procedure for performing a GCX measurement. The challenge with GCX is configuring a measurement that yields the true compression performance of YOUR DUT. This requires knowledge of the Gain Compression and Mixer settings, and knowledge of the DUT.

See specific dialog boxes below for details.

1. Disconnect the DUT if preset or default power levels may damage the PNA or DUT.
2. Preset the PNA, or configure a suitable User Preset that will be safe in case the DUT is connected.
3. Create a GCX channel. Learn how. The default trace is SC21.

4. Start the GCX Setup dialog and configure the measurement settings based on the DUT, adapters, attenuators, booster amplifiers, and fixtures to be used in the measurement. To start the dialog, click Stimulus, then Frequency, then GCX Setup. Learn about the setup dialogs.

5. Save the instrument state (optional).

6. Connect the DUT. Inspect the measurement to ensure the DUT is operating as expected.

7. Add GCX compression and mixer parameter traces. Learn more.

8. Adjust the measurement settings to yield satisfactory compression results. See GCA Measurement Tips.

9. Start and complete the GCX Calibration wizard.

## Create a GCX Measurement

1. On the PNA front panel, press Meas then [Measurement Class]

2. Select Gain Compression Converters, then either:
   - OK delete the existing measurement, or
   - New Channel to create the measurement in a new channel.

3. A default SC21 measurement is displayed. To select additional parameters to display, click Trace/Chan, then New Trace, then select a parameter from the list. Learn more about GCX Parameters.

### How to start the Gain Compression for Converters Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press FREQ</td>
<td>1. Click Stimulus</td>
</tr>
<tr>
<td>2. then [Gain Compression Setup]</td>
<td>2. then Frequency</td>
</tr>
<tr>
<td></td>
<td>3. then GCX Setup</td>
</tr>
</tbody>
</table>

### Valid Mixer Configuration / Sweep Type Combinations

The following are the Valid Sweep Type / Mixer Configurations:
### Sweep Type

<table>
<thead>
<tr>
<th>Sweep Type</th>
<th>Input</th>
<th>LO</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>Swept</td>
<td>Fixed</td>
<td>Swept</td>
</tr>
<tr>
<td></td>
<td>Swept</td>
<td>Swept</td>
<td>Fixed</td>
</tr>
<tr>
<td>CW</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
</tbody>
</table>

### GCX Measurement Parameters

**Note:** The following table assumes: DUT **Input** = PNA port 1 and DUT **Output** = PNA port 2.

When the Port mapping is different, the parameters in GCX are updated accordingly. For example, with Input = port 2 and Output = port 1, then "CompIn12" would be displayed.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mixer Parameters</strong></td>
<td></td>
</tr>
<tr>
<td>SC21</td>
<td>Linear Conversion Gain</td>
</tr>
<tr>
<td>SC12</td>
<td>Reverse Conversion Gain</td>
</tr>
<tr>
<td>S11</td>
<td>Input Match</td>
</tr>
<tr>
<td>S22</td>
<td>Output Match</td>
</tr>
<tr>
<td><strong>Compression Parameters</strong></td>
<td></td>
</tr>
<tr>
<td>CompIn21</td>
<td>Input power at the compression point.</td>
</tr>
<tr>
<td>CompOut21</td>
<td>Output power at the compression point.</td>
</tr>
<tr>
<td>CompGain21</td>
<td>Gain at the compression point.</td>
</tr>
<tr>
<td>CompS11</td>
<td>Input Match at the compression point.</td>
</tr>
<tr>
<td>RefS21</td>
<td>Linear Gain value used to calculate the compression level. This is calculated differently depending on the compression method.</td>
</tr>
<tr>
<td>DeltaGain21</td>
<td>CompGain21 MINUS Linear Gain (in Log Mag format). This trace can be used to learn a lot about the DUT compression point. Learn more.</td>
</tr>
</tbody>
</table>

**Unratioed** - Absolute test port receiver measurements. Learn more.

3353
<table>
<thead>
<tr>
<th>IPwr</th>
<th>Input power measured at DUT-IN @ Input frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>OPwr</td>
<td>Output power measured at DUT-OUT @ Output frequency</td>
</tr>
<tr>
<td>RevIPwr</td>
<td>Input power measured at DUT-OUT @ Output frequency</td>
</tr>
<tr>
<td>RevOPwr</td>
<td>Output power measured at DUT-IN @ Input frequency</td>
</tr>
</tbody>
</table>

**ADC Parameters - Learn more.**

<table>
<thead>
<tr>
<th>AI1</th>
<th>Measured at the specified Linear Input level.</th>
</tr>
</thead>
<tbody>
<tr>
<td>AI2</td>
<td>Measured at the specified Linear Input level.</td>
</tr>
<tr>
<td>CompAI1</td>
<td>AI1 at Compression</td>
</tr>
<tr>
<td>CompAI2</td>
<td>AI2 at Compression</td>
</tr>
</tbody>
</table>

**GCX Setup Dialogs**

All of the GCX Setup tabs are shared with other applications.

- **Frequency tab**
- **Power tab**
- **Compression tab**
  - Safe Sweep Mode dialog
- **Mixer Frequency tab**
- **Mixer Setup tab**
- **Mixer (LO) Power tab**

**GCX Calibration**

A GCX Cal is conceptually the same as a Gain Compression Calibration. This includes the ability to perform or downgrade to an Enhanced Response Cal. Learn how.

The following Guided Cal Wizard pages are unique to GCX:
**GCX Calibration Setup** dialog box help

**Waveguide/In-fixture/On-Wafer Setup**  Starts the following dialog box.

**Independent power cals for input and output ports (no thru)**  Check if a Thru standard is NOT available. During the power cal, you will be prompted to connect the power sensor to the Input, then the Output port.

**Additional Power Cal Steps**

**Enable LO1 / LO2 Power Cal**  Check when LO1 / LO2 is controlled (on the **Mixer Setup** tab) to perform a Power Cal on the source.

**Note:** Phase Correction is NOT allowed for GCX measurements.

---

**Waveguide/In-fixture/On-Wafer Setup** dialog box help

This dialog box appears ONLY if you checked the **Waveguide/In-fixture/On-Wafer Setup** box in the previous **Cal Setup** dialog.

Allows you to embed or de-embed circuit networks on the input and output of your mixer under test.

For Network1 (Input) and Network2 (Output) select **Embed**, **De-embed**, or **None**.

**Browse**  Click to navigate to the .S2P file that models the network to embed or de-embed.

**To Embed or De-embed**

- **De-Embed**  when there is a 2-port network that needs to be connected during the measurement, but it is NOT present during the calibration. An example might be when you do not have standards for a characterized test fixture, so you calibrate without the fixture, and make measurements with the DUT in the fixture. De-Embedding during the calibration extends the calibration reference plane to include the 2-port network.

- **Embed**  when there is a 2-port network that to be disconnected during the measurement, but is present during the calibration. An example might be when a characterized adapter is required during the calibration but NOT present during measurements of the DUT. Embedding during the calibration retracts
the calibration reference plane to exclude the 2-port network during the measurement.

**Notes**

- **Characterize Adaptor Macro** can be used to create the S2P file.
- The S2P file for Network1 (on the input of the mixer), must cover the Input frequency range. The S2P file for Network2 (on the output of the mixer), must cover the Output frequency range.
- The frequency range of the S2P file must be the same, or larger than, the frequency range of the measurement. If more frequencies are included in the file, and the data points do not exactly match those of the measurement, interpolation will be performed.
- As in the image on the dialog (above), in all cases:
  - Port 1 of each network is assumed to be connected to the PNA.
  - Port 2 of each network is assumed to be connected to the DUT.

---

**Select DUT Connectors and Cal Kits dialog box help**

Allows you to specify the connector type and Cal Kit for each DUT port.

**Port n**

For each listed PNA port, specify the DUT connector type and gender, and the Cal Kit to use.

**input pwr sensor**

Specify the connector type of the power sensor. Select **Ignored** to not compensate for the effects of the adapter that may be necessary to connect the power sensor to the input reference plane.

**output pwr sensor**

Available when **Independent power cals for input and output ports** is checked on the GCX Calibration Setup dialog. Specify the connector type of the power sensor. Select **Ignored** to NOT compensate for the effects of an adapter that may be necessary to connect the power sensor to the output reference plane.

**De-embed input power sensor adapter**

Check to measure, then remove the effects of the adapter that is used to connect the power sensor to the calibration reference plane.

**Source Cal Settings**

Click to start the **Source Cal Settings** dialog. These settings allow you change ALL Source Cal and Power Meter settings.

**Note:** If your DUT connectors are:

- **Waveguide** Change the system impedance to 1 ohm before performing a calibration. See **Setting System Impedance**.
- **Not listed** (male and female) Select **Type A** as the connector type. Type A requires a calibration kit file containing the electrical properties of the standards used for calibration (see [Calibration kits](#)).

- **Unspecified** (like a packaged device) Select **Type B** as the connector type. Type B requires a calibration kit file containing the electrical properties of the standards used for calibration (see [Calibration kits](#)).

**Modify Cal** Check, then click Next, to start the Modify Frequency Cal dialog.

---

**Last Modified:**

- 27-Apr-2011 Removed Copy Channels limitation
- 9-Aug-2010 New topic
Gain Compression Calibration

The GCA Calibration Wizard guides you through a calibration of GCA or GCX channel. The procedure is the same regardless of the Gain Compression Settings.

- A Source Power Calibration and Receiver Calibration is performed first.
- Then, your choice of a Full 2-port Cal or an Enhanced Response Cal.

See Also
Gain Compression Application
Gain Compression for Converters
Calibration Programming commands

How to start a GCA Calibration

Using front-panel HARDKEY [softkey] buttons | Using a mouse with PNA Menus
---|---
1. Press CAL | 1. Click Response
2. then [Start Cal] | 2. then Cal Wizard
3. then [Cal Wizard]

Overview - GCA Source Power Cal

The GCA Source Power Calibration is a little different from a standard Source Power Cal. Although GCA measurements are performed at many power levels, the GCA source power cal is performed at a single power level over the specified frequency span of your GCA measurement. The required source correction from that single power level is applied to all power levels. This method ensures that the 'absolute' power level being applied to the DUT is within the PNA-X source power linearity specification.

Although it is important for GCA to be able to set the absolute power level to the DUT, it is MOST important to be able to exactly measure the actual incident power. Therefore, during the GCA Source Power Cal, a receiver calibration is applied to the port 1 reference receiver, and indirectly to both test port receivers during the S-parameter calibration, correcting for impedance mismatch between the power meter and the PNA source, and the DUT and the PNA source.

Although the cal process is also at a single power level, the dynamic accuracy of the PNA-X receivers is typically about +/- .05 dB, which is comparable to the accuracy of Agilent's best power sensors. This allows GCA to very accurately measure and report ALL power levels that are actually applied to the DUT.

Full 2-port or Enhanced Response (ER) Cal
By default, a full 2-port calibration is performed as part of a GCA and GCX calibration. However, you can change to an Enhanced Response Cal. The following issues may help you decide between these two Cal types:

- **Accuracy**  A full 2-port correction is more accurate than ER when GCA measures linear gain. However, for non-linear measurements, ER yields identical compression values as a full 2-port cal, so this may not be a significant factor.

- **Measurement speed**  An ER correction only requires measurements in the forward direction. The reverse parameters (usually S22 and S12) are not measured unless requested. With a full 2-port cal applied, all four S-parameters are measured, which requires an additional reverse sweep. Learn more.

- **Ease**  A full 2-port cal is easiest with an ECal module. An ER Cal requires a Defined Thru or a Flush Thru Cal method. If these are possible, then an ER cal is easiest when using a mechanical Cal Kit.

- **High power**  The test port damage level of a standard PNA-X is +30 dBm. Therefore, external attenuation may be required on the output of high power amplifiers, which degrades calibration accuracy for reverse (full 2-port) measurements. In addition, the external attenuation improves the DUT output / load match error, which allows a better uncorrected response and makes an Enhanced Response Cal the better choice.

- **DUT limitations**  With an ER Cal applied, reverse measurements on the DUT are not performed unless requested.

**How to select Enhanced Response Cal**

At the Select DUT Connectors page of the GCA Cal Wizard:

1. Check **Modify Cal**, then click **Next**.
2. A **Defined Thru** or a **Flush Thru Cal** method must be selected.
3. Click **Cal Type/Stds**
4. Under Calibration type, select **EnhResp** (2 <= 1 refers to the receive port 2 and source port 1).

**Downgrade a Full 2-port Cal to Enhanced Response Cal**  if you prefer to perform a Full 2-port cal, but not perform reverse sweeps on the DUT.

To change the correction on the channel from Full 2-port to Enhanced Response:

1. Press **CAL**
2. then **[Manage Cal]**
3. then **[Correction Methods]**
4. Select **GCA EnhResp**, then **OK**
GCA Cal Wizard

Select DUT Connectors and Cal Kits - GCA Cal dialog box help

This is a standard Cal Wizard page except for the following:

**Power Sensor** Specify the connector type and gender of the power sensor. When the power sensor connector is not the same type and gender as the DUT Port 1 connector, then for optimum accuracy, extra cal steps are required to measure and correct for the adapter that is used to connect the power sensor to the port 1 reference plane during the Source Power Cal. Select the Cal Kit that will be used for that process.

Select **Ignored** (at the bottom of the DUT Connectors list) to NOT compensate for the adapter.

**Modify Cal** Check, then click **Next**, to Modify Cal (Standards AND Thru Method).

**Source Cal Settings** Click to launch the Source Cal Settings dialog.

Learn more about GCA Source Power Calibration
Gain Compression Calibration Step 1 dialog box help

**Power Level** at which to perform the Source Power Cal.

It is usually best to perform the Source Power Cal at 0 dBm because the power sensor is calibrated at that level.

However, if the Gain Compression measurement is performed entirely below or above 0 dBm, then perform the Source Power Cal at the **Stop** power which probably has the lowest level of measurement noise.

Learn more about GCA Source Power Calibration

The remaining Gain Compression Cal dialogs are the same as the standard SmartCal dialogs.

Return to Gain Compression Application.

Last Modified:

- 27-Sep-2010  Updated for GCX
- 26-Nov-2007  MX New topic
iTMSA (Integrated True Mode Stimulus Application)

This application is integrated into the PNA firmware as Option 460. A previous version of TMSA is run as a Macro on the PNA.

- **Features, Requirements, and Limitations**
- **True Mode Stimulus Concepts**
- **Using iTMSA**
  - Create a Measurement
  - Set Power Level
  - Calibrate
  - Ratioed Receiver Measurements

---

**Other PNA Applications**

**Notes:** When the first iTMSA measurement is created, an IFMUX Cal is performed, which takes a couple of minutes. If an IF board is replaced, this Cal should be performed again. To make this happen, delete C:/Program Files/Agilent/Network Analyzer/IFMUX.txt. When the next iTMSA measurement is created, the file will be recreated with new data.

**Features, Requirements, and Limitations**

**Features**

- A seamless extension of existing PNA Balanced Measurement, but with True Mode stimulus.
- True Mode Stimulus measurements are performed in a standard S-parameter channel.

**Requirements**

- 4-port PNA-X (opt 400) with Opt 460 (software option only): must be enabled.

**Limitations with iTMSA**

The following standard PNA features are NOT available with iTMSA measurements:

- External Test Set Control (Option 551)
- Interface Control
- Time Domain Pulse measurements in [Wideband Pulse](#) are NOT supported.
- [Unratioed reference receiver](#) measurements.
- [Segment Sweep](#)
- [Frequency Offset Measurements](#) (opt 080)
- [Receiver Calibration](#)

**True Mode Stimulus Concepts**

A balanced device is designed to receive input simultaneously across two ports. Standard PNA Balanced measurements apply stimulus to one port at a time, measures the responses, and calculates the theoretical balanced responses. [Learn more about balanced measurements](#).

True Mode Stimulus uses two PNA sources to apply either truly differential (180 degree out-of-phase) or truly common (in-phase) signals across the input of a balanced device. PNA receivers measure the single-ended response at the output of the device and calculate the balanced response.

When operating in non-linear regions, a device may respond differently to single-ended stimulus than to True Mode Stimulus. Thus, True Mode Stimulus capability allows you to understand when, and if, True Mode Stimulus is required.

For more detailed information on this measurement technique, see the following white papers (internet connection required):

- [New Methods & Non-Linear Measurements for Active Differential Devices](#)
- [New Measurement Results and Models for Non-linear Differential Amplifier Characterization](#)

**How iTMSA Works - Overview**

The following is an overview of how iTMSA gathers and displays True Mode Stimulus data:

1. **Initial sweep** is performed to gather the initial phase deviation between the two sources at the reference plane.

2. **Measurement** The phase deviation between the two sources is set to the required value and the response is measured. For a differential sweep, the phase deviation of the two sources is set to 180° at the reference plane. For a common sweep, the phase deviation is set to 0° at the reference plane.

A forward direction measurement requires 2 sweeps for differential and 2 sweeps for common. A reverse direction measurement requires the same. Therefore, a complete measurement in both directions requires 8 sweeps. The iTMSA traces are updated when all of the necessary data is gathered in each direction.

iTMSA computes the raw S parameters with the following matrix:
Using iTMSA

A Standard (default) S-Parameter channel is used for iTMSA measurements.

Legend:

- d1: differential stimulus in balanced port 1.
- d2: differential stimulus in balanced port 2
- c1: common stimulus in balanced port 1.
- c2: common stimulus in balanced port 2.

**Using iTMSA**

How to create an iTMSA trace:

1. Preset the PNA
2. Then do the following:

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>Meas</strong></td>
<td>1. <strong>click Response</strong></td>
</tr>
<tr>
<td>2. then <strong>[Balanced Parameters]</strong></td>
<td>2. then <strong>Measure</strong></td>
</tr>
<tr>
<td>3. then press the <strong>Change</strong> button</td>
<td>3. then <strong>Balanced Parameters</strong></td>
</tr>
<tr>
<td></td>
<td>4. then press the <strong>Change</strong> button</td>
</tr>
</tbody>
</table>
Balanced DUT Topology / Logical Port Mapping (with iTMSA Option 460)

Learn about iTMSA.

Topology

Only the following topologies and port mappings are available in True Mode Stimulus:

- **BAL-BAL** DUT has two balanced ports. The PNA ports are restricted to the following:
  - Balanced Port 1 = PNA ports 1 and 3
  - Balanced Port 2 = PNA ports 2 and 4

- **SE-BAL** DUT has one single-ended port and one balanced port. For a **BAL-SE** (balanced - single-ended) DUT, select this topology, and then select **Reverse True Mode**. The PNA ports are restricted to the following:
  - Single-ended Port 1 = PNA port 1.
  - Balanced Port 2 = PNA ports 2 and 4.

Stimulus

- **Single Ended** Standard PNA mode. Each DUT port receives stimulus individually.
- **True Mode** Each balanced port receives true balanced stimulus in both forward and reverse directions.
- **Forward True Mode** Available ONLY in BAL-BAL topology. Only Balanced port 1 receives true balanced stimulus.
stimulus.

**Balanced Port Offset**

**Select Port**  Select the physical PNA port to receive a phase or power offset. Only port 3 and port 4 are available for Phase or Power offset.

**Phase Offset**  Specify offset within +/- 180° for the balanced INPUT port. This is in addition to the standard offsets, which are 180° offset for the differential stimulus sweeps, and 0° for the common stimulus sweeps.

**Offset (phase) as Fixture**

When unchecked, output calculations are performed and displayed as though there is only the standard offset. Although additional Phase Offset is applied to the stimulus, it is ignored in the calculations of balanced differential and common mode output signals.

When checked, output calculations are performed and displayed using the actual phase offset that is applied to the DUT. Use this setting to compensate for a component or fixture that may present a phase delay before the DUT.

**Power Offset**  Specify power offset. Range is +/- 2 dB. This is in addition to the power level that is specified using the Power and Attenuators dialog. Offset Power is NOT reflected on the power dialog nor on the X-Axis during a power sweep.

**Offset (power) as Fixture**

When unchecked, output calculations are performed and displayed as though there is no stimulus power offset.

When checked, output calculations are performed and displayed using the power offset that is applied to the DUT. Use this setting to compensate for a component or fixture that may present a magnitude loss before the DUT.

**Phase Sweep**

- Available only when **Sweep type** = CW.
- Only PNA port 3 OR port 4 are eligible for Phase sweep.
- Available ONLY for balanced pairs.

The phase of the selected port (3 or 4) is swept relative to the phase of the other port (1 or 2) in the balanced pair.

Phase Sweep is similar to phase offset, except that for each data point, the phase 'offset' is incremented.

For example, with the topology that is selected in the above image: (Bal-Bal) Logical port 1 = PNA ports 1 and 3. For a phase sweep with 7 data points, from 0° to 180°, the phase difference between port 1 and port 3 increments 30° with each data point:

<table>
<thead>
<tr>
<th>Data point</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delta phase</td>
<td>0°</td>
<td>30°</td>
<td>60°</td>
<td>90°</td>
<td>120°</td>
<td>150°</td>
<td>180°</td>
</tr>
</tbody>
</table>

**Enable Phase Sweep**  Check to enable phase sweep for the specified port.
**Sweep Phase** Choose EITHER PNA Port 3 OR Port 4 to sweep. Only ONE port can be phase swept.

**Start / Stop Phase** Enter phase values in degrees. Each sweep will start and stop at these settings.

**Offset (phase) as Fixture**

In the following image, the PNA phase sweep is shown as a phase shifter that is "virtually" located EITHER before the reference receiver (top) OR after the reference receiver (bottom) - NOT BOTH. This image is useful as a model ONLY.

**NOT checked** - The phase shift occurs at each data point and the reference receiver measurement sees the same phase shift as the DUT.

The response is measured as though a true phase sweep is stimulating the differential input to the DUT.

**Checked** - The phase shift occurs at each data point, but the reference receiver measurement does NOT see the shift.

Use this setting to compensate for a component or fixture that may present a phase shift before or after the DUT.

---

**One use for Phase Sweep**

Do the following to find best the best operating point:

1. Phase sweep port 3, from 0° to 180°. Note the peak of Sdd21. This finds the phase offset of the input differential port.

2. Set port 3 **fixed offset** to the phase that has the highest Sdd21.

3. Then, phase sweep port 4 (balanced port 2) to find the phase offset of the output differential port.

4. Then set the port 4 to fixed phase offset found.

This measures the best possible case Sdd21 for the DUT.

---

**Power and Attenuators Settings**

Set stimulus power levels using the standard PNA Power and Attenuators dialog.

To set power, press **STIMULUS**, then [Power and Attenuators]
Power and Attenuation (with True Mode Stimulus) dialog box help

When a True Mode Stimulus is selected in the Balanced DUT Topology dialog, the balanced ports are shown on the top rows of the dialog. The individual PNA port settings are displayed but can not be changed directly.

**Port Powers Coupled** Check to couple all power settings for Balanced Port 1 and Balanced Port 2. Clear this box to make independent power settings for these logical ports.

**State** Should be left in Auto for iTMSA measurements. [Learn more about his setting.](#)

**Port Power** Set power for the balanced port. Power for PNA ports, shown below balanced ports, are set to 3 dB less, or half power. [Power Offset](#) is made in addition to this setting, and on only PNA port 3 and port 4. Offset Power is NOT reflected on this dialog nor on the X-Axis.

**Start and Stop Power** Available when sweep type is set to power sweep.

**Auto Range** Check to allow the PNA to select the optimum attenuation value to achieve the specified test port power. Clear to manually set the attenuation for each port. Type or select the attenuation value in the adjacent Attenuator Control box. When the attenuator setting of a logical port is changed, then the attenuators of the PNA ports associated with that logical port change to the same value.

**Leveling Mode** Open Loop leveling is available only on ports 1 and 3. [Learn more.](#)

**Note:** The range of leveled power (ALC range) for balanced ports is 3 dB higher than it is for each individual PNA port. For example, if a PNA source port with 0 dB attenuation will supply leveled power from -30dBm to +15dBm, then the balanced logical port has a range of -27dBm to +18dBm. [Learn more about Leveled Power.](#)
- **Port 4** for BAL Port 2

Port 1 and Port 2 power levels are adjusted to these ports in the initial sweep.

**Receiver Measurements**

Any pair of receivers can be viewed as a ratio using the following dialog.

Unratioed measurements using reference receivers (an ‘R’ receiver as numerator and 1 as denominator) are NOT accurate.

To select these measurements, press **Meas**, then **[Receivers]**

---

Create / Change Receiver Measurements (with True Mode Stimulus) dialog box help

Click **Activate**

- For RATIOED measurements, Select a receiver for the Numerator and select a receiver for the Denominator.

- For UNRATIOED measurements, Select a test port receiver (A, B, C, or D) for the Numerator. Reference receiver measurements are NOT accurate. Select 1 for the Denominator.

For example, with a Bal-Bal topology, the above selections show a R3/R1 measurement. R (reference) receivers measure the stimulus to the DUT. An R3/ R1 trace shows the difference between the two sources that comprise logical port 1. With Log format, a power offset between the two sources is visible. With Phase format, a phase offset is visible.

**Source Port** Specifies whether a Differential stimulus or Common mode stimulus is used for the measurement.

- With Source Port 1 or 2 selected, then **Differential** stimulus is used for the unratioed measurement which causes **180° offset** between the sources.
  
  - Source Port 1 = Differential stimulus on BAL1 port
  
  - Source Port 2 = Differential stimulus on BAL2 port

- With Source Port 3 or 4 selected, then **Common** mode stimulus is used for the unratioed measurement which causes **0° offset** between the sources.
  
  - Source Port 3 = Common-mode stimulus on BAL1 port
  
  - Source Port 4 = Common-mode stimulus on BAL2 port

---

Last Modified:
13-Nov-2008  Fixed allowed SE-BAL topology
19-May-2008  MX New topic
Noise Figure Application (Opt 028 and Opt 029)

The Noise Figure Application makes fast, easy, and accurate noise figure measurements. This topic discusses Noise Figure on Amplifiers.

See Also: Noise Figure on Converters (NFX)

- Noise Figure Options Explained (029, 028, H29)
- Features, Requirements, and Limitations
- Noise Concepts
- How the Noise Figure Application Works
- Scalar Noise Figure Measurements
- Noise Power Parameters
- Using Noise Figure App
  - Connect Tuner and Noise Source.
  - Create a Noise Figure Measurement.
  - Make Noise Figure Settings.
  - Perform Calibration
- Noise Figure Measurement Tips
- Using Noise Figure Traces in Equation Editor
- Noise Model and the Noise Correlation Matrix

See Also

- Noise Figure Calibration
- Programming commands
- PNA-X Noise Figure Options (PNA Configuration Guide)
- High-Accuracy Noise Figure Measurements Using the PNA-X
- Scalar Noise and TRL Cal

See other PNA Applications

Noise Figure Options Explained (029, 028, H29)

See Also: PNA-X Noise Figure Options (PNA Configuration Guide) - Internet connection required
• **029** - (PNA-X ONLY) Includes two low-noise receivers and noise tuner bypass switch to enable noise figure measurements to 26.5 GHz. Also has Opt 028 capability.

• **028** - Uses standard PNA receivers to measure noise figure. A noise source is NOT required during calibration. Any two ports can be used. Use with DUTs that have sufficiently high gain and noise figure. Additional filtering may be necessary. Learn more.

• **H29** - Noise Figure on N5244A (43.5 GHz PNA-X model) and N5245A (50 GHz PNA-X model). Includes both:
  
  - Opt 029 (noise measurements to 26.5 GHz)
  

**Noise Figure Application Features**

• Cold noise method includes correction for imperfect system source match, for highly accurate noise figure measurements.

• Operates from 10 MHz to 50 GHz, with choice of low-noise receivers to 26.5 GHz or standard receivers to 50 GHz. (Opt 028)

• Opt 029 measures devices with Noise Figure values ranging from about 0 to 50 dB and devices with GAIN ranging from about -40 to +60 dB. Learn more.

• Measure noise figure of frequency translating devices. Learn more.

• During calibration, ENR values are interpolated for frequencies between the supplied data points.

**Noise Figure Application Requirements**

• Noise Tuner (ECal Module - N4691B m-f recommended). Not required for Scalar Noise Figure. Opt 029 provides an additional cable and adapter to connect the ECal module to the front-panel connectors. Learn more.

• Power Meter - Required when calibrating NFX (Noise Figure on Converters).

• Recommended: An accurate thermometer. Learn more.
**Noise Source**

Beginning with PNA Rev. A.09.41 a Noise Source is NOT required to calibrate the Opt 029 Noise Receivers. Instead, the Noise Receivers can be calibrated using a calibrated PNA source. [Learn more](#).

When using a Noise Source, the following requirements apply:

- **Agilent 346C Noise Source (recommended):** Covers the same frequency range as the PNA. NOT required with Opt 028. The 346B Noise Source can be used up to 18 GHz. The 346A can also be used, but requires more [averaging](#) for calibration.

- **An adapter may be necessary to connect the Noise Source to the PNA port 2 reference plane during calibration.** Cal Kit (or second ECal module) with same connector type and gender as DUT connectors.

**Limitations with the Noise Figure Application**

The following features are NOT supported in a Noise Figure channel:

- **FCA (opt 083) or Frequency Offset (opt 080).**

- **Analog sweep.** All frequency sweeps are STEPPED.

- **External Test Set Control** (Opt 550 or 551)

- **Receiver calibration.**

- **Enhanced Response Cal**

- **ECal User Characterization.**

- **Some Fixturing Features**

- **Auto Formatted Citifile** data.

- **Pulsed** Noise Figure measurements are supported with the following limitations:

  - Minimum 300 microsecond pulse width using 24 MHz noise bandwidth

  - Narrower noise bandwidths cause larger minimum pulse widths

  - A drop-out may occur at start of sweep and at 3 GHz. This is corrected by a 1 ms pulse width at 24 MHz Noise BW.

**Noise Concepts**

The following conceptual information is a short summary taken from the [Agilent Noise Figure App Note 57-1](#).

All electronic circuits have some degree of random noise. The most common form is thermal noise, which increases as the temperature of the circuit increases.
The signal-to-noise (S/N) ratio of components in a communications system is a very important parameter. To improve the S/N ratio, it is usually easier and more cost-effective to reduce noise than to increase signal power. In order to reduce noise, an accurate method to measure noise is required.

**Noise Figure**

Noise figure is the degradation in the signal-to-noise ratio as a signal passes through a device. For example, in the following images:

(a) At the INPUT of an amplifier:
The noise floor is -100 dBm, the signal is at -60 dBm, **40 dB** above the noise floor.

(b) At the OUTPUT of the same amplifier:
The gain has boosted the signal AND the noise floor by 20 dB.
The amplifier then added 10 dB of its own noise.
The output signal is now only **30 dB** above the noise floor.
Since the degradation in signal-to-noise ratio is 10 dB, the amplifier has a **10 dB noise figure**.

For consistency, noise measurements are calculated as if using a 1 Hz bandwidth, although measurements are almost always made at higher bandwidths.

The following formula shows the lowest possible noise power in dBm at 290° K (room temperature). The only way to measure noise lower than this is to make the measurement at a lower temperature.

- \( P = 10 \log(4.0 \times 10^{-21} \text{ watts} / 0.001 \text{ watt}) \)
- \( P = -174 \text{ dBm} / \text{ Hz} \)

**How the Noise Figure Application Works**

The goal of the Noise Figure application is to accurately measure the noise that is generated by the DUT. This may be done using special low-noise receivers or using the standard PNA receivers depending on whether the PNA has Options 029 or 028. Learn more. The low-noise receivers are calibrated using a characterized noise source. The standard receivers are calibrated using a power meter and a measurement of the receivers effective
noise bandwidth. Learn more about the noise calibration process.

Some noise measurement error is caused by a poor source match presented to the DUT input. Therefore, during every measurement, the Noise Figure Application uses an ECal module to present at least four different impedances at the input of the DUT. This “Noise Tuner” is connected to the PNA port 1 front-panel loops that are in the PNA internal source path (see block diagram below). From the measurements at various impedance states, the PNA calculates the noise out of the DUT as though the PNA were exactly 50 ohms. No assumptions are made regarding the input impedance of the DUT.

Here is how a Vector Noise Figure measurement is made using Option 029. The sweep numbers are annotated on the PNA display as they occur.

1. With the noise tuner in the THRU state, S-parameter measurements are made to accurately characterize the gain of the DUT. This requires sweeps in both forward and reverse directions. (sweep #1 and #2).

2. The noise measurements are performed next. PNA source power is turned OFF and the noise tuner is switched to the first impedance state.

3. At each frequency, the noise receiver samples a large number of readings in order to attain one valid measurement. If Noise Averaging is selected, the specified number of measurements are made and averaged together to obtain one noise measurement. This continues for all frequencies (sweep #3).

4. The next noise tuner impedance state is switched IN and the noise measurements in step 3 are repeated. This occurs until measurements are made at all impedance states. At least four impedance states must be used. (sweeps #4, #5, #6+)

5. Calibration error terms are applied and calculations made to simulate the measurement with a perfect 50 ohm input impedance. The sweep result is plotted on the PNA display.

6. The PNA begins sweeping again with step 1.

Scalar Noise Figure Measurements

Scalar Noise Figure measurements can be made beginning with PNA Rev. A.08.50.

As described above, the noise tuner is switched to at least four different impedance states before a sweep is plotted. These sweeps are NOT made in a Scalar Noise Figure measurement, resulting in much faster measurements. Of course, a Scalar Noise Figure measurement is NOT as accurate as a Vector Noise Figure measurement because Scalar Noise Figure measurements assumes that all impedances are 50 ohms. Measurement accuracy can be improved by adding an attenuator as close to the DUT input as possible. This improves the effective system source match. The effect of the attenuator loss is removed during the calibration process.

With Scalar Noise Figure, it is not necessary to connect the noise tuner. If a noise tuner remains connected, it is switched to the THRU state for Scalar Noise Figure measurements. This results is a small amount of loss which slightly degrades measurement accuracy. To increase measurement accuracy, manually switch the noise tuner switch to the INTERNAL position. Learn how.

Select Scalar Noise Figure at the first page of a Noise Figure calibration.

PNA-X Option 029 - Block Diagram with Noise Figure components
Noise Figure Components are shaded yellow

- At test port 1 front-panel loops, a **noise tuner bypass switch** connects the noise tuner (ECal module) in series with Source1 providing several different input impedances.

- At test port 2, a **switch** and **coupler** to route RF from the DUT output to **two noise receivers**. The appropriate receiver is automatically switched as required for frequency being measured.

**Making S-parameter measurements and the Noise Tuner Switch**

The default setting for the port 1 noise tuner bypass switch is EXTERNAL, as shown in the above diagram. This setting always provides incident power through the front panel loop. When an ECal module is connected, it may NOT be in the THRU state, which is necessary for accurate S-parameter measurements. This can be changed in any of the following ways:

- Set the switch to INTERNAL for the S-parameter channel using the **path configuration** dialog.

- Set the switch to INTERNAL for the S-parameter channel using the following commands:
  - **SCPI** - `SENS:PATH:CONF:ELEM:STAT Port1NoiseTuner, Internal`
  - **COM** - `PathConfiguration.Element(Port1NoiseTuner).Value = Internal`

- Set the switch default to INTERNAL using a **preference setting**.
Set the Noise Tuner (ECal module) to the THRU state using SCPI: CONT:ECAL:MOD:PATH:STATE.

**Noise Figure App vs Noise Figure Analyzer**

In comparing the PNA Noise Figure App measurements with the NFA Series Noise Figure Analyzer measurements, you may obtain different results. This is because the Noise Figure Analyzer assumes that the noise source has a perfect 50 ohm match. This assumption is often valid for low-ENR sources, but is NOT a good assumption for high-ENR sources, or when there are cables, adapters, switches, probes, etc. between the noise source and the DUT. Using advanced calibration methods, the PNA Noise Figure App calculates noise figure as though it were a perfect 50 ohm match.

In addition, the PNA measures the amplifier gain with vector error correction applied to reduce measurement uncertainty.

**Noise Power Parameters**

Several noise power parameters, as well as standard parameters, can be measured in the same Noise channel.

**How to add Noise Power Parameters**

1. Create a Noise Figure channel.
2. Then do the following:

   **Using front-panel HARDKEY [softkey] buttons**
   1. Press TRACES
   2. then **[New Trace]**
   3. then select a parameter

   **Using a mouse with PNA Menus**
   1. Click Trace/Chan
   2. then **New Trace**
   3. then select a parameter

**Noise Figure Parameters that are offered:**

- **Noise Figure (NF)** - Explained in Noise concepts.
- **Excess Noise Ratio** - Select when measuring the noise source. Compare with the ENR table to validate accuracy of the system. ENR is calculated as:

  \[
  \text{ENR (in dB)} = 10 \log_{10}(\frac{T_{\text{hot}} - T_{\text{cold}}}{T_0}), \text{ where } T_0 = 290K.
  \]

  Learn more about the ENR table and Noise Source. Learn more about Noise Source ENR measurements.

- **T-Effective** - The effective temperature, in Kelvin, of the measured noise level. For example:
290° K = -174 dBm/Hz.

$T_e$ is the unknown variable

$$P_{out} = kBG_d(T_{cold} + T_e)$$

Available Gain $G_a$ is a function of $S_{11}$, $S_{22}$, and $\Gamma_s$

The Noise Power parameters below are offered in the following two formats:

- **Available Noise Power** The calculated power that is based on an ideal impedance match at the output of the DUT. These parameters have always been offered in the PNA Noise Figure App.

- **Incident Noise Power** - An 'I' is appended to the end of the Available Noise Power parameter. The calculated power into a perfect 50 ohm noise receiver, regardless of the output impedance of the DUT.

**Noise Power Parameters**

- **SYSNPD / SYSNPD** - System Noise Power Density: Total noise power available at the ADC, including the noise contributed by both the DUT and the internal noise receiver. This is generally expressed as an absolute power measurement in dBm, but can also be expressed in Watts or Kelvin.

  $\text{dBm} = 10 \log_{10}(k \times T \times B \times 1000)$

  where:

  $k =$ Boltzmann’s constant

  $T =$ the measured noise temperature

  $B =$ bandwidth

  $1000 =$ conversion from milliwatts

- **SYSRNP / SYSRNPI** - System Relative Noise Power: The noise temperature of the combined DUT and receiver relative to 290 Kelvin. This is generally reported as a ratio in dB. Therefore a perfectly quiet device would render a trace at 0 dB.

  $\text{dB} = 10 \log_{10}(T/290)$
• **DUTNPD / DUTNPDI** - DUT Noise Power Density: When correction is ON, this trace exhibits the available noise power, best described as the maximum power available from the DUT where the impedance of the noise port is equal to the output match of the DUT. To be more precise, this occurs when the noise port match is equal to the conjugate of the output match of the DUT. The noise power contributed by the receiver is removed.

When correction is OFF, the trace exhibits what is more accurately described as delivered power. Delivered power is the power actually seen by the ADC. Any mismatch between the receiver and the DUT is ignored. The noise power contributed by the receiver is removed.

This measurement is generally expressed in dBm:

\[ \text{dBm} = 10 \log_{10} \left( \frac{\text{DUT Temperature} - \text{Receiver Temperature}}{B \times 1000} \right) \]

where:

- \( B \) = bandwidth
- \( 1000 \) = conversion from milliwatts

• **DUTRNP / DUTRNP** - DUT Relative Noise Power: This measurement is rendered as a ratio of the DUT temperature to 290 Kelvin. It is generally expressed in dB. The same comments apply with respect to available versus delivered power as described above for DUTNPD.

\[ \text{dB} = 10 \log_{10} \left( \frac{\text{DUT Temperature} - \text{Receiver Temperature}}{290} \right) \]

**Standard Parameters that are offered**

• **S-parameters**: S11, S21, S22, S12

• **Unratioed parameters** using the following notation: (Receiver, source port). These parameters REPLACE the active measurement. To do this (from front-panel ONLY), press **Meas**, then **[More]**, then **[ Receivers]**.

  - (R1,1), (R2,2), (A,1), (A,2), (B,1), (B,2)

**Using the Noise Figure Application**

Use the following general procedure to make Noise Figure measurements:

1. **Connect Tuner and Noise Source**.
2. **Create a Noise Figure Measurement**.
3. **Make Noise Figure Settings**.
4. For Opt 029 and H29, copy your Noise Source ENR file to the PNA C:/Program Files/Agilent/Network Analyzer/Noise folder.
5. **Perform Calibration**
6. Connect the DUT. Learn more about DUT input and output ports.

7. Measure Noise Figure.

8. **Optional** Click File, then Save to save Noise Figure data in the following formats: (available ONLY when NF correction is ON.)

   - *.CTI  Citifile
   - *.PRN
   - *.nco  Noise Correlation Matrix data in S2P format. See Noise Model.

See Also: Measurement Tips

**Connect Noise Tuner and Noise Source**

1. Connect the **noise source** to the 28V connector on the PNA rear panel. **NOT required for Opt 028.** The Noise Source is turned ON and OFF automatically as needed during a calibration. Connect the noise source to Port 2 reference plane when prompted during calibration.

2. Connect the **noise tuner** (ECal module). **NOT required for Scalar Noise Figure** measurements.

   a. On the PNA front panel, remove the **Port 1** jumper cable SOURCE OUT / CPLR THRU. Opt 028 allows Noise Figure measurements using any two PNA ports. Connect the noise tuner to the front-panel jumpers for the source (DUT input) port.

   b. Connect M-F tuner (N4691B-M0F) using the supplied cable (N5242-20137) and adapter (85052-60013).

   c. When using F-F ECal module (N4691B-00F), order a 3.5 mm M-M adapter (85052-60014).

**Create a Noise Figure Measurement**

1. On the PNA front panel, press **Meas**, then [Measurement Class]

2. Select **Noise Figure Cold Source**, then either:

   - **OK** delete the existing measurement, or
New Channel to create the measurement in a new channel.

3. A Noise Figure measurement is displayed. To select additional parameters to display, click Response, then Measure, then select a parameter from the list.

How to start the Noise Figure Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

Using front-panel HARDKEY [softkey] buttons |
Using a mouse with PNA Menus

1. Press FREQ
2. then [Noise Figure Setup]
1. Click Response
2. then Measure
3. then Noise Setup

Noise Figure Setup dialog box help

Note: In this topic, the term Jitter is used to describe the trace-to-trace fluctuations in a measurement. In other topics, this is called 'trace noise'.

Bandwidth/Average

The following two settings work together to achieve the optimum balance of measurement accuracy versus speed:

**Noise Bandwidth** Increase the bandwidth to reduce the amount of trace noise on the noise power or noise figure measurement (jitter). However, a wider setting reduces the frequency resolution of the measurement. The noise bandwidth setting should always be smaller than the bandwidth of the DUT. The noise bandwidth setting is used only while measuring noise powers, and is independent from the IF bandwidth setting used to measure S-parameters. Noise figure is calculated from noise power and S-parameter measurements.

Note: The calibration and measurement should be performed using the SAME noise bandwidth. When the noise bandwidth is changed after calibration, noise figure measurements can change by 0.5 dB or more, depending on the DUT frequency range, gain, and noise figure.
Average Number  Increase the number of averages to reduce jitter. This also increases measurement speed. For maximum accuracy, use the following recommendations for the noise calibration. When using the noise receivers, 10 noise averages is recommended. When using the standard receivers, at least 100 averages are recommended.

During a measurement, the gain of the DUT helps overcome the noise of the PNA receivers, so the number of noise averages can be reduced to improve measurement speed with minimal or no degradation to measurement accuracy.

Noise Receiver

NA (Network Analyzer) Receiver (Opt 028) - Use a standard PNA receiver to measure noise figure.

- Connect the DUT to any PNA ports. For Vector NF measurements, connect the noise tuner to the source port.
- The sum of the gain and noise figure of the DUT must be at least 40 dB. This ensures that there is sufficient DUT noise power for the PNA to measure. Learn more.
- Additional filtering may be required. Learn more.

Noise Receiver (Opt 029) - Use internal low-noise receivers to measure noise figure.

- Opt 029 measures devices with Noise Figure values ranging from about 0 to 50 dB and devices with GAIN ranging from about -40 to +60 dB. Learn more.
- Amplifiers with higher gain can be measured by adding an attenuator to the output of DUT and using fixture de-embedding to remove the attenuator loss. An alternative for measuring high-gain devices is to use the standard receivers (Opt 028) as they have a higher compression level.

Receiver Gain

This setting is NOT available when Noise Receiver is set to NA (Network Analyzer) Receiver (Opt 028). With knowledge of your DUT gain, set the appropriate amount of receiver gain in order to optimize the power level at the noise receiver.

The following values reflect the SUM of the DUT gain (dB) PLUS NF (dB). For example: DUT gain = 20 dB; NF = 10 dB; SUM = 30 dB.

- Select High if the SUM is relatively low (<30 dB).
- Select Medium if the SUM is about average (20 dB to 45 dB).
- Select Low if the SUM is relatively high (>35 dB).

There is considerable overlap in these settings. Because all three gain settings are calibrated with each Noise Calibration, this setting can be changed after calibration to achieve the least amount of jitter without overpowering the noise receiver.

One of following messages appears when too much power is detected at the noise receiver:

- Compression in noise receiver: excess signal - The noise receiver is likely compressing. NF results
are possibly not accurate. Select a lower gain setting.

- **Compression in noise receiver: gain has been limited** - The gain has been limited to avoid damage to the receiver. NF results are NOT accurate. Select a lower gain setting.

- **ADC over-range in noise receiver: excess signal** - Often caused by a CW signal, an oscillation, or LO feedthru during an NF measurement. Find and correct the cause, or try a lower gain setting.

Only ONE gain setting can be used for the entire frequency range of your noise measurement. Therefore, it may be necessary to use two noise channels with different frequency ranges and gain settings to achieve the very highest noise figure accuracy.

**Ambient Temperature**

Enter the room temperature at the time of the measurement, in Kelvin. For best results, use a thermometer to read the temperature at the PNA test port 1 or the DUT input cable.

This ambient temperature number has an inverse relationship to the noise figure. When using the effective noise temperature (Te) format, a 3 degree increase in the ambient temperature will make the overall measurement result drop 3 degrees.

**Impedance States**

- **Noise Tuner** Displays the ECal module to be used as a noise tuner. Select the Noise Tuner during calibration on the Select Cal Method dialog.

- **Max Acquired Impedance States** Select the number of impedance states in which to make noise measurements. At least FOUR impedance states are required. Learn more

![Frequency Sweep Settings](image)
Frequency Tab - Noise Figure dialog box help

These settings can also be made from the normal PNA setting locations. Click links below to learn how.

**Sweep Type**
Choose a sweep type. Learn more.

**Segment Sweep Notes:**
- The segment table shown on the dialog is 'READ-ONLY'.
- Learn how to Create and edit the Segment Sweep table.
- Independent IFBW and Power are NOT available.
- X-axis point spacing is available beginning with A.09.10.

**Sweep Settings**
Click each to learn more about these settings.

- Number of points
- IF Bandwidth  This setting is important for improving noise measurement accuracy. Learn more.
- Start / Stop, Center / Span frequencies.
### Power Tab - Noise Figure dialog box help

#### Note:
S-parameter power settings are critical for accurate Noise Figure measurements. [See Noise Figure Measurement Tips](#).

Configures RF power settings for the S-parameter measurements that occur before noise measurements. Input power to the DUT is turned OFF during noise measurements.

These settings can also be made from the normal [Power setting](#) locations.

**Power ON (All channels)** Check to turn RF Power ON for all channels.

### DUT Input Port

- **Opt 028** - Select a PNA port to be connected to the DUT input.
- **Opt 029**
  - **Scalar Noise Figure** - Select a PNA port other than port 2.
  - **Vector Noise Figure** - The DUT input CAN be connected to any PNA port other than port 2. However, without a noise tuner bypass switch, measurements on other channels that use the same source port will always go through the noise tuner. The noise tuner must be connected to the source loop of the selected port.

#### Note: Input power levels are critical for accurate Noise Figure measurements. [Learn more](#).

**Power Level** The input power to the DUT during S-parameter measurements.

**Source Attenuator Auto** Check to automatically select the correct attenuation to achieve the specified input power. Clear, then select attenuator setting that is used achieve the specified Power Level. [Learn more about Source Attenuation](#).

All PNA channels in continuous sweep must have the same attenuation value. [Learn more](#).

**Receiver Attenuator** Specifies the receiver attenuator setting for port 1.

**Source Leveling** Specifies the leveling mode. Choose Internal. Open Loop should only be used when doing [Wideband Pulse measurements](#) (not available with Noise figure measurements).

### DUT Output Port

- **Opt 028** - Select a PNA port to be connected to the DUT output.
- **Opt 029** - Connect the DUT output to PNA port 2.

**Output Power** Sets power level in to the output port for reverse sweeps. Port power is automatically uncoupled. Reverse sweeps are always applied to the DUT when Full 2-port correction is applied. [Enhanced Response Cal](#) is NOT available for Noise Figure measurements.

**Source Attenuator** Specifies the source attenuator setting for reverse power.

**Receiver Attenuator** Specifies the receiver attenuator setting for the output port. [Learn more about Receiver Attenuation](#).

**Source Leveling** Specifies the leveling mode. Choose Internal.
Noise Path Configurator dialog box help

**Port 1 Noise Tuner Switch (Opt 029)** The orange line between CPLR THRU and SRC OUT represents the Noise Tuner. The **External** setting switches IN the Noise Tuner when making noise measurements.

**Port 2 Noise Receiver Switch (Opt 029)** allows you to make Noise Receiver measurements.

To prevent premature wear on the above two Noise switches, the PNA does not allow these switches to be thrown when sweeping a Noise channel and non-Noise channel. To make Noise Figure measurements and non-Noise Figure measurements in different channels and continuously trigger both, set these switches to the same state as the Noise channel:

- With the **non-Noise Figure channel** active, go to **Noise Path Configurator**.
- Set Noise Tuner switch to **External**. This routes source power to the front-panel loops, and to the Noise Tuner when connected. Use **CONT:ECAL:MOD:PATH:STATE** to set the internal state of the Noise Tuner to THRU, which creates a small amount of additional loss in the source path.
- Set Noise Receiver Switch to **Noise Receiver**.

**Noise Figure Measurement Tips**

**Note:** In this topic, the term **Jitter** is used to describe the trace-to-trace fluctuations in a measurement. In other topics, this is called 'trace noise'.

**Option 029**

*See Opt 028 (NF with Standard Receiver)*

- Measures devices with Noise Figure values ranging from about 0 to 50 dB and devices with GAIN ranging from about -40 to +60 dB.
- Highest Noise Figure accuracy is attained when the sum of device Noise Figure + GAIN is between 0 dB to
For highest Noise Figure accuracy and stability, there should be the least amount of electrical loss possible between the DUT output and PNA Port 2.

**Power level at the DUT Output - Opt 029**

S-parameters are used to measure the gain of the DUT before each series of noise measurements. Jitter in the S-parameter measurements corresponds directly to jitter in the noise measurements.

For best noise figure accuracy, the power level out of the DUT should be between 15 dB and 20 dB below the compression point of the DUT during the S-parameter portion of the noise figure measurement.

To reduce jitter, the power level at the B receiver (port 2) should be above approximately -20 dBm. Much below this level, S-parameter measurements have more jitter. Power must be below +10 dBm as the B receiver starts to compress at this point, although there is no warning or annotation that shows this condition is occurring in S-parameter measurements.

The best way to monitor power at the B receiver is to display a B,1 measurement. With your DUT in place and powered ON, change the input power to the device and note the power at the B receiver.

- For low-gain amplifiers, use 5 dB of source attenuation to improve the uncorrected match of port 1.
- For high-gain amplifiers, source and receiver attenuation may be required. Use the lowest possible attenuation values.

Attaining the optimum power level during calibration can also be challenging since calibration is performed without the DUT in place. Because of this, it is often necessary to set source power higher during the calibration than during the measurement. This will cause the 'CA' annotation on the status bar. Measurement results are accurate as long as the step attenuators and other configuration switches are in the same position and all receivers remain in their linear range (below +10 dBm).

It is best to find the optimum power and attenuation settings for both the calibration and subsequent noise measurements before performing a calibration.

**IF Bandwidth**

Jitter is further reduced by narrowing the IF bandwidth. If the calibration needs to be performed at a low source power, or with receiver attenuation due to high DUT gain, the IF bandwidth should be reduced during the calibration to reduce jitter. The IF bandwidth can then be increased to improve measurement speed. The CA annotation can be ignored when changing IFBW after calibration.

**Noise Settings**

See Noise Figure dialog box help for a complete description of these important settings.

**Temperature**

Noise Figure measurements are extremely sensitive to temperature. As such, there are two settings that require an accurate temperature measurement: At the DUT input, and at the Noise Source connector.

**Interference**

When measuring the noise figure of an unshielded device, like an amplifier on a printed-circuit board, it is very common to pick up interference from external signals such as cellular phones, wireless LAN, or mobile radios. This
interference shows up as non-repeatable spikes in the measurement, as shown below.

![Graph showing interference spikes](image)

Usually, the interference adversely affects the noise figure measurement only at the frequency where it occurs. However, if the interference is large enough and present all of the time, it can cause the noise receivers to compress, which results in inaccurate measurements at many frequencies. In this case, the noise figure measurements should be done in a shielded environment like a screen room.

**Option 028**

**Noise Figure of PNA receiver** - Option 028 gives you the flexibility to measure noise figure using a standard PNA receiver. For best measurement accuracy, the DUT excess noise power (gain plus noise figure in dB) should meet or exceed the noise figure of the receiver (typically about 40 dB, reference to the test port). For devices that do not meet this requirement, there are two alternatives:

1. Reverse the main arm and coupled arm of the test-port coupler (see following images). This increases the signal to the receiver port by about 15 dB, while lowering the available port power by the same amount. This is a good tradeoff for noise figure measurements, as it lowers the excess noise threshold mentioned above from 40 dB to about 25 dB.

   ![Block diagram](image)  
   ![Re-configured coupler](image)

   Re-configure the receiver port front-panel loops to a vertical orientation.

   Block diagram showing port 2 through coupler main arm to B receiver.
2. Add an low-noise amplifier (LNA) to the receiver loop (see following image). This boosts the noise power at the receiver by the gain of the LNA. The disadvantage is the possibility of measurement drift and receiver compression. Any change in the gain of the LNA will have an impact on measurements that use the receiver with the LNA, so frequent calibration may be required. Care should also be taken when setting the channel power (used during the gain portion of the measurement) to ensure that the added gain of the LNA does not cause receiver damage or compression. A filter is also required on the output of the LNA (not shown). Learn more.

Filtering Requirement (Option 028)
Opt 029 includes noise receivers with filtering to keep mixing-product noise out of the low-noise receivers. These filters are not available when measuring noise with the standard PNA receiver. Therefore, for best measurement accuracy, a filter should be used at the output of the DUT (or LNA preamp if used).

- A bandpass filter at the frequencies of interest can always be used.
- A lowpass filter can be used when the PNA is doing fundamental mixing (up to 26.5 GHz). The lowpass filter must pass the fundamental frequency of the measurement but suppress the third harmonic. A measurement at 1 GHz would need a lowpass filter with a cutoff below 3 GHz, while a 5 GHz measurement would need a filter with a cutoff below 15 GHz.
- A single highpass filter can be used when the PNA is doing 3rd harmonic mixing (from 26.5 to 50 GHz). Use a highpass filter with cutoff about 18 GHz.

Using Noise Figure Traces in Equation Editor
In a Noise Power trace, the underlying unit is noise temperature.

\[10 \times \log_{10} \left( \text{temperature} \times 1000 \text{mw/w} \times 1.38 \times 10^{-23}\right)\]

(1.38e-23 is Boltzmann's constant)

Any time you use Equation Editor on a Noise Power trace, the LogMag formatting will apply the above equation. Therefore, first select REAL format and then generate the equation.

Noise Model and the Noise Correlation Matrix
The noise wave model of any linear 2-port network may be represented by the following image:
This shows a noiseless 2-port network with noise waves \( a_{n1} \) and \( b_{n1} \) added to the input terminals. 

The noise correlation matrix relates to the noise waves as follows:

\[
C_t = \begin{bmatrix}
|a_{n1}|^2 & \bar{a}_{n1} \bar{b}_{n1} \\
\bar{b}_{n1} \bar{a}_{n1}^* & |b_{n1}|^2
\end{bmatrix} = \begin{bmatrix}
c_{11} & c_{12} \\
c_{21} & c_{22}
\end{bmatrix}
\]

Where:

- \(|a_{n1}|^2\) and \(|b_{n1}|^2\) are time-averaged noise power in 1 Hz bandwidth.
- \(\bar{a}_{n1} \bar{b}_{n1}\) and \(\bar{b}_{n1} \bar{a}_{n1}^*\) are time-averaged cross correlation terms, correlation of \(a_{n1}\) to \(b_{n1}\).

---

**Radio-Frequency Electromagnetic Field Immunity**

When a 3V/m-1 radio-frequency electromagnetic field is applied to an PNA with Opt 029 according to IEC 61000-4-3:1995, degradation of performance may be observed. When the frequency of the incident field matches the frequency of a measured noise figure or gain, the values displayed will deviate from those expected. This phenomenon will only affect that specific frequency, and the analyzer will continue to perform to the specification at all other frequency sample points.

The PNA with Opt 029 may be unable to calibrate a chosen frequency sample point if the frequency matches that of an incident electromagnetic field.

---

**Last Modified:**

- 4-May-2011 Edited for N522x
- 27-Apr-2011 Removed Copy Channels limitation
- 7-Apr-2011 Edited cal bandwidth and average
- 11-Mar-2011 Added note for Noise Sources
- 3-Dec-2010 Clarified Noise BW setting
- 2-Sep-2010 Added using equation editor
- 28-Apr-2010 Added port mapping (9.22)
- 12-Mar-2010 Added 028 and H29 - removed some fixturing limitations (9.20)
19-Nov-2009  Added X-axis point spacing (9.1)
31-Jul-2009  Added interference tip
15-Jul-2009  Added reference to other tuner models
6-Apr-2009  Replaced N5242a with PNA
25-Feb-2009  Added Incident parameters and Scalar (A.08.50)
18-Feb-2009  Added Noise model
29-Sep-2008  Added Port Ext to NOT supported list
22-Apr-2008  Added link to preferences
12-Mar-2008  Modified Noise Params
29-Jun-2007  MX New topic
Noise Figure on Converters (NFX) - Opt 028 and Opt 029

This topic discusses Noise Figure on Converters:

- Requirements and Limitations
- How to Configure your Hardware
- NFX Parameters
- Create a NFX Measurement
- Valid Mixer Configuration / Sweep Type Combinations
  - Frequency tab
  - Power tab
  - Mixer Frequency tab (separate topic)
  - Mixer Power tab (separate topic)
  - Mixer Setup tab (separate topic)

The following general information (contained in Noise Figure Application for Amplifiers) is also relevant for NFX measurements:

- Noise Figure Options Explained (029, 028, H29)
- Features and Requirements
- Noise Concepts
- How the Noise Figure Application Works
- Scalar Noise Figure Measurements
- Perform Calibration
- Noise Figure Measurement Tips
- Noise Model and the Noise Correlation Matrix

### Other Application Topics

### Requirements and Limitations

Noise Figure on Converters requires Noise Figure (Option 0pt 028 or 029) and FCA (Opt 082 or 083).

Learn more about Noise Figure requirements.
Limitations

- Upconverters OR downconverters ONLY (not both).
- Image rejection is NOT provided in NFX.

The following PNA features are **NOT** available with NFX:

- Analog Sweep ([Stepped sweep](#) mode only)
- Log frequency sweeps
- ECal User Characterization
- Time Domain
- Balanced measurements
- Save [Formatted Citifile](#) data.
- Interface Control
- Some Fixturing Features
- External Test Set Control (Option 551)
- Pulse measurements
- See Frequency Limitations

**Embedded LO** measurements **ARE** supported in NFX. [Learn more.]

How to Configure your Hardware

The PNA-X is extremely versatile, and can be configured in many ways to make NFX measurements. While not all conceivable configurations are documented here, a few of the most common examples are provided to show the basic concepts.

**DUT Configuration**

[Learn more about connecting the DUT input and output to the PNA.](#)

The DUT LO can be connected to an external source OR [PNA internal second source (if available).](#)

Select an LO Source on the [Mixer Setup tab.](#)

**Note:** Noise that is present on the LO source will be directly transferred to the DUT output. This noise can NOT be calibrated out of the noise measurement. Therefore, choose a low-noise source for the LO, such as an Agilent ESG or PSG, which is better than the PNA internal source.

**External LO Source though Port 3 or Port 4 (4-port PNA-X only)**

- Connect the DUT LO to PNA-X Port 3 or Port 4.
- Connect external source to rear-panel J7 (Rear-panel load on below diagram) for Port 3; J3 for Port 4.
  - For Port 3, NO switching is required.
  - For Port 4, switch Port 4 Bypass Switch to Rear Panel.

**PNA-X Internal LO Source**

- SRC 2 Out 1 or Out 2 on front-panel or 2-port / 2-source PNA-X
- Port 3 or Port 4 on 4-port PNA-X

**Path Configurator - Default configuration on PNA-X option 423**

- By default, Internal source2 (Src2) is supplied at ports 3 and 4.
- Bypass switches must be in “Thru Path”

**NFX Parameters**

Several noise power parameters, as well as mixer parameters and unratioed parameters, can be measured in the same NFX channel.
How to add NFX Parameters

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press Traces</td>
<td>1. Click Trace/Chan</td>
</tr>
<tr>
<td>2. then [New Trace]</td>
<td>2. then New Trace</td>
</tr>
<tr>
<td>3. then select a parameter</td>
<td>3. then select a parameter</td>
</tr>
</tbody>
</table>

Noise Figure Parameters that are offered:

- **Noise Figure (NF)** - Explained in Noise concepts.
- **Excess Noise Ratio** - Select when measuring the noise source. Compare with the ENR table to validate accuracy of the system. ENR is calculated as:

  \[
  \text{ENR (in dB)} = 10 \log_{10} \left( \frac{T_{\text{hot}} - T_{\text{cold}}}{T_0} \right), \text{ where } T_0 = 290 \text{K.}
  \]

  Learn more about the ENR table and Noise Source. Learn more about Noise Source ENR measurements.

- **T-Effective** - The effective temperature, in Kelvin, of the measured noise level. For example:

  \[
  290^\circ \text{K} = -174 \text{dBm/Hz.}
  \]

  \( T_e \) is the unknown variable

  Available Gain \( G_a \) is a function of \( S_{11}, S_{22}, \) and \( \Gamma_s \)

  The Noise Power parameters below are offered in the following two formats:

  - **Available Noise Power** - The calculated power that is based on an ideal impedance match at the output of the DUT. These parameters have always been offered in the PNA Noise Figure App.
• **Incident Noise Power** - An 'I' is appended to the end of the Available Noise Power parameter. The calculated power into a perfect 50 ohm noise receiver, regardless of the output impedance of the DUT.

**Noise Power Parameters:**

• **SYSNPD / SYSNPDI** - System Noise Power Density: Total noise power available at the ADC, including the noise contributed by both the DUT and the internal noise receiver. This is generally expressed as an absolute power measurement in dBm, but can also be expressed in Watts or Kelvin.

\[
\text{dBm} = 10 \log_{10}(k \times T \times B \times 1000)
\]

where:

- \(k\) = Boltzmann's constant
- \(T\) = the measured noise temperature
- \(B\) = bandwidth
- 1000 = conversion from milliwatts

• **SYSRNP / SYSRNPI** - System Relative Noise Power: The noise temperature of the combined DUT and receiver relative to 290 Kelvin. This is generally reported as a ratio in dB. Therefore a perfectly quiet device would render a trace at 0 dB.

\[
\text{dB} = 10 \log_{10}(T/290)
\]

• **DUTNPD / DUTNPDI** - DUT Noise Power Density: When correction is ON, this trace exhibits the available noise power, best described as the maximum power available from the DUT where the impedance of the noise port is equal to the output match of the DUT. To be more precise, this occurs when the noise port match is equal to the conjugate of the output match of the DUT. The noise power contributed by the receiver is removed.

When correction is OFF, the trace exhibits what is more accurately described as delivered power. Delivered power is the power actually seen by the ADC. Any mismatch between the receiver and the DUT is ignored. The noise power contributed by the receiver is removed.

This measurement is generally expressed in dBm:

\[
\text{dBm} = 10 \log_{10}((\text{DUT Temperature} - \text{Receiver Temperature}) \times B \times 1000)
\]

where:

- \(B\) = bandwidth
- 1000 = conversion from milliwatts

• **DUTRNP / DUTRNPI** - DUT Relative Noise Power: This measurement is rendered as a ratio of the DUT
temperature to 290 Kelvin. It is generally expressed in dB. The same comments apply with respect to available versus delivered power as described above for DUTNPD.

\[ dB = 10 \log_{10} (\text{DUT Temperature} - \text{Receiver Temperature}) \]

**Mixer Parameters**

- **SC21**: Conversion Loss
- **SC12**: Reverse Conversion Loss
- **S11**: Input match
- **S22**: Output match
- **IPWR**: Input power to mixer/converter. Same as R1 (Source 1)
- **OPWR**: Output power to mixer/converter. Same as B (Source 1)
- **RevIPWR**: Power applied to mixer/converter Output. Same as R2 (Source 2)
- **RevOPWR**: Power measured at mixer/converter Input. Same as A (Source 2)

**Raw Receiver Parameters**

Specify a receiver to measure at LO1 frequencies with the notation:

- `<Receiver>LO1`
- For example: **ALO1** or **R1LO1**

Specify a receiver to measure using a source port, with the notation:

- `<Receiver>_<source port>`
- For example: **A_3** or **R1_1**

**Using the Noise Figure Application**

Use the following general procedure to make measurement with the Noise Figure App:

1. Connect Tuner and Noise Source.
2. Create a Noise Figure Measurement.
3. Make Noise Figure Settings.
4. For Opt 029 and H29, copy your Noise Source ENR file to the PNA `C:/Program Files/Agilent/Network Analyzer/Noise` folder.
5. Perform Calibration
6. Connect the DUT. Learn more.

7. Measure Noise Figure.

8. **Optional** Click File, then Save to save Noise Figure data in the following formats: (available ONLY when NF correction is ON.)

   - *.CTI Citifile
   - *.PRN
   - *.nco Noise Correlation Matrix data in S2P format. See Noise Model.

See Also: Measurement Tips

**Connect Noise Tuner and Noise Source**

1. Connect the **noise source** to the 28V connector on the PNA-X rear panel. **NOT required for Opt 028.** The Noise Source is turned ON and OFF automatically as needed during a calibration. Connect the noise tuner to Port 2 reference place when prompted during calibration.

2. Connect the **noise tuner** (ECal module). **NOT used for** Scalar Noise Figure **measurements.**

   a. On the PNA front panel, remove the **Port 1** jumper cable SOURCE OUT / CPLR THRU. Opt 028 allows Noise Figure measurements using any two PNA ports. Connect the noise tuner to the front-panel jumpers for the source (DUT input) port.

   b. Connect M-F tuner (N4691B-M0F) using the supplied cable (N5242-20137) and adapter (85052-60013).

   c. When using F-F ECal module (N4691B-00F), order a 3.5 mm M-M adapter (85052-60014).

![Image of connectors]

**Create a Noise Figure Measurement**

1. On the PNA front panel, press Meas, then **[Measurement Class]**

2. Select **Noise Figure Converters**, then either:

   - OK delete the existing measurement, or
• **New Channel** to create the measurement in a new channel.

3. A Noise Figure measurement is displayed. To select additional parameters to display, click **Response**, then **Measure**, then select a parameter from the list.

### How to start the Noise Figure Setup dialog

To provide quicker access, use the Setup softkey. [Learn how](#).

#### Using front-panel HARDKEY [softkey] buttons

1. Press **FREQ**
2. then **[Noise Figure Setup]**

#### Using a mouse with PNA Menus

1. Click **Response**
2. then **Measure**
3. then **Noise Setup**

### Valid Mixer Configuration / Sweep Type Combinations

The following are the **Valid Mixer Configurations**:

<table>
<thead>
<tr>
<th>Sweep Type</th>
<th>Input</th>
<th>LO</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>Swept</td>
<td>Swept</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Swept</td>
<td>Fixed</td>
<td>Swept</td>
</tr>
<tr>
<td>CW</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
</tbody>
</table>

**RED** messages, like the following, appear at the bottom of the NFX Setup dialog to notify you of an invalid setup.

```
ERROR: All ranges must be fixed for Tone Power sweep mode
```

The following tabs are available on the NFX Setup dialog:

- **Frequency**
- **Power**
- **Noise Figure Setup**
- **Mixer Frequency** (separate topic)
- **Mixer Power** (separate topic)
- **Mixer Setup** (separate topic)

### Frequency tab - NFX Setup

#### Sweep Type
Linear - Use for Swept Input parameters
CW Frequency - Use for Fixed Input parameters

#### X-Axis Display
Annotation: Select the frequency range to display on the X-axis.

#### Sweep Settings
Click to learn more about these settings.

- **Number of points**
- **IF Bandwidth**  For standard PNA receiver measurements. This setting is important for improving noise measurement accuracy. [Learn more.](#)
- **Start / Stop, Center / Span** frequencies.

[Learn about the Load/Save *.mxr files](#), and other buttons across the bottom of all NFX Setup tabs.

---

### Power Tab - NFX Setup

---
Note: S-parameter power settings are critical for accurate Noise Figure measurements. See Noise Figure Measurement Tips.

Configures RF power settings for the S-parameter measurements that occur before noise measurements. Input power to the DUT is turned OFF during noise measurements.

These settings can also be made from the normal Power setting locations.

**Power ON (All channels)** Check to turn RF Power ON for all channels.

**DUT Input Port**

- Opt 028 - Select a PNA port to be connected to the DUT input.
- Opt 029 Scalar Noise Figure - Select a PNA port other than port 2.
- Opt 029 Vector Noise Figure - The DUT input CAN be connected to any PNA port other than port 2. However, without a noise tuner bypass switch, measurements on other channels that use the same source port will always go through the noise tuner. The noise tuner must be connected to the source loop of the selected port.

Note: Input power levels are critical for accurate Noise Figure measurements. Learn more.

**Power Level** The input power to the DUT during S-parameter measurements.

**Source Attenuator Auto** Check to automatically select the correct attenuation to achieve the specified input power. Clear, then select attenuator setting that is used achieve the specified Power Level. Learn more about Source Attenuation.

All PNA channels in continuous sweep must have the same attenuation value. Learn more.

**Receiver Attenuator** Specifies the receiver attenuator setting for the input port.

**Source Leveling** Specifies the leveling mode. Choose Internal.

**DUT Output Port**

- Opt 028 - Select a PNA port to be connected to the DUT output.
- Opt 029 - Connect the DUT output to PNA port 2.

**Output Power** Sets power level in to the output port for reverse sweeps. Port power is automatically uncoupled. Reverse sweeps are always applied to the DUT when Full 2-port correction is applied. Enhanced Response Cal is NOT available for Noise Figure measurements.

**Source Attenuator** Specifies the source attenuator setting for reverse power.

**Receiver Attenuator** Specifies the receiver attenuator setting for the output port. Learn more about Receiver Attenuation.

**Source Leveling** Specifies the leveling mode. Choose Internal.

Learn about the Load/Save *.mxr files, and other buttons across the bottom of all NFX Setup tabs.
Noise Path Configurator dialog box help

Port 1 Noise Tuner Switch (Opt 029) The orange line between CPLR THRU and SRC OUT represents the Noise Tuner. The External setting switches IN the Noise Tuner when making noise measurements.

Port 2 Noise Receiver Switch (Opt 029) allows you to make Noise Figure measurements using the internal low-noise receivers.

To prevent premature wear on the above two Noise switches, the PNA does not allow these switches to be thrown when sweeping between a Noise channel and non-Noise channel. To make Noise Figure measurements and non-Noise Figure measurements in different channels and continuously trigger both, set these switches to the same state:

- With the non-Noise Figure channel active, go to Noise Path Configurator.
- Set Noise Tuner switch to External. This routes source power to the front-panel loops, and to the Noise Tuner when connected. Use CONT:ECAL:MOD:PATH:STATE to set the internal state of the Noise Tuner to THRU, which creates a small amount of additional loss in the source path.
- Set Noise Receiver Switch to Noise Receiver.

NFx Noise Figure Setup dialog box help
Bandwidth/Average

The following two settings work together to achieve the optimum balance of measurement accuracy versus speed. Both settings can be changed after calibration to make faster measurements with minimal effect on calibration accuracy.

**Noise Bandwidth** Increase the bandwidth to make faster measurements. However, a wider setting reduces the frequency resolution of the measurement. More frequencies are essentially smoothed together to produce a flatter response, which could hide the actual noise performance of the DUT.

**Noise Averaging Factor** Increase the number of averages to reduce jitter. This also increases measurement speed. For maximum accuracy, increase the averaging factor for the noise calibration. It can then be reduced to improve measurement speed.

NA (Network Analyzer) Receiver (Opt 028) - Use a standard PNA receiver to measure noise figure.

- Connect the DUT to any PNA ports. For Vector NF measurements, connect the noise tuner to the source port.
- The sum of the gain and noise figure of the DUT must be at least 40 dB. This ensures that there is sufficient DUT noise power for the PNA to measure. Learn more.
- Additional filtering may be required. Learn more.

Noise Receiver (Opt 029) - Use internal low-noise receivers to measure noise figure.

- Measures noise figure values ranging from about 0 to 50 dB. Measures devices with gain ranging from about -40 to +60 dB.
- Converters with higher gain can be measured by adding an attenuator to the output of DUT and using fixture de-embedding to remove the attenuator loss. An alternative for measuring high-gain devices is to use the standard receivers (Opt 028) as they have a higher compression level.

Receiver Gain

This setting is NOT available when Noise Receiver is set to NA (Network Analyzer) Receiver (Opt 028).

With knowledge of your DUT gain, set the appropriate amount of receiver gain in order to optimize the power level at the noise receiver.

The following values reflect the SUM of the DUT gain (dB) **PLUS** NF (dB). For example: DUT gain = 20 dB; NF
= 10 dB; SUM = 30 dB.

- Select **High** if the SUM is relatively low (<30 dB).
- Select **Medium** if the SUM is about average (20 dB to 45 dB).
- Select **Low** if the SUM is relatively high (>35 dB).

There is considerable overlap in these settings. Because all three gain settings are calibrated with each Noise Calibration, this setting can be changed after calibration to achieve the least amount of jitter without overpowering the noise receiver.

One of following messages appear when too much power is detected at the noise receiver:

- **Compression in noise receiver: excess signal** - The noise receiver is likely compressing. NF results are possibly not accurate. Select a lower gain setting.
- **Compression in noise receiver: gain has been limited** - Damage to the receiver is possible, but not likely. NF results are NOT accurate. Select a lower gain setting.
- **ADC over-range in noise receiver: excess signal** - Often caused by a CW signal, an oscillation, or LO feedthru during an NFX measurement. Find and correct the cause, or try a lower gain setting.

Only ONE gain setting can be used for the entire frequency range of your noise measurement. Therefore, it may be necessary to use two noise channels with different frequency ranges and gain settings to achieve the very highest noise figure accuracy.

**Ambient Temperature**

Enter the room temperature at the time of the measurement, in Kelvin. For best results, use a thermometer to read the temperature at the PNA test port 1 or the DUT input cable.

This ambient temperature number has an inverse relationship to the noise figure. When using the effective noise temperature (Te) format, a 3 degree increase in the ambient temperature will make the overall measurement result drop 3 degrees.

**Impedance States**

- **Noise Tuner** Displays the ECal module to be used as a noise tuner. Select the Noise Tuner during calibration on the Select Cal Method dialog.

- **Max Acquired Impedance States** Select the number of impedance states in which to make noise measurements. At least FOUR impedance states are required. Learn more.

- **Enable Source Pulling for S-Parameters** (For mixers with low reverse isolation). When checked, during S22 (output match) measurements, the noise tuner is switched to present different impedance states to the DUT input. From these measurements, S22 is computed as though the input is seeing a 50 ohm match. This requires more sweeps. Check this box when the converter has low reverse isolation, as is the case when the NO output path is NOT padded with attenuation. Otherwise, clear this checkbox as S22 measurements will not be improved.

An accurate S22 measurement is essential when measuring S-parameters during an NFX calibration. Learn about the Load/Save *.mxr files, and other buttons across the bottom of all NFX Setup tabs.
Mixer Frequency tab - NFX Setup dialog box help

Learn about this dialog.

Mixer (LO) Power tab - NFX Setup dialog box help

Learn about this dialog.
Learn about this dialog.

Last Modified:

27-Apr-2011  Removed Copy Channels limitation
16-Aug-2010  Divert mixer tabs
30-Apr-2010  Added 028 and port mapping
3-Mar-2010   Fixed Meas Class - removed some fixturing limitations
6-Oct-2009   MX New topic (A.09.10)
Calibration for Noise Figure on Amplifiers and Converters (NFX)

This topic discusses calibration for both Noise Figure on Amplifiers and Noise Figure on Converters (NFX).

- **Overview**
- **How to Perform a Noise Figure Cal**
  - Select Calibration Method
  - Configure Noise Source
  - Select DUT Connectors and Cal Kits
  - Measure Standards Steps
  - Validate Noise Source Cal

See Also: Scalar Noise and TRL Cal

See Noise Figure or NFX Applications

**Overview**

Noise Figure results are NOT at all accurate without a Noise Figure calibration.

Noise Figure Calibration is very similar for both amplifiers and converters (NFX).

- NFX does NOT offer 'S-params ONLY' calibration.
- NFX includes an optional LO Power Cal.
- NFX and NF with a standard receiver includes a Power Sensor Cal on the source port.

The Noise Figure calibration process is different depending on whether a Noise Source is used to calibrate the noise receiver. In this discussion, the term 'noise receiver' is used to refer to the receiver that is used to measure noise. It can be a standard PNA receiver or the dedicated noise receivers that are supplied with Opt. 029.

**Calibrating the Noise Receiver using a Power Meter**

When Use Power Meter is selected on the Noise Cal Select Method dialog a noise source is NOT used to calibrate the noise receiver. Instead, a calibrated PNA source is used to calibrate the noise receiver.

1. A Source Power Cal is performed at the power level that is specified on the first measurement step of the calibration wizard.
2. A THRU connection is made from the calibrated source port to the noise receiver.
3. The gain bandwidth and noise level of the noise receiver is characterized. This lengthy process is performed
every time a Noise Figure calibration is performed.

4. During the noise measurements, noise averaging and noise bandwidth is automatically turned ON to the values that you specify. Learn more about Noise Averaging.

Calibrating the Noise Receiver using a Noise Source

See Noise Source requirements

A Noise Source is a device that generates two very consistent levels of noise over its operating frequency range:

- Hot (On) - the Noise Source is biased in order to provide a high level of noise.
- Cold (Off) - the Noise Source is unbiased to provide ambient temperature noise level.

These levels are measured by the Noise Source manufacturer and provided in table and electronic format with each Noise Source by serial number. The electronic file is known as the ENR (Excess Noise Ratio) file.

1. The Noise Source is connected to the noise receiver through test port 2. **Note:** For highest accuracy, the noise source should be connected as close as possible (the least amount of electrical loss) to the PNA port 2 connector. This causes the largest difference between the Noise Source HOT (on) and COLD (off) settings.

2. The Noise Source is measured by the noise receivers at each measurement frequency. The differences between the known ENR noise levels and the measured noise levels are the noise error terms. These values are removed from subsequent noise measurements.

3. During the Noise Source measurements, noise averaging and noise bandwidth is automatically turned ON to the values that you specify. Learn more about Noise Averaging.

Following the Noise Receiver Cal

- A 2-port S-parameter calibration is performed on the Noise Figure channel. This is because S-parameters are measured at each frequency step before a noise measurement. Also during the S-parameter cal, at least FOUR different impedance states are presented at port 2 in order to later characterize the noise generated by the noise receiver. This cal can be either a SOLT or TRL cal. See Scalar Noise and TRL Cal.

- After calibration, correction is automatically turned ON. The PNA status bar shows VNC_2P (for Vector) and SMC_2P (for Scalar).
How to Perform a Noise Figure Calibration

- Make the Noise Figure channel the active channel.
- Connect the Noise Figure Tuner to the PNA (for Vector Noise Figure cal).

Using front-panel HARDKEY [softkey] buttons

1. Press CAL
2. then [Cal Wizard]

Using a mouse with PNA Menus

1. Click Response
2. then Cal

The following Cal Wizard pages are unique to Noise Figure Calibration. The remaining pages that are presented are the same as those in the standard Cal Wizard SmartCal.

Select Calibration Method dialog box help

Calibration Method

- Vector Noise - Comprehensive Noise Figure calibration
- S-Parameter Only - Amplifiers ONLY. Does NOT calibrate the noise receivers.
- Scalar Noise - Calibration for Scalar Noise Figure measurements. Learn more.

Enable LO Power Cal - NFX ONLY. Check to cause the Cal Wizard to guide you through a Power Calibration on the LO source.

Note: NO correction is provided for an adapter that may be used to connect the power sensor to the LO source.

Noise Tuner

- Not available when Scalar Full is selected.
- Select from the ECal modules that are connected to the USB.
**Orientation**

**AutoOrient Tuner** Check to allow the noise tuner orientation to be auto-detected. When cleared, use the following two fields to provide manual orientation of the noise tuner.

**Tuner In (SOURCE OUT) / Tuner Out (CPLRTHRU):** Specify the ECal module labels that are connected to the PNA front panel jumper connectors. Learn how to connect the noise tuner.

**Detect Tuners** Click to re-detect the Noise Tuners (ECal modules) that are connected to the USB. If the ECal module is not detected, check the USB connection, then click this button. The label below the button indicates the total number of ECal modules that are connected to the USB.

**Receiver Characterization** - Learn more about this process

- **Use Noise Source** - A noise source is used to characterize the noise receiver.
- **Use Power Meter** - A calibrated PNA source is used to characterize the noise receiver. The PNA source is calibrated using a Power Meter/Sensor. This selection is made for you and can NOT be changed when **NA Receiver** is selected on the **Noise Figure Setup dialog**.

---

**Configure Noise Source (Opt 029) dialog box help**

**ENR File** Select the Noise Source ENR file. If not already there, copy your Noise Source ENR file to the PNA C:/Program Files/Agilent/Network Analyzer/Noise folder. Then click Browse to find the ENR file.

**Clear ENR List** Scroll to the bottom of the ENR list, then click to removes the selected ENR file. Then browse or select to find a new file.

**Edit ENR** Click to launch the **ENR Editor** dialog box which is used to change or create ENR files. This is NOT usually necessary.

**Temperature** Specify the current temperature at the Noise Source connector. The Noise source is kept ON during Noise Figure measurements. This results in the Noise Source being 4 to 5 degrees warmer than Ambient temperature, and a more accurate calibration. See **Noise Figure tips** to learn more about the significance of temperature.

See **Noise Source requirements**.
Click either Create or Edit to launch the same dialog box, shown below.

- **Edit** populates all fields with existing data which can then be edited and stored.
- **Create** has empty fields except for frequencies.

## Edit / Create ENR File dialog box help

### ENR Numeric Data
Use Previous and Next buttons to scroll to Entry # to edit. Type ENR value in dB, then press Enter.

Done Click when finished editing all values. Then click Store ENR File to save the file.

### Identifying Data
- **Model #** of the Noise Source. This can NOT be changed.
- **Serial #** of the Noise Source.
- **Temperature and Humidity** in which the Noise Source was calibrated. This is for information only. The ENR data is always normalized to 290 Kelvin.
- **KeyBd** launches a mouse-driven keyboard.
- **Store ENR File** Click to launch a dialog to save the new or edited ENR file.
Port 1 and Port 2

**DUT (Device Under Test) Connectors** Specify the connector and gender of the **DUT**.

**Cal Kits** Select the Cal Kit to be used to calibrate each test port. The list for each DUT Port displays kits having the same connector type as the DUT. Using incorrect calibration standards can significantly degrade measurement accuracy. [Learn more](#).

**Power Sensor** NFX Cal and **NF with a standard receiver ONLY**. Used to calibrate the source port.

**Noise Src** Used to calibrate the noise receivers (Opt 029). The Agilent 346C has an "APC 3.5 male" connector.

**Note:** For highest accuracy, the noise source should be connected as close as possible to the PNA port 2 connector. This causes the largest difference between the Noise Source HOT (on) and COLD (off) settings.

For both Cal devices (power sensor and noise source, specify the connector type and gender. When the Cal device connector is **NOT** the same type and gender as the DUT Port connector, then for optimum accuracy, extra cal steps are used to measure and correct for the adapter that is used to connect the Cal device to the reference plane.

Select **Ignored** (at the bottom of the DUT Connectors list) to NOT compensate for the adapter.

Select the Cal Kit that will be used for that process.

**De-embed power sensor adapter / noise source adapter** The PNA uses the connector type and gender of the DUT along with the connector type and gender of the cal device to determine if an adapter removal operation is taking place AND whether or not that removal operation requires an additional cal step.

However, the use of the connector type can, in special cases, hide the need for the extra cal step. Check the “De-embed...” box in these cases to inform the PNA that the extra step is needed.

Such a case is illustrated below where the noise source is connected close to test port 2 for higher accuracy. If unchecked, the PNA would assume in this case that the Noise Source is connected to the Thru standard at the port 1 (DUT input) reference plane.
**Source Cal Settings**  Click to launch the [Source Power Cal (for apps)](https://example.com) dialog. This dialog is used to set Power Meter / Sensor settings for both the Port 1 Power Cal, and the optional LO Power Cal.

**Modify Cal**  Check, then click **Next**, to Modify Cal (Standards AND Thru Method).

**Note:** Enhanced Response Calibration is NOT supported with Noise Figure.

---

**Measure Standards Steps** dialog box help

**Power Level** at which to perform the Power Cal.

- It is usually best to set power level to 0 dBm at the power sensor because the power sensor is calibrated at that level. Lower power levels will yield a slower and noisier calibration.
- However, with 20 dB of source attenuation (default NF setting), the PNA may not be capable of achieving this power level at higher frequencies. To check the max leveled power, view an R1 (port 1 reference receiver) trace over the frequency range of interest, then increase the power until roll-off appears. Power levels at the test port may be approximately 2 dB lower than at the R1 receiver.
- If an external component is used between the PNA test port and the calibration reference plane, then adjust the power level so that the power at the sensor is about 0 dBm if possible.
- The current source attenuation value is shown on the dialog.

**LO Power Cal (Optional)** When enabled, perform a Source Power Cal at the DUT LO input. An LO must already be selected. Learn how. The power level of the LO source calibration is set on the [NFX (LO) Power Tab](https://example.com).

**Connect Noise Source to the Port 2 measurement (reference) plane**

When the "De-embed Adapter.." boxes are checked, additional cal steps are required.

**Subsequent Steps**

- **Connect Port 1 to Port 2** - Connect port 1 reference plane to the port 2 reference plane using the required Thru standard or adapter.
- **Connect ECal to Ports 1 and 2** - Connect the ECal module between the port 1 reference plane and the port 2 reference plane.

**Validate Noise Source Cal**

To validate a Noise Source Calibration, connect the Noise Source to Port 2 and measure **ENR**.

Compare the measured values to the values in the ENR table.

**How to manually turn the Noise Source ON | OFF**
1. Press **POWER**

2. then **[Noise Source ON | OFF]**
Swept IMD and IM Spectrum Concepts

- Swept IMD Concepts
- Swept IMD for Converters (separate topic)
- Swept IMD Parameters
- How the PNA Measures IMD
- How an IM Spectrum Channel Works
- IM Spectrum Parameters

Other IMD topics

Swept IMD (Intermodulation Distortion) Concepts

When a device or system is subjected to multiple input frequencies, the non-linearity of the DUT can generate undesired outputs at other frequencies. Typically, two input tones of equal power separated in frequency by a specified amount are used to stimulate the device while observing the resulting frequency spectra at the output. A variety of measurements can then be utilized to determine the intermodulation distortion characteristics of the device.

The frequencies of the resulting distortion products are predictable. While many mixing products can be generated, the high and low signals of the "odd order" products (3rd, 5th, and so forth) are close enough to the original two signals to potentially interfere with adjacent communication channels. With the exception of the 2nd order product, the higher "even order" products are usually far enough away to be of no interest.

The following image and table shows two equal-power main-tones and the nearby odd-order distortion products, as well as the 2nd order product (not shown in the image). Notice that the frequency separation between adjacent odd-order products is the same as the separation of the main tones (Delta F) frequency. For most devices, these distortion products become worse as the device is pushed further into compression.

![Diagram showing two main tones and odd-order intermodulation products.](image)

*Two main tones (f1 and f2) with odd order intermodulation products.*

The following table shows the calculations and example frequencies *(Blue text)* of the intermodulation products that are closest to the two main tones.
### Swept IMD Parameters

The following basic parameters, offered for both Amps and Converters, are expanded to over 150 by selecting specific product tones (2,3,5,7,9), the Low-side, High-side, or Average of these tones, measured at the Input or Output of the DUT.

- **Tone Power**
- **Tone Gain**
- **Intermodulation Distortion**
- **Intercept Point Parameters**
- **Composite Triple Beat (CTB)**
- **Composite Second-order Beat (CSO)**
- **Cross-Modulation Distortion**

Learn how to select these IMD parameters.

### Tone Power Parameters

Tone Power parameters measure the **absolute** power level of the main tones, odd-order product tones up to the 9th order, and the 2nd order product tones. These tone powers can be measured at the input and output of the
DUT. Because the tones come in pairs, the **Low tone**, **High tone**, and the **Average** of the two can be measured and displayed.

The Average Tone Power is calculated as follows:

\[
\text{Avg} = \frac{\text{High tone (dBm)} + \text{Low tone (dBm)}}{2}
\]

When measuring the 2nd order products, only the Low tone and High tones are allowed. When the main tones are separated by less than 10 MHz, the Low tone (f2-f1) is below the frequency range of the PNA.

**Tone Gain**

Tone Gain (in dB) calculates the main tone Output Tone power / Input Tone power. Because the tones come in pairs, Tone Gain can be calculated for the **Low tone**, **High tone**, and the **Average** of the two as indicated in the Average Tone Power calculation.

For IMDx for Converters, the Input and the Output tones are typically at different frequencies.

**Intermodulation Distortion Parameters**

IMD parameters measure the **difference** in power level between the specified product tone and the main tones. These IMD parameters are calculated from the Tone Power measurements. IMD parameters can measure the odd-order product tones up to the 9th order, and the 2nd product tones, at the DUT Input or Output. For each specified product, the difference between the **Low** product and main tone, difference between the **High** product and main tone, and difference between the **Averages** of the product and main tone can be measured and displayed.

Swept IMD supports IMD parameters which are calculated as follows:

\[
\begin{align*}
\text{IM}x\text{Lo} &= \text{Pwr}x\text{Lo} - \text{PwrMainLo} \\
\text{IM}x\text{Hi} &= \text{Pwr}x\text{Hi} - \text{PwrMainHi} \\
\text{IM}x &= \text{Pwr}x - \text{PwrMain} \\
\text{IM}x\text{LoIn} &= \text{Pwr}x\text{LoIn} - \text{PwrMainLoIn} \\
\text{IM}x\text{HiIn} &= \text{Pwr}x\text{HiIn} - \text{PwrMainHiIn} \\
\text{IM}x\text{In} &= \text{Pwr}x\text{In} - \text{PwrMainIn}
\end{align*}
\]

where:

- \( x \) = the IM product of interest (2, 3, 5, 7, 9)
- **Avg** is implied if **Hi** or **Lo** is not stated
- **Output** is implied when **In** is not stated

[Learn how to select IMD parameters.](#)

**Intercept Point Parameters**

As the main tone output power increases (black arrow), output power in the specified product tone increases at a predictable, and steeper, rate (green arrow). At some point, the power in the product tone will be equal to the power in the main tone. The power level at which this occurs is known as the intercept point. Measuring this point directly is typically not possible. Therefore, it is calculated by measuring the main tone power and the specified product tone power.
The Swept IMD App can display either the DUT Input power or DUT Output power that is required to achieve the theoretical intercept point. This is called either Input Referred (IIP) or Output Referred (OIP).

This measurement can be made for the 2nd, 3rd, 5th, 7th, and 9th order intercept points. In addition, the measurements can be made for either the Low tone, the High tone, or the Average of the two. However, for the 2nd order intercept point, only Low and High tone parameters are supported; not Average.

Swept IMD supports Intercept Point parameters which are calculated as follows:

\[
\begin{align*}
\text{OIP}_{x \text{Hi}} &= \frac{\text{PwrMain} - \text{IM}_{x \text{Hi}}}{x-1} \\
\text{OIP}_{x \text{HiIn}} &= \frac{\text{PwrMain} - \text{IM}_{x \text{HiIn}}}{x-1} \\
\text{OIP}_{x \text{Lo}} &= \frac{\text{PwrMain} - \text{IM}_{x \text{Lo}}}{x-1} \\
\text{OIP}_{x \text{LoIn}} &= \frac{\text{PwrMain} - \text{IM}_{x \text{LoIn}}}{x-1} \\
\text{OIP}_x &= \frac{\text{PwrMain} - \text{IM}_x}{x-1} \\
\text{OIP}_x &= \frac{\text{PwrMain} - \text{IM}_x}{x-1} \\
\text{OIP}_{x \text{In}} &= \frac{\text{PwrMainIn} - \text{IM}_{x \text{In}}}{x-1} \\
\text{IIP}_{x \text{Hi}} &= \frac{\text{PwrMainIn} - \text{IM}_{x \text{Hi}}}{x-1} \\
\text{IIP}_{x \text{HiIn}} &= \frac{\text{PwrMainIn} - \text{IM}_{x \text{HiIn}}}{x-1} \\
\text{IIP}_{x \text{Lo}} &= \frac{\text{PwrMainIn} - \text{IM}_{x \text{Lo}}}{x-1} \\
\text{IIP}_{x \text{LoIn}} &= \frac{\text{PwrMainIn} - \text{IM}_{x \text{LoIn}}}{x-1} \\
\text{IIP}_x &= \frac{\text{PwrMainIn} - \text{IM}_x}{x-1} \\
\text{IIP}_x &= \frac{\text{PwrMainIn} - \text{IM}_x}{x-1}
\end{align*}
\]

where:

- \( x \) = the IM product of interest (2, 3, 5, 7, 9)
- \( \text{PwrMain} \) = average power of the main tones at the DUT Input or Output
- \( \text{IM}_x \) = main tone power - product tone power (from above IMD parameter)
- \( \text{Avg} \) is implied if \( \text{Hi} \) or \( \text{Lo} \) is not stated
Learn how to select IMD parameters.

Composite Triple Beat (CTB)

From the NCTA Standard, composite triple beat is defined as the modulation beat of the target channel signal caused by triple beat resulting from the nonlinear characteristic of the DUT. Composite triple beat is expressed as the ratio of the target channel signal level to the maximum mean level of beat components dispersed around the carrier of that target channel.

Swept IMD supports two parameters of this type:

- CTB is based upon an approximation for the number of beats in mid-band
- CTBE is based upon an approximation for the number of beats at the band edge.

The equations for these two parameters are as follows:

<table>
<thead>
<tr>
<th>Equation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mid-Band CTB (dB) = -2(Pi - Ps) + 6 + 10Log(3N²/8) + CTB Offset</td>
<td>Mid-band CTB equation</td>
</tr>
<tr>
<td>Band Edge CTBE (dB) = -2(Pi - Ps) + 6 + 10Log(N²/4) + CTB Offset</td>
<td>Band edge CTBE equation</td>
</tr>
</tbody>
</table>

Where:

- **Pi** = Output power level at the third order intercept point (dBm): OIP3 (Lo | Hi)
- **Ps** = One of the following values based upon the Composite Normalization Mode:
  - For **PDBM** or **PDBMV** mode, Ps = CompositeNormalizedCTBPower
  - For **Number of Carriers** mode, Ps = PwrMain (AVG) – 10Log(N/2)
  - For **None** mode, Ps = PwrMain (AVG)
- **CTB Offset** = Offset value for CTB calculation
- **N** = Total number of carriers.

Note: CTB Offset and N values can ONLY be set using SCPI or COM commands.

Learn how to select IMD parameters.

Composite Second-Order (CSO)

From the NCTA Standard, composite second order is defined as the modulation beat of the target channel signal caused by second order beat resulting from the nonlinear characteristic of the DUT. Composite second order beat is expressed as the ratio of the target channel signal level to the maximum mean level of beat components dispersed around 0.75 MHz and 1.25 MHz above and below the carrier of that channel.

Swept IMD supports a CSO parameter which is calculated as follows:

<table>
<thead>
<tr>
<th>Equation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CSO (dB) = (Pi - Ps) + 10Log(N) + CSO Offset</td>
<td>Composite second-order equation</td>
</tr>
</tbody>
</table>

Where:
• $P_i$ = Output power level at 2nd order intercept point: OIP2 (Lo | Hi)

• $P_s$ = One of the following values based upon the **Composite Normalization Mode**:

  - For **PdBm** or **PdBmV** mode, $P_s = \text{CompositeNormalizedCSOPower}$
  - For **Number of Carriers** mode, $P_s = \text{PwrMain (AVG)} - 10\log(\text{N}/2)$
  - For **None** mode, $P_s = \text{PwrMain (AVG)}$

• **CSO OFFSET** = Offset value for CSO calculation

• $N$ = Number of distortion products.

**Note:** **CSO OFFSET** and $N$ values can ONLY be set using **SCPI or COM commands**.

Learn how to select IMD parameters.

**Cross-Modulation Distortion**

From the NCTA Standard, cross modulation is defined as the distortion that causes modulated carrier components of undesired channels to amplitude-modulate the target channel carrier due to the nonlinear characteristic of the unit under test. Cross modulation distortion is expressed as the ratio of the target channel carrier level to the level of modulated components of the carrier of the target channel resulting from modulated signals of undesired channels.

Swept IMD supports an XMOD parameter which is calculated as follows:

$$XMOD=-2(P_i-P_s)+6dB+20\log(N)$$

Where:

• $P_i$ = Output power level at third order intercept point: OIP3 (Lo | Hi)

• $P_s$ = Power level of each carrier: PwrMain (AVG)

• $N$ = Total number of carriers.

Make Cross Modulation settings using **SCPI or COM commands**.

**How the IMD Application Works**

The IMD App. requires The following diagram illustrates how the PNA is configured to generate the two main tones. This shows a PNA-X with dual sources and the internal combiner. A 2-port or 4-port N522x model can also be used. Learn how to **Configure External Source and Combiner**.
2-port PNA-X generates the f1 and f2 main tones.

Depending on the specified parameters and sweep type, the sources and receivers are tuned to the appropriate frequencies in order to measure all of the required main and product tone powers. For example, an IM3 parameter requires the measurement of both main tones, and the 3rd order High and Low tone powers.

The Narrowband IF path is used for IMD measurements to help reduce spurious responses. Because the narrowband filter has a bandwidth of about 28 kHz, using an IFBW greater than 30 kHz does nothing to improve measurement accuracy. Learn how to set IFBW for IMD.

Limiting Stimulus Settings and Out of Range Product Tones

Because the main tones are generated by the PNA internal sources and external sources, the frequencies of the main tones must always be within the frequency range of the PNA or external source. Sweep parameter values are adjusted when necessary to ensure that f1 and f2 frequencies are within these limits.

However, the PNA DOES allow you to make settings that cause the selected IM products to fall outside the frequency range of the PNA. For example, with the main tones at 10 MHz and 15 MHz, the PNA will allow you to select the parameter IM3Lo (3rd low side product tone). However, the frequency of this product will be at 2f1-f2 or 5 MHz, which is below the frequency range of the PNA. In these cases, the trace data is set to zero, which converts to -200 dB in Log Mag format.

Limited Number of Acquisitions

The total number of acquisitions per sweep can not exceed 32,001 points. The number of acquisitions is determined by multiplying the number of trace points, by the number of tones frequencies, then by 2 (for both Input and Output frequencies). The PNA will automatically reduce the number of trace points to ensure the total number of acquisition points does not exceed 32,001.

How an IM Spectrum Channel Works

Before reading this topic, you should become familiar with IMD Concepts.
The IM Spectrum channel provides a traditional spectrum analyzer view of the intermodulation distortion behavior of a device. Unlike the Swept IMD channel, the main tones (F1 and F2) are fixed while the receiver is swept over a frequency range of interest in order to generate a display as shown below.

**IM Spectrum trace (bottom) with Swept IMD traces (top)**

A typical spectrum analyzer does NOT have a signal source. This one does. The signal source, or stimulus, settings for the F1 and F2 main tones can be set in either the IM Spectrum channel or the Swept IMD channel. These settings include the frequencies and power levels of the main tones.

**Receiver Settings**

The settings for the IM Spectrum receiver can be set ONLY in the IM Spectrum channel. These settings include how many tone products to view - which determines the center and frequency span - and the Resolution Bandwidth.

You can choose from several Resolution Bandwidths which also determines the number of data points used in the channel. The higher the Res BW, the fewer the number of data points. The formula for determining the number of points is:

$$N = \frac{\text{Span}}{\text{ResBW}} \times 3$$

The IM Spectrum channel performs multiple measurements for each data point in order to reject unwanted images which are generated by the PNA internally. This provides a high degree of confidence that signals captured in a trace are real and are not spurious responses generated in the measurement process.

**IM Spectrum Parameters**

You can select from three different IM Spectrum parameters.
1. The tones OUT of the DUT (default parameter).

2. The tones IN to the DUT (to be sure that the input signals are pure). **NOT supported** in IMx Spectrum

3. Reflected tones off the DUT input. **NOT supported** in IMx Spectrum

Learn how to select IM Spectrum parameters for Amplifiers or Converters.
Learn all about IM Spectrum for Amplifiers or Converters.
See list of all IMD topics.

Last Modified:

12-Mar-2009  Added Tone Gain (8.5)

9-Mar-2009  MX New topic
Features and Limitations

Create a Swept IMD Measurement

- Frequency tab
- Power tab
- Configure tab
- How to specify IMD Parameters

IMD Calibration

Saving Swept IMD Power Data

Other IMD (Opt 087) Topics

Features and Limitations

See requirements.

Features

- Fast and easy setup for the measurement of a variety of distortion related parameters up to 9th order.
- Measurement at both input and output of a DUT.
- Supports a variety of sweep-modes for the main-tones: center-frequency linear and segment sweep, tone-separation sweep, power sweep, or CW sweep (fixed main-tone).
- Make very fast, accurate measurements using the PNA sources with high-power, high linearity, and low harmonics.
- Supports calibration and correction of Swept IMD parameters.
- Independently set IFBW for measuring main-tones versus product tones.

Limitations

- 2-port non-frequency converters ONLY. For frequency converters, use Swept IMDx.
- DUT to PNA port mapping is limited. Port selections are made on the Power tab. Learn more.

The following features are NOT available with Swept IMD:
- **Number of points limited to 20,001**
- Analog Sweep (Stepped sweep mode only)
- Log frequency sweeps
- Unratioed receiver measurements (A, B, R)
- ECAl User Characterization
- Time Domain
- Balanced measurements
- Save Formatted Citifile data.
- Save SnP data.
- External sources
- Interface Control
- Port extensions
- Some Fixturing Features
- External Test Set Control (Option 551)
- Integrated Narrowband or Narrowband Pulse App
- See Frequency limitations in a Swept IMD channel.

### Create a Swept IMD Measurement

1. On the PNA front panel, press **Meas** then [Measurement Class]
2. Select Swept IMD, then either:
   - **OK** delete the existing measurement, or
   - **New Channel** to create the measurement in a new channel.
3. A Swept IMD measurement is displayed. To select additional parameters to display, click **Response**, then **Measure**, then select a parameter from the list.
How to start the Swept IMD Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

Using front-panel HARDKEY [softkey] buttons

1. Press FREQ
2. then [Swept IMD Setup]

Using a mouse with PNA Menus

1. Click Stimulus
2. then Frequency
3. then Swept IMD Setup

Frequency tab - IMD Setup - dialog box help

Configures the Sweep Type and frequency range for SweptIMD and Swept IMDX measurements.

Sweep Type and Sweep Settings
**Sweep fc (center frequency)**

Maintaining a constant tone spacing (Fixed DeltaF) and tone power, fc is swept from Start fc to Stop fc. Center Frequency can also be specified as Center fc and Span fc.

At each fc, the receivers are tuned to all of the required frequencies to measure the power of the appropriate tones.

---

**Swept DeltaF (tone spacing)**

The specified fc (center frequency) and tone power is held constant. The tone spacing is increased from Start DeltaF to Stop DeltaF in the specified number of points.

---

**Power Sweep**

The main tone frequencies are specified as either f1 and f2, or as fc and DeltaF. These frequencies are held constant while the power of each main tone is varied from the Start-Power to Stop-Power in the specified number of power points. The power of each tone can be set (on the Power tab) individually or as a pair by checking Coupled Tone Power.

---

**CW**

The main tone frequencies and power levels are held constant. Measurements are taken for the specified Number of Points. The X-axis is number of points.

---

**Segment Sweep fc (Swept IMD ONLY)**

Same as Sweep fc except that the center frequencies are constructed using the standard segment table. Learn how.

---

**LO Power Sweep (Swept IMDX ONLY)**

The main tone frequencies and power levels are held constant. Measurements are taken for the specified number of points. The X-axis is LO Power.

---

**Segment Sweep Notes: (Swept IMD ONLY)**

- The segment table shown on the dialog is 'READ-ONLY'.
- Learn how to Create and edit the Segment Sweep table.
- Independent IFBW and Power are NOT available.
- X-axis point spacing is available beginning with A.09.10.

**Number of Points** Enter the number of data points for each sweep. See Limited Number of Acquisitions.

**IFBW**
The Narrowband IF path is used for IMD measurements to help reduce spurious responses. Because the narrowband filter has a bandwidth of about 28 kHz, using an IFBW greater than 30 kHz does nothing to improve measurement accuracy.

**Main Tone and IM Tone IFBW**  IF Bandwidth is specified separately for the main tones (f1 and f2) and for the intermodulation tones. This allows the higher-power main tones to be accurately measured at a higher - and faster - IFBW, while the lower-power product tones to be accurately measured a lower - and slower - IFBW.

**Reduce IF BW at Low Frequencies**  On the PNA, the trace noise becomes worse below 748 MHz. This is especially obvious between 10 MHz and 45 MHz. When this box is checked, the PNA uses a smaller IF Bandwidth than the selected value at frequencies below 748 MHz. Learn more about the selected values.

---

**Power tab - IMD Setup** -dialog box help

Configures RF power and Power Sweep settings for IMD measurements.

**Power ON (All channels)**  Check to turn RF Power ON or clear to turn power OFF for all channels.

**DUT Input Port**

- **Input Port**  Choose Port 1 or Port 3. An external combiner is required for Port 3.
- **Source Attenuator**  Specifies the port 1 attenuator. This attenuator affects the range of available power into the DUT Learn more about Source Attenuation.
- **Receiver Attenuator**  This attenuation setting protects the A receiver from damage.

**DUT Output Port**

- **Output Port**  Choose Port 2 or Port 4 (with limitations).
- **Source Attenuator**  This setting is used to improve the load match at the DUT output. Select 0 dB for power levels up to 10 dBm, and increase by 10 dB for every 10 dBm more output power.
**Receiver Attenuator** Specifies the attenuator setting for port 2. When the power into the receiver test port is around +10 dBm, the PNA receiver may be in compression. However, with receiver attenuation, lower input power levels may become too noisy to make accurate power measurements. In this case, lower IFBW for the IM tones to reduce noise. [Learn more about Receiver Attenuation.]

**Tone Powers**

**Coupled Tone Power** Check to set the same power level for each main tone using the f1 Power setting. Clear to set different f1 and f2 power levels.

**Fixed f1 Power** Specify the power level for f1 at either the DUT input or output depending on the Power Leveling setting. Choose a value between -30 dBm and +30 dBm. When “Coupled Tone Power” is checked, power is set for both f1 and f2 tones.

**Fixed f2 Power** Available when Coupled Tone Power is NOT checked. Specify the power level for f2 at either the DUT input or output depending on the Power Leveling setting. Choose a value between -30 dBm and +30 dBm.

**Start, Stop, and Step f1 and f2 Power** Available when Power Sweep is selected on the Frequency tab. Sets the Start and Stop power levels for f1 and f2, either individually or together with Coupled Tone Power checked.

**Power Leveling**

Because the gain of the amplifier can be different for the f1 and f2 tone frequencies, you can set tone power at either the input of the DUT (default setting) OR the output. The one that you do NOT select will NOT be equal or flat across the frequency span.

**Set Tone Power at Output** The specified f1 and f2 power levels are set at the DUT output. Receiver Leveling is used to accurately set the specified power level of each tone within the tolerance value that is set in the [Receiver Leveling dialog](#).

**Set Tone Power at Input** (Default) The specified f1 and f2 power levels are set at the DUT input.

  **Equalize at Output** Available when “Coupled Tone Power” is checked AND "Set Tone Power at Input" is selected.

  - When cleared (default setting), input power level accuracy is based ONLY on the source power cal that is performed during the IMD cal.
  - When checked, the average of the specified tone powers is set at the input, but amplifier gain differences are compensated for so that the output tones are equal and flat across the frequency span. Receiver Leveling is used to set EACH output tone to within the tolerance value that is set in the [Receiver Leveling dialog](#). This could result in the output tone power levels being different by twice the tolerance value.

Open Loop leveling should only be used when doing [Wideband Pulse measurements](#). Set to Open Loop on the [Power and Attenuators dialog](#).

**Path Configuration** click to launch the [RF Path Configuration](#) dialog.

**Highlighted Note: RF2 tone power offset to compensate for combiner loss.**

This message appears when the f2 tone is being supplied by an external source. The tone power has been increased on the external source to compensate for loss through the internal combiner. For example, if the tone power at the DUT should be 0 dBm, the power out of the source will be about 15 dB higher.
To accommodate single-source PNA models, an external source can be used for the f2 tone. Learn how to configure an external source and combiner to make Swept IMD and IMDx measurements.

**f1** Always uses PNA internal source 1.

**f2** Select a source to be used for the f2 tone. This selection is available when an external source is configured and the **Active** box is checked on the **External Source Configuration dialog**.

**Buttons**

**Add Source** Click to configure an external source using the **External Source Configuration dialog**.

**Path Configuration** Click to launch the **Path Configuration dialog** (PNA-X models only).

### How to add IMD Parameters

<table>
<thead>
<tr>
<th><strong>Using front-panel HARDKEY [softkey] buttons</strong></th>
<th><strong>Using a mouse with PNA Menus</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>TRACES</strong></td>
<td>1. Click <strong>Trace/Chan</strong></td>
</tr>
<tr>
<td>2. then <strong>[New Trace]</strong></td>
<td>2. then <strong>New Trace</strong></td>
</tr>
<tr>
<td>3. then select a parameter</td>
<td>3. then select a parameter</td>
</tr>
</tbody>
</table>

**IMD New Trace** dialog box help
Use this dialog to select IMD and IMDx parameters to measure and display.

Up to five parameters can be selected at a time, then click **Apply** to create those traces. Then select more without closing the dialog. There is no limit to the number of traces and windows allowed in the PNA.

**Note:** Calculations are NOT performed to determine if the frequency of a selected intermod (Order) product will be within the frequency range of the PNA. Measurements that fall outside of the frequency range of the PNA are displayed as -200 dB.

**Param Name** This name is built from the selected **Type**, **Tone Select**, **Order**, and **Measure At** settings. Once built, check to measure this parameter.

**Type**

Select the type of Swept IMD measurement to make.

The characters below (in parenthesis) are used in the Param Name.

- **Tone Power (Pwr)** Measures the absolute power of the specified (ordered) tone. [Learn more](#).

- **IMD Relative to Carrier (IM)** Measures the power of the specified (ordered) tone relative to the power of the f1 or f2 tone. [Learn more](#).

- **Input Referred Intercept Point (IIP)** From DUT measurements, calculates the theoretical power level at which the specified (ordered) intermod product will be the same power level as the carrier at the output of the DUT. The input power to the DUT at which this output power occurs is reported and displayed. [Learn more](#).

- **Output Referred Intercept Point (OIP)** From DUT measurements, calculates the theoretical power level at which the specified (ordered) product will be the same power level as the main tone at the output of the DUT. This value is reported and displayed. [Learn more](#).

- **CTB Band-Edge Distortion (CTBE)** Composite ‘Triple Beat’ Distortion - 3rd order DUT OUT only. [Learn more](#).
- **CTB Mid-Band Distortion (CTB)**  Composite 'Triple Beat' Distortion - 3rd order DUT OUT only. [Learn more.]
- **CSO Distortion (CSO)**  2nd order DUT OUT only. [Learn more.]
- **XMOD 3rd Order Crossmod (XMOD)**  3rd order DUT OUT only. [Learn more.]
- **Tone Gain (ToneGain)**  From tone power measurements, calculates the Output Tone power / Input Tone power for the specified tones. [Learn more. The Input and Output tones are different frequencies.]

**Tone Select**
Select the tone (High, Low, or Both) to be measured and displayed ([See image]).

- **High**  Measure and display the power of the specified (ordered) tone on the High side of the main tones.
- **Low**  Measure and display the power of the specified (ordered) tone on the Low side of the main tones.
- **Avg**  Measure the specified (ordered) tones on both the High and Low sides. Then calculate and display the Average power level.
- **Max**  Measure the specified (ordered) tones on both the High and Low sides. Then find and display the Maximum power level.
- **Min**  Measure the specified (ordered) tones on both the High and Low sides. Then find and display the Minimum power level.

Used to build the parameter name:
- **Hi, Lo, Max, and Min** are appended to the Param Name when selected.
- Nothing is appended to the Param Name when **Avg** (default setting) is selected.

**Important Notes - 2nd-order products**

- 2nd-order products are likely to be outside of the PNA frequency range. Trace data will show all will show **all zeros** (linear) or -200 dB (log magnitude).
- If either the High or Low side falls outside the frequency range of the PNA, then **Avg** is NOT allowed.
- When displaying 2nd-order traces, **Avg** is NOT allowed, even when both the Lo and Hi products are displaying valid data. This is because 2 Low and 2 High products are usually very different from one another.
- When performing a calibration that is meant to include 2nd-order products, be sure Include 2nd Order Products is checked in the first Calibration dialog box.

**Order**
Specify the intermodulation product to measure. Choose from 1, 2, 3, 5, 7, 9.
- **Main** is appended to the Param Name when 1 is selected.
- Otherwise, the tone number is appended to the parameter name.

**Measure At**

Measure the selected parameter at either:

**DUT Input**

- **In** is appended to the Param Name (ex: PwrMainIn).
- The input port reference receiver is used to measure the fundamental tones and the required products.

**DUT Output**

- Nothing is appended to the Param Name (ex: PwrMain).
- The output port measurement receiver is used to measure the fundamental tones and the required products.

**IMD Calibration**

**Overview**

1. At the first page of the IMD Cal Wizard, you tell the calibration routine the frequencies at which the calibration is to be performed. Optionally, you can choose to perform the source power cal at only the center frequency midway between the main tones. For IMDx ONLY, you can also choose to perform a source power cal of the LO source.

2. The PNA calculates an array of source and receiver frequencies that incorporate all the main tone frequencies (low and high) and all the product tone frequencies.

3. Using a power meter at port 1, a source and receiver calibration is performed to calibrate the R1 reference receiver to be a fast and ‘tunable’ power meter.

4. The R1 reference receiver is then used to perform a source power cal of the main tone frequencies: first the Source 1 / Low tone then the Source 2 / High tone. Both sources are left ON while each tone is measured in order to duplicate the impedance match under which the measurement will be performed.

5. Then a standard 2-port SOLT cal is performed at all frequencies using either an ECal module or mechanical standards. The 2-port cal is used to correct the source calibration R1 tracking terms for the match of the power sensor. It is also used to transfer the R1 tracking term to the B receiver.

**Notes**

- If the main tone frequencies change but are within the frequency range in which the calibration was performed, then the calibration becomes interpolated C*. This can occur by changing the
start/stop/center/span frequencies, the number of points, or the sweep type. Learn more about Interpolation.

- **Receiver calibrations** that are performed in a standard channel can be applied to a Swept IMD channel. However, **Source Calibrations** can NOT be applied.

**See Also**

IM Spectrum Calibration

---

**How to start a Swept IMD Calibration**

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CAL**
2. then **[Start Cal]**
3. then **[Cal Wizard]**

**Using a mouse with PNA Menus**

1. Click **Response**
2. then **Cal Wizard**

---

**Select Tone Products - IMD Cal** dialog box help

**Calibration Mode**

Match corrected Response Cal  This selection performs a full IMD calibration as described in the above Calibration 'Overview'.

**Response ONLY (Normalization)** This IMD Cal does NOT correct for the mismatch of the power sensor. Choose this if you have a test configuration that does not easily accommodate making match measurements. Instead of a standard 2 port SOLT cal (Step 5 above), only the transmission tracking term is measured and used to transfer the R1 receiver tracking term (produced by the power sweep) to the B receiver.

**Note:** For the Response (Normalization) Cal, it is assumed that a zero-length THRU standard is being used to connect port 1 to port 2. If an adapter is used, there is NO compensation for delay or loss of the adapter. This can NOT be changed.

---

3434
Tone Power Cal

**Calibrate only at center frequencies** The source power cal portion is performed at only the center frequency, which is midway between the main tones. This cuts the source power calibration time (the slowest part of the calibration) in half. The measurement is interpolated although the C* annotation is not shown in the status bar.

**Calibrate at all frequencies** The source power cal portion is performed at all main tone frequencies.

**Enable LO Power Calibration** (IMDx ONLY) Check to perform a standard power calibration of the LO source as part of the calibration process.

Select Product Tones

**Max Product** Select the highest product that you will be measuring. The low and high frequencies for that product, and all lower 'odd' order products will be calibrated. For example, when 5th Order Product is selected, the frequencies for the Main Tones, and the Low and High order products for the 3rd and 5th order products will be included in the calibration.

**Include 2nd Order Products** Check to calibrate the 2nd-order products in the frequencies to be calibrated. The frequencies of these products are usually far from the main tones.

Select DUT Connectors and Cal Kits - IMD Cal dialog box help

If **Response Cal Only** is selected on the previous page, click View/Modify to change the Source Cal settings, or click Next> to continue.

Otherwise, this is a standard PNA Cal Wizard page except for the following:

**Power Sensor** When the power sensor connector is not the same type as the DUT Port 1 connector, then for optimum accuracy, an extra cal step is required to measure the adapter that is used to connect the power sensor to the port 1 reference plane during the Source Power Cal. Specify the connector type and the Cal Kit that will be used for that step. Select **Ignored** to NOT compensate for the adapter.

**Modify Cal** Check, then click Next, to Modify Cal (Standards AND Thru Method).

**View/Modify Source Cal Settings** Click to launch the Source Cal Settings (for Apps) dialog.

Learn more about IMD Calibration
Power Level at which to perform the Port 1 Source Power Cal.

It is usually best to perform the Source Power Cal at 0 dBm because the power sensor is calibrated at that level.

However, if a component is used between the PNA source and the calibration reference plane, then adjust the power level so that the power at the sensor is about 0 dBm if possible.

Learn more about IMD Calibration

The remaining dialog pages are the same as the standard Cal Wizard.

Saving Swept IMD Power Data

Swept IMD power data, Log Mag ONLY, can be saved to a *.csv file. This data type can be read by spreadsheet programs, such as Microsoft Excel. Data from the last complete sweep is saved to the specified *.csv file.

How to save Swept IMD (and IMDx) data

With a Swept IMD or IMDx channel active...

1. Press Save
2. then [Save Data As]
3. then, File Type = IMD Sweep Data (*.csv)

Using a mouse with PNA Menus

1. Click File
2. then Save Data As
3. then File Type= IMD Sweep Data (*.csv)

Notes:

- For every tone, six power levels are saved in this order: OUT Avg | OUT Lo | OUT Hi | IN Avg | IN Lo | IN Hi
- Power levels for the Main tones are always saved, regardless of the active measurement.
- All tones that are displayed are also saved. For example, any displayed 3rd order tone parameter causes the
3rd tone power levels to be saved.

- Only tone powers are saved. Calculated parameters, such as IMD Relative to Carrier (IM) are NOT saved.
- If calibration is turned ON when the file is saved, then all data is calibrated. Otherwise, raw data is saved.

This image shows the 6 Main tone power levels that are always saved.

In the power parameter labels, **Output** and **Avg** are implied as in the parameter selection. For example: **PwrMain** = Average Output power of the main tones.

---

### Last Modified:

- 5-May-2011  Added tone power leveling and Min/Max parameters
- 27-Apr-2011  Removed Copy Channels limitation
- 1-Apr-2011  Added note to 'Measure At'
- 22-Feb-2011  Enhanced 2nd order notes
- 12-Apr-2010  Removed some fixturing limitations
- 19-Nov-2009  Added X-axis point spacing (9.1)
- 11-Aug-2009  Added limited port mapping (A.09.00)
- 13-Jul-2009  Added Tone select notes
- 27-Feb-2009  Added data save and modified cal for IMDx
- 14-Aug-2008  New topic
**IM Spectrum for Amplifiers (Opt 087)**

Intermodulation Spectrum measurements can be made independently, or coupled with Swept IMD measurements. They are a distinct measurement class. Therefore, IM Spectrum measurements are always made in a separate channel.

- **Requirements and Limitations**
- **Create an IM Spectrum Channel**
- **Select IM Spectrum Parameters**
- **Start the IM Spectrum Setup dialog**
  - **Frequency tab**
  - **Power tab**
  - **Tracking tab**
  - **Configure tab**
- **Calibration**

**See Also**

- **IM Spectrum Concepts**
- **IMx Spectrum for Converters**
- **Swept IMD Measurements**
- **Programming commands**

**Requirements and Limitations with IM Spectrum**

The following PNA features are NOT available in an IM Spectrum channel:

- **Unratioed receiver measurements** (A, B, R)
- **ECal User Characterization**
- **FOM** or **FCA**
- **Time Domain**
- **Balanced measurements**
- **Save Formatted Citifile** data.
Create an IM Spectrum Channel

An IM Spectrum channel can be created independently from a Swept IMD channel or coupled with the stimulus settings of an existing Swept IMD measurement.

To create an independent IM Spectrum channel

1. On the PNA front panel, press **Meas** then **[Measurement Class]**

2. Select **IM Spectrum**, then either:
   - **OK** delete the existing measurement, or
   - **New Channel** to create the measurement in a new channel.

To create an IM Spectrum channel from an existing Swept IMD channel

1. With a Swept IMD channel active, press the **MARKER** key on the front panel.

2. Move the marker to the data point of interest. It can be moved later.

3. Press **[Marker Functions]** then **[Marker -> IM Spectrum]**

This creates or configures an existing IM Spectrum channel based upon the configuration of the Swept IMD channel.

Select IM Spectrum Parameters
**How to add IM Spectrum traces**

With the IM Spectrum channel active and a **Tr1 Output** trace displayed:

### Using front-panel HARDKEY [softkey] buttons

1. Press **TRACES**
2. then [New Trace]
3. then select a new parameter

### Using a mouse with PNA Menus

1. Click **Trace/Chan**
2. then **New Trace**
3. then select a new parameter

---

**New Trace - IM Spectrum -dialog box help**

<table>
<thead>
<tr>
<th>Output</th>
<th>Port 2, Incident</th>
</tr>
</thead>
<tbody>
<tr>
<td>□ Input</td>
<td>Port 1, Output</td>
</tr>
<tr>
<td>□ Reflection</td>
<td>Port 1, Incident</td>
</tr>
<tr>
<td>□ b3</td>
<td>Port 3, Incident</td>
</tr>
<tr>
<td>□ b4</td>
<td>Port 4, Incident</td>
</tr>
<tr>
<td>□ b2−2</td>
<td>Port 2, Incident</td>
</tr>
<tr>
<td>□ b2−1</td>
<td>Port 2, Incident</td>
</tr>
<tr>
<td>□ b2+1</td>
<td>Port 2, Incident</td>
</tr>
<tr>
<td>□ b2+2</td>
<td>Port 2, Incident</td>
</tr>
<tr>
<td>□ b4−2</td>
<td>Port 4, Incident</td>
</tr>
<tr>
<td>□ b4−1</td>
<td>Port 4, Incident</td>
</tr>
<tr>
<td>□ b4+1</td>
<td>Port 4, Incident</td>
</tr>
<tr>
<td>□ b4+2</td>
<td>Port 4, Incident</td>
</tr>
</tbody>
</table>

Check the IM Spectrum measurement to add to the display.

- **Output** View signals OUT of the DUT and into PNA port 2 (B receiver).
- **Input** View signals IN to the DUT (R1 receiver). Use this when measuring IM product frequencies to determine the power level of spurious signals into the DUT at those frequencies.
- **Reflection** View signals reflected off the DUT input and back into PNA port 1 (A receiver)

---

**IM Spectrum Setup Dialog**
How to start the IM Spectrum Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

With an IM Spectrum measurement active:

### Using front-panel HARDKEY [softkey] buttons

1. Press **Sweep**
2. then **IM Spectrum Setup**

### Using a mouse with PNA Menus

1. Click **Stimulus**
2. then **Frequency**
3. then **IM Spectrum Setup**

---

**Programming Commands**

---

**Frequency tab - IM Spectrum dialog box help**

![IM Spectrum Setup dialog box](image)

**Note:** The number of data points in an IM Spectrum channel = 3 * SPAN / ResBW. This can NOT be changed directly.

Configures the Sweep Type and frequency range for IM Spectrum measurements.

**Sweep Type**

Provides several methods to tune the IM Spectrum receivers (NOT stimulus) to view the power spectrum of arbitrary frequency ranges or various distortion products.

- **Linear**  Allows the start/stop or center/span receiver frequencies to be set arbitrarily for f1 and f2. Enter the Response Settings below.

- **2nd Order**  Couples the receiver frequency range to f1 and f2 to provide a convenient means of observing the spectrum surrounding ONLY the high-side 2nd order harmonic where Center = (f1 + f2), Span = 3 * (f2 – f1). **Note:** The center frequency is NOT set on the main tones.

- **3rd Order**  Couples receiver frequency range to f1 and f2 to provide a convenient means of observing the spectrum surround the main-tones including the 3rd Order products where Center = (f1 + f2) / 2, Span = 4 * (f2 – f1).

- **Nth Order**  Couples receiver frequency range to f1 and f2, providing a convenient means of observing the
spectrum surrounding the main-tones for an arbitrary span where Center = \((f_1 + f_2) / 2\), and Span = \(N \times (f_2 - f_1)\). This allows you to set the span arbitrarily, but have the center frequency track the main tone frequencies.

**Resolution BW**

The IM Spectrum channel utilizes a set of Gaussian filters instead of the standard PNA IF filters in order to provide similar behavior to traditional spectrum analyzers.

The comprehensive list of filters that are available for IM spectrum are: 3 MHz, 1 MHz, 600 kHz, 300 kHz, 150 kHz, 100 kHz, 60 kHz, 10 kHz, 3 kHz, and 1 kHz.

**Note:** The 10 kHz Res BW filter can generate image signals which are not a product of the DUT. You can verify the integrity of a questionable signal by switching to the 3 kHz or 60 kHz filter and looking for the image signal in the same location.

Not all of these filters are available for all measurements. Narrower filters are available for use with narrower frequency spans, and wider filters are available for wider spans.

**Stimulus Settings**

Allows configuration of the main-tone frequencies. Available ONLY when Tracking is OFF.

- \(f_c\) (main-tone center frequency) = \((f_1 + f_2) / 2\)
- \(\Delta f\) (main-tone frequency separation) = \((f_2 - f_1)\).
- \(f_1\) = Low-side main-tone frequency
- \(f_2\) = High-side main-tone frequency

**Response Settings**

Allows configuration of the frequency range to sweep the receivers in terms of start/stop or center/span. Available for Linear sweep types ONLY.

- **Start Spectrum** = first frequency point of the power spectrum sweep
- **Stop Spectrum** = last frequency point of the power spectrum sweep
- **Center Spectrum** = \((\text{Start Spectrum} + \text{Stop Spectrum}) / 2\)
- **Span Spectrum** = \((\text{Stop Spectrum} - \text{Start Spectrum})\)
Learn about this dialog

These selections are NOT available when there is no Swept IMD channel.

Allows the IM Spectrum channel to use (track) the stimulus (Main Tone) settings of an existing Swept IMD channel.

**Tracking Enable**  Check to use the frequency and power stimulus settings from the specified IMD channel. When enabled, stimulus settings on the Frequency Tab are disabled and ALL stimulus settings, such as frequencies, power and attenuator settings, and calibration, are copied from the Swept IMD channel to the IM Spectrum channel.
In the top Swept IMD window the main tones are swept from 4 GHz to 6 GHz with some specified delta F tone separation. The marker is on the center data point at 5 GHz.

The bottom IM Spectrum window center frequency is the same as the above marker: 5 GHz, but it has a much narrower frequency range of +/- 500 kHz. The IM Spectrum channel sets the receiver to see the two main tones, plus the third, fifth, and seventh-order products.

**Step Mode**

When tracking is enabled, set the method by which the IM spectrum measurement tracks the IMD channel.

**Manual Step** When selected, IM Spectrum measurements occur at only the specified IMD channel data point. **Stimulus Point** specifies the data point, by number, in the Swept IMD channel which has the stimulus settings to use for the IM Spectrum sweep.

**Automatic Step** When selected, causes the IM Spectrum channel to sequentially setup each of the stimulus conditions through which the Swept IMD channel sweeps the DUT. Each sweep of IM Spectrum is performed using the next set of stimulus conditions.

For example, in the above image, the first Swept IMD data point is at 4 GHz. The first IM Spectrum sweep uses a center frequency of 4 GHz. The following IM Spectrum sweep would be at the second Swept IMD data point or 4.01 GHz, and so forth. After the last data point in the sweep is reached, the IM Spectrum channel begins again at the first Swept IMD data point. The only indication that Tracking is enabled is in Automatic mode, you can see the center frequency increment with each IM Spectrum sweep.
Configure tab- IM Spectrum Setup - dialog box help

To accommodate single-source PNA models, an external source can be used for the RF2 tone. Learn how to configure an external source and combiner to make Swept IMD, IMDx, IM Spectrum, and IM Spectrum for Converters measurements.

RF1  Always uses PNA internal source 1.
RF2  Available for selection when an external source is configured and Active.
Add Source  Click to configure an external source using the External Source Configuration dialog.
Path Configuration  Click to launch the Path Configuration dialog (PNA-X models only).

Calibration

A calibration of the IM Spectrum channel is NOT performed using a calibration wizard.

An IM Spectrum channel is calibrated from a Cal Set that is used on a Swept IMD channel. The Cal Set can be applied to the IM Spectrum channel using the Manage Cal Sets dialog (Learn how) or from the Marker =>IM Spectrum softkey (Learn how).

However, a Swept IMD channel with Sweep Type = Power Sweep can NOT be applied to a IM Spectrum channel. This is because a Cal Set for power sweep contains only a CW frequency and the IM Spectrum channel requires a swept frequency range. Zero Span is not supported in an IM Spectrum channel.

A calibrated IM Spectrum trace corrects the source and receiver power level accuracy of the displayed Tones. See Swept IMD Calibration

Last Modified:

27-Apr-2011  Removed Copy Channels limitation
16-Sep-2009  Fixed calibration
11-Aug-2009  Added limited port mapping
18-Aug-2008  MX New topic
**Swept IMD for Converters (IMDx)**

- IMDx Concepts
- Requirements and Limitations
- How to Configure your Hardware
- Create a Swept IMDx Measurement
- Valid Mixer Configuration / Sweep Type Combinations
- Setup Dialog
- How to specify IMDx Parameters
- IMDx Calibration
- Saving IMDx Data (Swept IMD topic)

**Other IMD Topics**

**IMDx Concepts**

Conceptually, Swept IMD for Converters (IMDx) is like [IMD for Amplifiers](#), except that there are two sets of products on the DUT output.

This image shows the simplest measurement configuration: the Input, LO, and Output are all CW frequencies.

- **DUT INPUT** - two fundamental tones (f1 and f2)
- **DUT LO** - a single frequency.
• **DUT OUTPUT** - two sets of frequencies

With IMDx, you can view EITHER the High side or Low side products; NOT BOTH.
Make this selection on the **Mixer Frequency tab**.

- High side (Input **PLUS** LO and all ordered products)
- Low side (Input **MINUS** LO and all ordered products)

**Requirements and Limitations**

Swept IMDx requires **Swept IMD** (Opt 087) and **FCA** (either Opt 082 or 083).
The following PNA features are **NOT** available with Swept IMDx:

- Analog Sweep (**Stepped sweep** mode only)
- **Log frequency** sweeps
- Unratioed receiver measurements (A, B, R)
- **ECal User Characterization**
- **Time Domain**
- **Balanced measurements**
- Save **Formatted Citifile** data.
- Save SnP data.
- **Interface Control**
- **Port extensions**
- **Some Fixturing Features**
- **External Test Set Control** (Option 551)
- **Integrated Narrowband** or **Narrowband Pulse App**
- **See Frequency Limitations**

**Note:** Beginning with A.09.00, **Embedded LO** measurements are allowed in IMDx and IMSpectrum. Configure the measurement as you would with SMC. [Learn how](#).

**How to Configure your Hardware**

The PNA is extremely versatile, and can be configured in many ways to make IMDx measurements. While not all conceivable configurations are documented here, a few of the most common examples are provided to show the basic concepts.
**DUT Configuration**

- The DUT Input must be connected to PNA Port 1 which supplies the f1 and f2 tones.
- The DUT Output must be connected to PNA Port 2 which uses the PNA B receiver.
- See LO Source configuration below.

**Source Configuration**

Three sources are required to make IMDx measurements: Two sources are PNA internal; the third is an external source.

- **F1 tone** - Must come out PNA Port 1  Default is Src 1
- **F2 tone** - Must come out PNA Port 1 through the internal combiner. This source can come from an internal or external source.  Default is the internal Src 2.
- **LO** - Can come from internal or external source. Default is Not controlled, set to 0 Hz.

  - If using the internal Src 2 source, the f2 tone must come from an external source through the rear-panel.
  - If using an external source, it can be connected directly to the DUT, or through the PNA Port 3 or Port 4 using the Path Configurator.

**Configuration Examples**

Three configurations are shown below to illustrate how to use the Path Configurator to manually make switch settings.

IMDx and IMx Spectrum channels are ALWAYS configured using the default configuration switch settings shown in the image below. This is NOT the same default configuration that is used for an S-parameter channel. A manual switch setting is required whenever a custom configuration is used.

The manual switch setting, which must be done every time an IMDx or IMx Spectrum channel is created, can be saved using the Store button in the Path Configurator. Then save the entire IMDx setup as an Instrument State. This will load the custom Path Configuration when the Instrument State is recalled.
1. Default Configuration  No manual switching required.

- f1 - Internal Src1
- f2 - Internal Src2 through combiner in Normal
- LO - Connect external source directly to the DUT LO.

2. External LO though Port 3 or Port 4  (4-port PNA only)
Use this configuration to monitor LO power using R3 or R4. (Future parameters)

- f1 - Internal Src1
- f2 - Internal Src2 through combiner in Normal
- LO - Connect external source through the rear-panel ( J7 for Port 3; J3 for Port 4 ).
  
  - Connect the DUT LO to PNA Port 3 or Port 4.
  - For Port 3, NO switching is required.
  - For Port 4, switch Port 4 Bypass Switch to Rear Panel.

3. Internal LO through Port 3 or Port 4; External f2 through rear-panel  (4-port PNA only)
This configuration is commonly used for FCA measurements where the internal second source is used as the LO.

- f1 - Internal Src1
- f2 - External f2 through rear-panel J9. No switching required.
- LO - Internal Src2 through:
Create a Swept IMD Measurement

1. On the PNA front panel, press Meas then [Measurement Class]

2. Select Swept IMD, then either:
   - OK delete the existing measurement, or
   - New Channel to create the measurement in a new channel.

3. A Swept IMD measurement is displayed. To select additional parameters to display, click Response, then Measure, then select a parameter from the list.

How to start the Swept IMD Setup dialog
To provide quicker access, use the Setup softkey. Learn how.

Using front-panel HARDKEY [softkey] buttons

1. Press FREQ
2. then [Swept IMDx Setup]

Using a mouse with PNA Menus

1. Click Stimulus
2. then Frequency
3. then Swept IMDx Setup

Valid Mixer Configuration / Sweep Type Combinations

Configuring the IMDx Setup dialog can be challenging at first. RED messages like this one appear at the bottom of the IMDx Setup dialog to notify you of an invalid setup.

The following information can provide you with a 'mental model' of the VALID mixer configuration / sweep type combinations which can make it easier to setup your measurement.

The following are the Valid Mixer Configurations in table format:
<table>
<thead>
<tr>
<th>Sweep Type</th>
<th>Input</th>
<th>LO</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swept Input (Fixed Tones)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sweep fc</td>
<td>Swept</td>
<td>Swept</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Swept</td>
<td></td>
<td>Swept</td>
</tr>
<tr>
<td>Fixed Input</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CW</td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
<tr>
<td>Sweep Delta F</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td>(Tone) Power Sweep</td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
<tr>
<td>LO Power Sweep</td>
<td>Fixed</td>
<td>Swept</td>
<td>Swept</td>
</tr>
</tbody>
</table>

The following is an explanation of the table:

- **SWEPT Input** (There is ONLY one: **Sweep fc**) - Sweeps the center frequency of the tones, but the tone spacing remains fixed.
  
  - Either the LO or Output MUST also be swept.
  
  - On the Mixer Frequency tab, select Start/Stop, or Center/Span for each range to be swept.

- **FIXED Input** - The center frequency of the tones is fixed. On the Mixer Frequency tab - Input range, select Fixed for the following sweep types:
  
  - **Sweep Delta F** Tone separation changes. The LO and Output frequencies are always fixed.
  
  - **Power Sweep** - Tone power changes. The LO and Output frequencies are always fixed.
  
  - **LO Power Sweep** - All frequencies are fixed as the LO power is swept.
  
  - **CW** - Tones do NOT change. The LO or Output frequencies CAN be swept.
Tips

Although you will soon become comfortable navigating these tabs, at first it may be best to complete the dialog in the following order:

1. On the Tone Frequency tab, set the Sweep Type and Tone Frequencies.

2. On the Mixer Frequency tab, set a valid mixer configuration.

   - Input center / fixed frequency CAN be set on both the Tone Frequency tab and the Mixer Frequency tab. When you set one, the other is updated automatically.

Setup Dialog

The following tabs are shared with the Swept IMD Setup dialog:

- Tone Frequency tab
- Tone Power tab

The following tabs are shared with all Mixer / Converter Applications

- Mixer Frequency tab
- Mixer (LO) Power tab
- Mixer Setup tab (NOT shared)

Tone Frequency tab - IMDx Setup dialog box help

Learn about this dialog.
### Tone Power tab - IMDx Setup - dialog box help

Learn about this dialog.

### Mixer Frequency tab - IMDx Setup - dialog box help

Learn about this dialog.
To accommodate single-source PNA models, an external source can be used for the RF2 tone.
Learn how to configure an external source and combiner to make Swept IMD and IMDx measurements.

Converter Stages  Choose from 1 or 2 stage DUT (# of LOs).

Converter Model

f1  Always uses PNA internal source 1.

(DUT Input) Port N  Available for selection on 4-port PNA models. Select the PNA port (1 or 3) to connect to the DUT Input.

f2  Select a source to be used for the f2 tone. This selection is available when an external source is configured and the Active box is checked on the External Source Configuration dialog.

LO1 (and LO2 for 2-stage DUTs)  Select the source to use for the specified LO. Available for selection when an external source is configured and Active.

(DUT Output) Port N  Available for selection on 4-port PNA models. Select the PNA port to connect to the DUT Output.

Hardware Configuration

Add Source  Click to configure an external source using the External Source Configuration dialog.

Path Configuration  Click to launch the Path Configuration dialog (PNA-X models only).
Mixer (LO) Power tab - IMD Setup - dialog box help

Learn about this dialog

How to add IMDx Parameters

Using front-panel HARDKEY [softkey] buttons

1. Press TRACES
2. then [New Trace]
3. then select a parameter

Using a mouse with PNA Menus

1. Click Trace/Chan
2. then New Trace
3. then select a parameter
This dialog is shared with Swept IMD for Amplifiers.
Learn about this dialog.

**IMDx Calibration**

Calibration for IMDx is exactly the same as [calibration for IMD](#) with the following exception:

- You can choose to perform a source power cal of the LO source. If the LO is a fixed frequency, this step is performed very fast.

The results of an IMDx calibration are very similar to the results that are achieved from an [SMC calibration](#).

---

**Last Modified:**

- 27-Apr-2011   Removed Copy Channels limitation
- 27-Sep-2010   Merged with Swept IMD and Mixer tabs
- 12-Apr-2010   Removed some fixturing limitations
- 11-Aug-2009   Added limited port mapping
- 13-Jul-2009   Added 2nd order notes
- 2-Feb-2009    New topic
IMx Spectrum for Converters

IMx Spectrum measurements are a distinct measurement class and therefore always made in a separate channel. This topic discusses all aspects of an IMx Spectrum measurement.

- Requirements and Limitations
- How an IMx Spectrum Channel Works
- Create an IMx Spectrum Channel
- Select IMx Spectrum Parameters
- Start the IMx Spectrum Setup dialog
  - Frequency tab
  - Tone Power tab
  - Mixer Frequency tab (separate topic)
  - Mixer Power tab (separate topic)
  - Mixer Setup tab
- Calibration

See Also

- Swept IMD Measurements
- Programming commands

Requirements and Limitations

IMx Spectrum requires IMD (Opt 087) and FCA (either Opt 082 or 083). The following PNA features are NOT available in an IMx Spectrum channel:

- Unratioed receiver measurements (A, B, R)
- ECal User Characterization
- FOM or FCA
- Time Domain
- Balanced measurements
How an IMx Spectrum for Converters Channel Works

Before reading this topic, you should become familiar with IMD Concepts.

The IMx Spectrum channel provides a traditional Spectrum Analyzer view of the intermodulation distortion behavior of a converter output. Unlike the Swept IMDx channel, the main tones (F1 and F2) are fixed while the receiver is swept over a frequency range of interest in order to generate a display as shown below.

![IMx Spectrum trace (bottom) with Swept IMDx traces (top)](image)

A typical Spectrum Analyzer does NOT have a signal source. This one does. The signal source, or stimulus, settings for the F1 and F2 main tones can be set in either the IMx Spectrum channel or the Swept IMDx channel. These settings include the frequencies and power levels of the main tones.
Although the stimulus settings can be set in either the IMx Spectrum channel or the Swept IMDx channel, the receiver settings are set ONLY in the Frequency tab of the IMx Spectrum dialog. These settings include how many tone products to view, which determines the center and frequency span and the resolution bandwidth. You can also set a Linear sweep type, then enter an arbitrary receiver frequency range.

You can choose from several resolution bandwidths which also determines the number of data points used in the channel. The higher the Res BW, the fewer the number of data points.

The IMx Spectrum channel performs multiple measurements for each data point in order to reject unwanted images which are generated by the PNA internally. This provides a high degree of confidence that signals captured in a trace are real and are not spurious responses generated in the measurement process.

**Differences from IM Spectrum for Amplifiers**

An IMx Spectrum for Converters channel has the following important differences from an IM Spectrum channel for Amplifiers.

- In an IMx Spectrum channel, **all converter frequencies are fixed**. (Input, Output, LO, Tone Spacing, and Tone Power).
- The IMx Spectrum channel **CANNOT track** with the IMDx channel, as it can in IMD for Amplifiers.

**Create an IMx Spectrum Channel**

An IMx Spectrum channel can be created independently, or from an existing Swept IMDx channel.

**To create an independent IMx Spectrum channel**

1. On the PNA front panel, press **Meas** then **[Measurement Class]**
2. Select **IMx Spectrum**, then either:
   - **OK** delete the existing measurement, or
   - **New Channel** to create the measurement in a new channel.

**To create an IMx Spectrum channel from an existing Swept IMD channel**

This creates or configures an existing IMx Spectrum channel based upon the configuration of the Swept IMD channel at the Marker frequency.

1. With a Swept IMD channel active, press the **MARKER** key on the front panel.
2. Move the marker to the data point of interest. It can be moved again later.
3. Press **[Marker Functions]** then **[Marker -> IMx Spectrum]**

**Select IMx Spectrum Parameters**
How to add IMx Spectrum traces
With the IMx Spectrum channel active and a Tr1 Output trace displayed:

Using front-panel HARDKEY [softkey] buttons
1. Press TRACES
2. then [New Trace]
3. then select a new parameter

Using a mouse with PNA Menus
1. Click Trace/Chan
2. then New Trace
3. then select a new parameter

New Trace - IM Spectrum - dialog box help
Output Port 2, Incident
View signals OUT of the DUT and into PNA port 2 (B receiver).
Note: Only viewing the OUTPUT tones is supported.

IMx Spectrum Setup Dialog

How to start the IMx Spectrum Setup dialog
To provide quicker access, use the Setup softkey. Learn how.
With an IMx Spectrum measurement active:

Using front-panel HARDKEY [softkey] buttons
1. Press SWEEP
2. then [IMx Spectrum Setup]

Using a mouse with PNA Menus
1. Click Stimulus
2. then Frequency
3. then IMx Spectrum Setup
Learn about this dialog and about *.mxr files.
Learn about this dialog and about *.mxr files.

Learn more about this dialog and about *.mxr files.
To accommodate single-source PNA models, an external source can be used for the RF2 tone.
Learn how to configure an external source and combiner to make Swept IMD and IMDx measurements.

**Converter Stages** Choose from 1 or 2 stage DUT (# of LOs).

**Converter Model**

- **f1** Always uses PNA internal source 1.

- **(DUT Input) Port N** Available for selection on 4-port PNA models. Select the PNA port (1 or 3) to connect to the DUT Input.

- **f2** Available for selection when an external source is configured and Active.

- **LO1 (and LO2 for 2-stage DUTs)** Select the source to use for the specified LO. Available for selection when an external source is configured and Active.

- **(DUT Output) Port N** Available for selection on 4-port PNA models. Select the PNA port to connect to the DUT Output.

**Hardware Configuration**

- **Add Source** Click to configure an external source using the External Source Configuration dialog.

- **Path Configuration** Click to launch the Path Configuration dialog (PNA-X models only).

**Calibration**

A calibration of the IMx Spectrum channel is NOT performed using a calibration wizard.

An IMx Spectrum channel is calibrated from a Cal Set that is used on a Swept IMDx channel. The Cal Set can be applied to the IMx Spectrum channel using the Manage Cal Sets dialog (Learn how) or from the Marker =>IMx Spectrum softkey (Learn how).

However, a Swept IMDx channel with **Sweep Type = Power Sweep** can NOT be applied to a IMx Spectrum channel. This is because a Cal Set for power sweep contains only a CW frequency and the IMx Spectrum channel requires a swept frequency range. Zero Span is not supported in an IMx Spectrum channel.
A calibrated IMx Spectrum trace corrects the source and receiver power level accuracy of the displayed Tones.

See Swept IMD Calibration
Learn how to apply a Cal Set to the IMx Spectrum channel

Last Modified:

9-May-2011 Edited Mixer Setup
27-Apr-2011 Removed Copy Channels limitation
16-Aug-2010 Diverted to common Mixer topic
16-Sep-2009 Fixed Calibration
11-Aug-2009 Added limited port mapping
2-Feb-2009 New topic
Time Domain

Time Domain allows you to view a device response as a function of time. The following are discussed in this topic:

- Overview
- How the PNA Measures in the Time Domain
- Calibration for Time Domain
- Transmission Measurements
- Measurement Response Resolution
- Measurement Range and Alias Responses
- How to make Time Domain Settings
- Gating
- Window Settings

**Note:** Time Domain measurements are only available on PNAs with Option 010. See [PNA Options](#). See the updated [App Note: Time Domain Analysis Using a Network Analyzer](#).

**Overview**

In normal operation, the PNA measures the characteristics of a test device as a function of frequency. With Time Domain (opt 010), the frequency information is used to calculate the inverse Fourier transform and display measurements with time as the horizontal display axis. The response values appear separated in time, allowing a different perspective of the test device's performance and limitations.

The graphic below compares the same cable reflection measurement data in both the frequency and time domain. The cable has two bends. Each bend creates a mismatch or change in the line impedance.

- The frequency domain S11 measurement shows reflections caused by mismatches in the cable. It is impossible to determine where the mismatches physically occur in the cable.

- The time domain response shows both the location and the magnitude of each mismatch. The responses indicate that the second cable bend is the location of a significant mismatch. This mismatch can be [gated out](#), allowing you to view the frequency domain response as if the mismatch were not present. Distance Markers can be used to pinpoint the distance of the mismatch from the reference plane.
How the PNA Measures in the Time Domain

Time domain transform mode simulates traditional Time-Domain Reflectometry (TDR), which launches an impulse or step signal into the test device and displays the reflected energy on the TDR screen. By analyzing the magnitude, duration, and shape of the reflected waveform, you can determine the nature of the impedance variation in the test device.

The PNA does not launch an actual incident impulse or step. Instead, a Fourier Transform algorithm is used to calculate time information from the frequency measurements. The following shows how this occurs.

A single frequency in the time domain appears as a sine wave. In the following graphic, as we add the fundamental frequency (F₀), the first harmonic (2F₀), and then the second harmonic (3F₀), we can see a pulse taking shape in the Sum waveform. If we were to add more frequency components, the pulse would become sharper and narrower. When the PNA sends discrete frequencies to the test device, it is in effect, sending individual spectral pieces of a pulse separately to stimulate the test device.

During an S11 reflection measurement, these incident signals reflect from the test device and are measured at the receiver. This is when the time domain transform calculations are used to add the separate spectral pieces together.

For example, consider a short length of cable terminated with an open. All of the power in the incident signal is reflected, and the reflections are 'in-phase' with the incident signal. Each frequency component is added together, and we see the same pattern as the simulated incident would have looked (above). The magnitude of the reflection is related to the impedance mismatch and the delay is proportional to the distance to the mismatch. The x-axis (time) scale is changed from the above graphic to better show the delay.

Alternately, the same cable terminated with a short also reflects all of the incident power, but with a phase shift of 180 degrees. As the frequency components from the reflection are added together, the sum appears as a negative impulse delayed in time.
**Calibration for Time Domain**

For simplicity, we have discussed incident signals reflecting off discontinuities in the test device. By far the most common network analyzer measurement to transform to time domain is a ratioed S11 measurement. An S11 reflection measurement does not simply display the reflections measured at the A receiver - it displays the ratio (or difference) of the A receiver to the Reference receiver. In addition, the S11 measurement can also be calibrated to remove systematic errors from the ratioed measurement. This is critical in the time domain as the measurement plane, the point of calibration, becomes zero on the X-axis time scale. All time and distance data is presented in reference to this point. As a result, both magnitude and time data are calibrated and very accurate.

The following shows where the time domain transform occurs in the PNA data flow: (see Data Access Map)

1. Acquire raw receiver (A and R1) data
2. Perform ratio (A/R1)
3. Apply calibration
4. Transform data to time domain
5. Display results

Therefore, although a time domain trace may be displayed, a calibration is always performed and applied to the frequency domain measurement which is not displayed.

**Transmission Measurements**

The most common type of measurement to transform is an S11 reflection measurement. However, useful information can be gained about a test device from a transformed S21 transmission measurement. The frequency components pass through the test device and are measured at the B receiver. If there is more than one path through the device, they would appear as various pulses separated in time.

For example, the following transmission measurement shows multiple paths of travel within a Surface Acoustic Wave (SAW) filter. The largest pulse (close to zero time) represents the propagation time of the shortest path through the device. It may not be the largest pulse or represent the desired path. Each subsequent pulse represents another possible path from input to output.
Triple travel is a term used to describe the reflected signal off the output, reflected again off the input, then finally reappearing at the output. This is best seen in a time domain S21 measurement.

**Measurement Response Resolution**

In the previous paragraphs, we have seen that using more frequency components causes the assembled waveform to show more detail. This is known as measurement response resolution, which is defined as the ability to distinguish between two closely spaced responses.

Note: Adjusting the transform time settings improves display resolution, but not measurement resolution.

The following graphic shows the effect of both a narrow and wide frequency span on the response resolution. The wider frequency span enables the analyzer to resolve the two connectors into separate, distinct responses.

**Resolution Formula**

For responses of equal amplitude, the response resolution is equal to the 50% (-6 dB) points of the impulse width, or the step rise time which is defined as the 10 to 90% points as shown in the following image.
The following table shows the **approximated** relationship between the frequency span and the window selection on response resolution for responses of equal amplitude.

<table>
<thead>
<tr>
<th>Window</th>
<th>Low-pass step (10% to 90%)</th>
<th>Low-pass impulse (50%)</th>
<th>Bandpass impulse</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum</td>
<td>0.45 / f span</td>
<td>0.60 / f span</td>
<td>1.20 / f span</td>
</tr>
<tr>
<td>Normal</td>
<td>0.99 / f span</td>
<td>0.98 / f span</td>
<td>1.95 / f span</td>
</tr>
<tr>
<td>Maximum</td>
<td>1.48 / f span</td>
<td>1.39 / f span</td>
<td>2.77 / f span</td>
</tr>
</tbody>
</table>

For example, using a 10 GHz wide frequency span and a normal window in Bandpass impulse mode, response resolution (in time) equals:

- Time Res = \( \frac{1.95}{\text{frequency span}} \)
- Time Res = \( \frac{1.95}{10 \text{ GHz}} \)
- Time Res = 195 ps

To calculate the physical separation (in distance) of the responses which can be resolved, multiply this value times the speed of light (c) and the relative velocity (Vf) of propagation in the actual transmission medium. In this case, Vf = 0.66 for polyethylene dielectric.

- Distance Res = \( 195 \text{ ps} \times c \times Vf \)
- Distance Res = \( 195 \text{ ps} \times (2.997925 \text{ E8 m/s}) \times 0.66 \)
- Distance Res = 38 mm

For reflection measurements, because of the 2-way travel time involved, this means that the minimum resolvable separation between discontinuities is half of this value or 19 mm.

Although a wider frequency span causes better measurement resolution, the **measurement range** becomes limited. Also, increasing the frequency range can cause a measurement calibration to become invalid. Be sure to adjust the frequency span BEFORE performing a calibration.
Measurement Range and Alias Responses

Measurement range is the length in time in which true time domain responses can be seen. The measurement range should be large enough to see the entire test device response without encountering a repetition (alias) of the response. An alias response can hide a true time domain response.

To increase measurement range in both modes, change either of these settings:

- Increase the number of points
- Decrease the frequency span

Notes:

- After making these settings, you may need to adjust the transform time settings to see the new measurement range.
- Decreasing the frequency span degrades measurement resolution.
- Make frequency span and number of points settings BEFORE calibrating.
- Maximum range also depends on loss through the test device. If the returning signal is too small to measure, the range is limited regardless of the frequency span.

Alias Responses

An alias response is not a true device response. An alias response repeats because each time domain waveform has many periods and repeats with time (see How the PNA Measures in the Time Domain). Alias responses occur at time intervals that are equal to 1/ frequency step size.

The PNA adjusts the transform time settings so that you should only see one alias free range on either side (positive and negative) of zero time. However, these settings are updated only when one of the toolbar settings are changed.

To determine if a response is true, put a marker on the response and change the frequency span. A true device response will not move in time. An alias response will move.

For example, in the above graphic, the marker 1 response occurs at 14.07 inches. When the frequency span is changed, this response remains at 14.07 inches. The marker 2 response moves.

Range Formula
You can calculate the alias-free measurement range (in meters) of the PNA using the following formula for TDR (reflection) measurements:

Range (meters) = \((1/\Delta f) \times V_f \times c\)

Where:

- \(\Delta f\) = frequency step size (frequency span/number of points - 1)
- \(V_f\) = the velocity factor in the transmission line
- \(c\) = speed of light = 2.997925 E8 m/s

For example: For a measurement with 401 points and a span of 2.5 GHz, using a polyethylene cable (\(V_f = 0.66\))

- \(\text{Range} = (1 / (2.5E9 / 400)) \times 2.997925 \times 0.66\)
- \(\text{Range} = 6.25E6 \times 2.997925 \times 0.66\)
- \(\text{Range} = 32\) meters

In this example, the range is 32 meters in physical length. To prevent the time domain responses from overlapping or aliasing, the test device must be 32 meters or less in physical length for a transmission measurement.

To calculate the one-way distance for a reflection measurement rather than round-trip distance, simply divide the length by 2. In this case, the alias-free range would be 16 meters.

**How to make Time Domain Settings**

The following launches the Time Domain toolbar

On the toolbar, click More... to launch the Time Domain dialog box

Using front-panel HARDKEY [softkey] buttons

1. Press ANALYSIS
2. then [Transform]
3. then [More]
4. then [Transform Tool]

Using a mouse with PNA Menus

1. Click Marker/Analysis
2. then Transform
**Transform** dialog box help

**Category**  Select Transform, Window, or Gating

**Transform**  Turns time domain transform ON and OFF.

**Coupling Settings**  Launches the [Trace Coupling Settings](#) dialog box.

**Time Settings**
The following settings adjust the **display resolution**, allowing you to zoom IN or OUT on a response. They do NOT adjust **measurement range** or **measurement resolution**.

These settings automatically update (when one of these values are updated) to limit the display to one **alias-free response** on either side of zero time.

**Start**  Sets the transform start time that is displayed on the PNA screen.

**Note:** Zero (0) seconds is always the **measurement reference plane**. Negative values are useful if moving the reference plane.

**Stop**  Sets the transform stop time that is displayed on the PNA screen.

**Center**  Sets the transform center time that is displayed in the center of the PNA screen.

**Span**  Sets the transform span time that is split on either side of the Center value.

**Transform Mode**
Transform modes are three variations on how the time domain transform algorithm is applied to the frequency domain measurement. Each method has a unique application.
<table>
<thead>
<tr>
<th>Mode</th>
<th>Benefit - application</th>
<th>Limitation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Low pass Impulse</strong></td>
<td>Highest resolution. Most useful for seeing small responses in devices that pass low frequencies, such as cables.</td>
<td>In both Low pass modes, frequencies down to DC and negative frequencies are extrapolated. Therefore, the Start frequency is adjusted when you click <strong>Set Freq. Low Pass</strong>. Because this will affect calibration accuracy, be sure to calibrate AFTER completely setting up your time domain measurement.</td>
</tr>
<tr>
<td><strong>Low pass Step</strong></td>
<td>Easiest to identify inductive and capacitive discontinuities in devices that pass low frequencies, such as cables.</td>
<td>Does NOT show capacitive and inductive reactance. For the same frequency span and number of points, band pass mode has twice the impulse width, which hides closely spaced responses degrading the response resolution.</td>
</tr>
<tr>
<td><strong>Band pass Impulse</strong></td>
<td>Easiest method - can be used with any frequency sweep. Most useful for measuring band limited devices such as filters and DC blocked cables.</td>
<td>Does NOT show capacitive and inductive reactance. For the same frequency span and number of points, band pass mode has twice the impulse width, which hides closely spaced responses degrading the response resolution.</td>
</tr>
</tbody>
</table>

The following chart shows how to interpret results from various discontinuity impedances using Low pass Step and either Low pass or Band pass Impulse modes.

<table>
<thead>
<tr>
<th>IMPEDANCE</th>
<th>STEP RESPONSE</th>
<th>IMPULSE RESPONSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>OPEN</td>
<td>Unity Reflection</td>
<td>Unity Reflection</td>
</tr>
<tr>
<td>SHORT</td>
<td>Unity Reflection = 180</td>
<td>Unity Reflection = 180</td>
</tr>
<tr>
<td>RES/STOR R &gt; Z_0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RES/STOR R &lt; Z_0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>INDUCTOR</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CAPACITOR</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Effect on Measurement Range**

**Band pass mode** - measurement range is inversely proportional to frequency step size.

**Low pass mode** - measurement range is inversely proportional to the fundamental (start) frequency AFTER clicking Set Freq. Low Pass.

**Set Freq. Low Pass** USE ONLY IN LOW PASS MODES

Recomputes the start frequency and step frequencies to be harmonics of the start frequency. Start frequency is computed by the following formula: **Low Pass Start Frequency = Stop Frequency / Number of points.**
The computed value must always be greater than or equal to the analyzer's minimum frequency.

**Note:** The number of points or stop frequency may be changed in order to compute this value.

**Distance Marker Settings** Launches the Distance Marker Settings dialog box.

---

**Gating**

Perhaps the most beneficial feature of time domain transform is the Gating function. When viewing the time domain response of a device, the gating function can be used to "virtually" remove undesired responses. You can then simultaneously view a frequency domain trace as if the undesired response did not exist. This allows you to characterize devices without the effects of external devices such as connectors or adapters.

**Note:** When a discontinuity in a test device reflects energy, that energy will not reach subsequent discontinuities. This can "MASK", or hide, the true response which would have occurred if the previous discontinuity were not present. The PNA Gating feature does NOT compensate for this.

The following measurements images show a practical example how to use and perform gating. The test device is a 10inch cable, then a 6 dB attenuator, terminated with a short. The following four discontinuities are evident in window 2, from left to right:

1. A discontinuity in the test system cable which appeared after calibration. It is identified by marker 2 at -10.74 inches (behind the reference plane).
2. A discontinuity in the 10 inch device cable shortly after the reference plane.
3. The largest discontinuity is the attenuator and short shown by marker 1 at -12.67 dB (6 dB loss in both forward and reverse direction).
4. The last discontinuity is a re-reflection from the device cable.

We will gate IN the attenuator response. All other responses will be gated OUT.

**Window 1.** Create original S11 frequency domain trace. Shows ripple from all of the reflections.

**Window 2.** Create a new S11 trace - same channel; new window. Turn Transform ON.

**Window 3.** On the transformed trace, turn gating ON. Center the gate on the large discontinuity (2.500ns). Adjust gate span to completely cover the discontinuity. Select Bandpass gating type.

**Window 4.** On the original frequency measurement, turn Gating ON (Transform remains OFF). View the measurement without the effects of the two unwanted discontinuities. The blue trace is a measurement of the 6 dB attenuator with the unwanted discontinuities PHYSICALLY removed. The difference between the two traces in window 4 is the effect of "masking".
Learn how to launch the Transform dialog box

Programming Commands

Transform Gating dialog box help

Gating  Turns Gating ON and OFF.
Coupling Settings  Launches the Setup Trace Coupling dialog box.
Start  Specifies the start time for the gate.
Stop  Specifies the stop time for the gate.
Center  Specifies the value at the center of the area that is affected by the gating function. This value can be anywhere in the analyzer range.
Span  Specifies the range to either side of the center value of area that is affected by the gating function.
Gate Type  Defines the type of filtering that will be performed for the gating function. The gate start and stop
flags on the display point toward the part of the trace you want to keep.

- **Bandpass** - KEEPS the responses within the gate span.
- **Notch** - REMOVES the responses with the gate span.

**Gate Shape** Defines the filter characteristics of the gate function. Choose from Minimum, Normal, Wide, Maximum

<table>
<thead>
<tr>
<th>Gate Shape</th>
<th>Passband Ripple</th>
<th>Sidelobe Levels</th>
<th>Cutoff Time</th>
<th>Minimum Gate Span</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum</td>
<td>±0.1 dB</td>
<td>-48 dB</td>
<td>1.4/Freq Span</td>
<td>2.8/Freq Span</td>
</tr>
<tr>
<td>Normal</td>
<td>±0.1 dB</td>
<td>-68 dB</td>
<td>2.8/Freq Span</td>
<td>5.6/Freq Span</td>
</tr>
<tr>
<td>Wide</td>
<td>±0.1 dB</td>
<td>-57 dB</td>
<td>4.4/Freq Span</td>
<td>8.8/Freq Span</td>
</tr>
<tr>
<td>Maximum</td>
<td>±0.01 dB</td>
<td>-70 dB</td>
<td>12.7/Freq Span</td>
<td>25.4/Freq Span</td>
</tr>
</tbody>
</table>

**Cutoff time** -- is the time between the stop time (-6 dB on the filter skirt) and the peak of the first sidelobe. The diagram below shows the overall gate shape and lists the characteristics for each gate shape.

- T₁ is the gate span, which is equal to the stop time minus the start time.
- T₂ is the time between the edge of the passband and the 6 dB point, representing the cutoff rate of the filter.
- T₃ is the time between the 6 dB point and the edge of the gate stopband.
- For all filter shapes T₂ is equal to T₃, and the filter is the same on both sides of the center time.

**Minimum gate span** -- is twice the cutoff time. Each gate shape has a minimum recommended gate span for proper operation. This is a consequence of the finite cutoff rate of the gate. If you specify a gate span that is smaller that the minimum span, the response will show the following effects:

- distorted gate shape that has no passband
- distorted shape
- incorrect indications of start and stop times
may have increased sidelobe levels

Window Settings

There are abrupt transitions in a frequency domain measurement at the start and stop frequencies, causing overshoot and ringing in a time domain response. The window feature is helpful in lessening the abruptness of the frequency domain transitions. This causes you to make a tradeoff in the time domain response. Choose between the following:

- **Minimum Window = Better Response Resolution** - the ability to resolve between two closely spaced responses.
- **Maximum Window = Dynamic Range** - the ability to measure low-level responses.

![Minimum Window and Maximum Window](image)

Learn how to launch the Transform dialog box

**Programming Commands**

![Transform Dialog Box](image)
**Transform - Window** dialog box help

**Coupling Settings** Launches the [Setup Trace Coupling](#) dialog box. The window settings balance response resolution versus dynamic range.

- Minimum Window = Best Response Resolution
- Maximum Window = Best Dynamic Range

The following three methods all set window size. For best results, view the time domain response while making these settings.

- **Minimum - Maximum** Move the slider with a mouse to change the window size
- **Kaiser Beta** Changes window size using a Kaiser Beta value
- **Impulse Width** Changes window size using an Impulse Width value

Learn more about [Windowing](#) (top)

---

**How to make Trace Coupling Settings**

You can launch the **Trace Coupling Settings** dialog box from any of the following dialog boxes:

- **Transform**
- **Gating**
- **Window**

---

[Programming Commands](#)
Trace coupling allows you to change time domain parameters on a measurement, and have the same changes occur for all other measurements in the channel.

For example:

If you are simultaneously viewing a frequency domain measurement and time domain measurement, and **Coupling** is enabled in this dialog box, and ALL **Gating Parameters** are checked in this dialog box, and on the time domain measurement you change the **Gate Span** parameter,

Then the frequency domain measurement will automatically change to reflect the time domain gated span.

**Coupling ON/OFF** Check to enable coupling. All of the measurements in the active channel are coupled.

The following parameters are available for coupling:

**Transform Parameters**
- **Stimulus** Start, Stop, Center, and Span TIME settings.
- **State** (On/Off) Transform ON and OFF
- **Window** Kaiser Beta / Impulse Width
- **Mode** Low Pass Impulse, Low Pass Step, Band Pass

**Gating Parameters**
- **Stimulus** Start, Stop, Center, and Span TIME settings.
- **State** (On/Off) Gating ON and OFF
- **Shape** Minimum, Normal, Wide, and Maximum
- **Type** Bandpass and Notch

To launch the Distance marker dialog box, click **Dist. Marker Settings** on the **Transform** dialog box.
When markers are present on a time domain measurement, distance is automatically displayed on the marker readout, marker table, and print copy. To learn how to create markers on your measurement see marker settings.

This dialog box allows you to customize the time domain distance marker readings. These settings affect the display of ALL markers for only the ACTIVE measurement (unless Distance Marker Unit is coupled on the Trace Coupling dialog box).

**Marker Mode** Specifies the measurement type in order to determine the correct marker distance.

- Select **Auto** for S-Parameter measurements.
- Select **Reflection** or **Transmission** for arbitrary ratio or unratioed measurements.

**Auto** If the active measurement is an S-Parameter, automatically chooses reflection or transmission. If the active measurement is a non S-Parameter, reflection is chosen.

**Reflection** Displays the distance from the source to the receiver and back divided by two (to compensate for the return trip.)

**Transmission** Displays the distance from the source to the receiver.

**Units** Specifies the unit of measure for the display of marker distance values.

**Velocity Factor** Specifies the velocity factor that applies to the medium of the device that was inserted after the measurement calibration. The value for a polyethylene dielectric cable is 0.66 and 0.7 for PTFE dielectric. 1.0 corresponds to the speed of light in a vacuum. This is useful in Time Domain for accurate display of time and distance markers.

This setting can also be made from the Electrical Delay and Port Extensions dialog boxes.
Integrated Pulse Measurements

Beginning with PNA Rev. A.09.20, the Pulse Setup dialogs shown in this topic are integrated in the PNA firmware and are available with Opt 008 or Opt H08.

Previously, setup was performed with the Narrowband or Wideband pulse programs (E836x models can continue to use these programs) and with SCPI and COM commands. With the appropriate hardware options (Opt 021, 022, 025) these commands are still available without Opt 008 or H08.

In this topic

- Pulse Setup
- Pulse Generator Setup
- Pulse Trigger Tab
- Pulse Gens and IF Block Diagram
- Calibration in Pulse

See Also (separate topics)

- IF Path Configuration
- Programming commands
- Narrowband Pulsed Application (Opt H08)
- WB Pulsed App
- IF Access User Interface Settings (PNA "C" models)

How to start the Pulse Setup dialog

To provide quicker access, use the Setup softkey. Learn how.

Using front-panel HARDKEY [softkey] buttons

1. Press Sweep
2. then [Pulse Setup]

Using a mouse with PNA Menus

1. Click Stimulus
2. then Sweep
3. then Pulse Setup

Programming Commands
The Basic (upper box) controls allow simple pulse measurements using the default (Autoselect) settings in the Advanced section (lower box) of the dialog.

Pulsed measurements are performed in a Standard channel. See Measurement Class. However, several PNA measurement settings are controlled by the Pulse setup, such as sweep type, number of points, and so forth.

**Pulse Measurement**

- **Off** - Source and Receivers are NOT pulsed

- **Standard Pulse** - With pulsed RF, the PNA can be configured to sweep in frequency, power sweep, and CW time. Narrow the receiver pulse width and enter delay to make 'point-in-pulse' measurements.

- **Pulse Profile** - Pulse profile measurements provides a time domain (CW frequency) view of the pulse envelope. Profiling is performed using a measurement technique that "walks" a narrow receiver "snapshot" across the width of the pulse. This is analogous to using a camera to take many small snapshots of a wide image, then piecing them together to form a single, panoramic view.
Pulse Profiling can be performed using ratioed or unratioed measurements. You can preview the pulse on port 1 by using an R1 receiver measurement.

- Pulse Profiling is performed at a single CW frequency in either Narrowband or Wideband mode.
- To select the CW Frequency, click **Stimulus**, then **Sweep Type**.
- In Narrowband mode, the delay increment value, which is responsible for "walking" the receiver snapshot across the pulse, is selected by the PNA and is accessible only with a programming command.
- In Wideband mode, the receiver is walked across the pulse by making a sequence of closely-spaced measurements in real-time.

**Pulse Timing**

- **Pulse Width** - Sets the width of the source pulse. See [measurement timing](#) to learn how to control the receiver width and delay.
- **Pulse Period** - The time to make one complete pulse.
- **Pulse Frequency (PRF)** - The reciprocal of Period (1/ Period). See [Internal Pulse Generators](#) to learn more.

By default, these settings configure Pulse Gen 1 to drive Source Modulators 1 and 2. This can be changed from the Advanced Settings Pulse Generator Setup dialog.

**-------- Advanced Settings --------**

The following settings allow maximum control of a Pulse measurement.

**Note:** When the "Auto" check boxes are cleared, it is possible to configure settings to make an invalid measurement.

**Properties**

- **Autoselect pulse detection method** - check to automatically switch between Narrowband and Wideband based on the Pulse Width.
  - In Standard Pulse:
    - **Wideband** - used when the (source) Pulse Width is WIDER than the fastest receiver acquisition time. This allows the receiver to measure all pulse ON time - no pulse OFF time. The PNA will select Wideband whenever possible.
    - **Narrowband** - used when the (source) Pulse Width is NARROWER than the fastest receiver acquisition time (267 ns). This measurement requires a spectral nulling technique to measure the pulse response through the DUT.
  - In Pulse Profile:
    - **Wideband** - used when the (source) Pulse Width is greater than 1.600 us. This allows the receiver make several sequential measurements to measure the entire pulse.
    - **Narrowband** - used when the (source) Pulse Width is less than or equal to 1.600 us.
**Autoselect IF Path Gain and Loss** - For future use.

**IF Path** - Click to launch the IF Path dialog.

**Optimize Pulse Frequency** - Automatically selects the Pulse Frequency and Pulse Period.

- In Narrowband, the pulse frequency is adjusted slightly to get the best spectral-nulling filtering possible.
- In Wideband, this checkbox is ignored.

**Autoselect Profile Sweep Time** - In Pulse Profile mode, adjusts the default X-axis start time to zero and the stop time double the Pulse Width. This allows you to see one complete pulse. If unchecked, the Sweep Time will not be changed.

To adjust the X-axis manually, click **OK** to close the dialog. Then click **Stimulus**, then **Sweep**, then **Sweep Time**, then change the **Start Time** and **Stop Time**.

**IFBW** - Select the IFBW for the measurement.

- In Narrowband, an IFBW as close as possible to the entered value will be used.
- In Wideband, this setting determines the receiver acquisition time - approximately 1/IFBW.

**Measurement Timing**

**Source1** and **Source2** - Used as RF Source Modulation Drive.

- **Width** - source pulse width.
- **Delay** - source pulse delay relative to the pulse generator clock.
- **Pulse Gen** - Pulse generator used to modulate the source. Select **CW** to have NO source modulation.

The receiver settings in this table change depending on whether the PNA is in Narrowband or Wideband mode.

- In Narrowband, for each IF receiver path, configure the Pulse Width, Delay, and Pulse Generator to be used to drive the receiver gate.
- In Wideband, all receiver paths are the same.

**Autoselect Width and Delay** - When checked, for Wideband mode and Pulse0, the receiver is adjusted to approximately 75% of the source pulse width, with 20% delay. This leaves approximately 5% of the source pulse ON after acquisition is complete.

When checked for Narrowband mode and Pulse Gen = CW, then the delay and width have no values. When a Pulse Gen is selected, then Delay and Width matches the RF Source.

**Autoselect Pulse Generators** - When checked:

- Pulse1 is selected for Modulator Drive.
- Pulse2 is CW (OFF).
- For Wideband, Pulse0 is selected to gate the ADC.
- For Narrowband, Pulse2 is selected.

**Pulse Generators** Click to launch the Pulse Generators Setup dialog.

---

**Pulse Generators Setup** dialog box help

This dialog is available with Option 025 (pulse generators).

To see this dialog, press Pulse Gen Setup on the Pulse Setup dialog.

**Internal Pulse Generators**

For each Pulse Generator configure the width, delay, and polarity.

Pulse0 triggers the PNA receivers when Enabled. The Width can NOT be configured. Delay defines the delay between the pulse clock and when the receiver begins to make a measurement. Learn more.

<table>
<thead>
<tr>
<th>Name</th>
<th>Width</th>
<th>Delay</th>
<th>Invert/Enable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pulse0</td>
<td>16.667 us</td>
<td>28.333 us</td>
<td></td>
</tr>
<tr>
<td>Pulse1</td>
<td>100.000 us</td>
<td>0.000 psec</td>
<td></td>
</tr>
<tr>
<td>Pulse2</td>
<td>100.000 us</td>
<td>0.000 psec</td>
<td></td>
</tr>
<tr>
<td>Pulse3</td>
<td>100.000 us</td>
<td>0.000 psec</td>
<td></td>
</tr>
<tr>
<td>Pulse4</td>
<td>100.000 us</td>
<td>0.000 psec</td>
<td></td>
</tr>
</tbody>
</table>

```
D = Delay; the time before each pulse begins
W = Width; the time the pulse is ON
Duty Cycle = W/P
```

**Internal Pulse Clock (P)**

- P = Period; one complete pulse cycle
- Pulse Frequency (PRF) = 1 / Period
Important: If $D + W$ is greater than $P$, then undefined PNA behavior results. There is NO error message or warning.

**Invert**  Check to cause the pulse ON time to be active low and OFF be active high.

Learn more about the Pulse Generators.

**Internal Pulse Modulators**

Check to enable one or both internal source modulators.

These are switches 8 and 9 in the [Block Diagram](#).

**Important:** When internally modulating the sources, **source leveling is automatically set to Open-loop**.

**Modulator Drive**  Choose the pulse generator to modulate the specified source. Choose from **CW** (NO pulse), Pulse 1, 2, 3, 4, External.  This is switch 7 [Block Diagram](#).

**Pulse Trigger**  click to start the Pulse Trigger dialog.

---

**Pulse Trigger Tab - Trigger dialog box help**

To see this dialog, press **Pulse Trigger** on the Pulse Generator Setup dialog or select **Stimulus**, then **Trigger** from the PNA Menu.

**Pulse Sync Trigger - Source**

Select **Internal** or **External** to provide sync capability for the internal pulse generators.

- **Internal** - The pulse generator is internally triggered and puts out a periodic pulse train with a period defined by the Pulse Generator Setup dialog.

- **External** - The period is ignored and the internal pulse generator puts out one set of pulses (P0-P4) per external trigger. All five pulse outputs have unique delay and pulse width settings. External trigger input is on the Pulse I/O connector pin 7 (PulseSyncIn). The PulseSyncIn line provides a configurable trigger signal into the Pulse Generators. If a level trigger is still valid when the first pulse set is finished, another set will be generated. Only one set of pulses is emitted when edge triggering is used. The length of time that it takes to emit one set of pulses is the end time of the last enabled pulse (largest of width + delay of
all the pulses P0-P4).

**Level/Edge**
Sets the edge or level of the trigger signal to which the internal pulse generators will respond when being externally triggered at the PulseSyncIn pin.
Positive = rising edge; Negative = falling edge.
These selections are available ONLY with DSP version: 4.0 FPGA: 34 or higher. Learn more. Otherwise, the pulse generators respond only to positive, level input trigger signals.

**Receiver**

**Synchronize receiver to pulse generator Pluse0** - Check to enable Pulse Gen 0 (P0) which is used to gate the ADC for wideband receiver measurements. Learn more about Pulse0.

**Receiver delay** - Set the amount of time to wait before triggering the ADC to begin acquisition.

---

**Pulse Trigger Block Diagram**

See complete description at IF Path Configuration

**Calibration in Pulse Mode**

To perform a calibration in pulse mode (option H08 / 008), first configure and apply the pulse parameters (PRF, Pulse Width, Delays, IF gating, and so forth) before calibrating the system. This will ensure the PNA is configured properly during the calibration and measurement.
Narrowband Pulsed Application

Note: The PNA-X with Option 025 has an integrated Pulse Measurement Setup (Opt 008) that makes this application no longer necessary on the PNA-X.

The Narrowband Pulsed Application is a Visual Basic program that provides a user interface for making pulsed measurements.

In this topic:

- Required Options
- Connecting External Pulse Generators
- Using the Narrowband Pulsed Application
- How to Configure Pulse Generators and Receivers
- Calibration in Pulse Mode
- Pulse Profiling
- Signal Reduction versus Gate Width
- Pulsed Frequency Converter Measurements
- Writing your own Narrowband Pulsed Application

The following enhancements were made in PNA Rev. 7.2:

- Enhanced Pulse Measurement Capabilities
- Support for Internal Pulse Generators / Modulators (PNA-X only)

See Also

- Learn about the Wideband Pulsed Application.
- For more conceptual information see our Pulsed Measurement App Notes.
- See PNA-X Block Diagram of IF Path / Pulse Generators / Source Modulation
- Programming commands
- Programming examples

Other IF Access Topics

Required Options and Equipment
The PNA H08 option provides the Narrowband Pulsed Application. The following options are also required. If your PNA does not have the required options, a message is displayed on the screen. For more information, see Pulsed-RF Measurements Configuration Guide.

**PNA-X Models**

- PNA-X models: None; however **Opts 021, 022, and 025** greatly enhance speed, performance, and convenience.
- PNA-X Opt 224 (2-port Dual Source) select the **2-port Dual Source** Configuration in the Path Configuration dialog to provide Pulse modulation on both port 1 and port 2.
- Use the PNA-X **Pulse I/O connector** to access the internal pulse generators
- See the PNA-X **IF Path Configuration block diagram** which includes the Pulse Modulators and Generators.

**Other PNA Models**

- E836x models: **Opt 014 (front panel access)** and **Opt 080 (frequency offset)**. To use the internal receiver gating feature of the Narrowband Pulsed Application, your PNA must have the **H11 hardware option**.
- PNA-L models: H08 NOT available
- Agilent 81104A or 81110A Pulse Generator with ONLY the 81105A or 81111A output modules. The 81112A module does NOT have selectable 50 ohm/1K ohm output impedance/load compensation to drive the 1K ohm PNA IF gates. For more information, see the 81100 Family of Pulse Pattern Generators Technical Specifications at: [http://cp.literature.agilent.com/litweb/pdf/5980-1215E.pdf](http://cp.literature.agilent.com/litweb/pdf/5980-1215E.pdf)

**Connecting External Pulse Generators**

Each 81110A Pulse Generator has two output modules. Each output can drive a PNA IF Receiver or Source Modulation (Z5623A H81). Connect the Pulse Generators as follows:

**81110A front panel connectors**

- Connect GPIB cables to the 81110A and PNA.
- Connect the PNA 10 MHz Ref Out to the 81110A 10 MHz IN.
- If using two 81110As for a total of 4 outputs, then connect the TRIGGER OUT of one to the EXT INPUT of the other 81110A.
- Connect the 81110A OUTPUTs to the PNA rear panel IF inputs to be gated. The outputs are mapped in the Pulsed Generator Configuration dialog box.

**Connect the Z5623A H81Pulse Test Set (optional) to the PNA front-panel port 1 loops as follows:**
<table>
<thead>
<tr>
<th>PNA</th>
<th>H81</th>
</tr>
</thead>
<tbody>
<tr>
<td>Src Out</td>
<td>Source IN</td>
</tr>
<tr>
<td>CPLR THRU</td>
<td>CPLR THRU</td>
</tr>
<tr>
<td>RCVR R1 IN</td>
<td>RCVR R1 Out</td>
</tr>
</tbody>
</table>

See Also

- PNA Front-panel loops
- PNA-X rear-panel
- PNA IF connectors
- 81110A Documentation
- Z5623A H81 Documentation

Using the Narrowband Pulsed Application

How to start the Narrowband Pulsed Application

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press MACRO</td>
<td>1. Click Utility</td>
</tr>
<tr>
<td>2. then [Pulse]</td>
<td>2. then Macro</td>
</tr>
<tr>
<td></td>
<td>3. then Pulse</td>
</tr>
</tbody>
</table>

See Also

- See programming commands to launch the Macro remotely.
- See how to write your own custom Narrowband Pulsed Application.

Keypad Data Entry

The PNA front-panel Numeric Entry and Navigation keys can be used for dialog box input. Also, a keyboard can be used to enter values, including alpha characters for prefixes (for example, u for usec.) After typing values, first press Enter, then press Tab to go to the next field.
Note: An error message may appear on the PNA stating that the response frequency has exceeded the maximum allowed frequency.

The Narrowband Pulsed Application may set the offset frequency (option 080) of the PNA to some value other than zero (the default value). If the stop frequency is set to the maximum of the PNA model, then the error message will appear.

To fix this, set the stop frequency to a value that is at least 2 KHz less than the maximum allowed. For example, if you have a 20 GHz PNA, and the stop frequency is set to 20 GHz, and the error message appears, then set the stop frequency to 19.999998 GHz.

See Block Diagram of IF Path / Pulse Generators / Source Modulation

Configure

You can configure more than one channel to make pulsed measurements, but the channels must use the same pulse generator settings.

Only the Agilent 81110A Pulse Generator is supported with the Narrowband Pulsed Application. Refer to the 81110A documentation for pulse repetition frequency and duty cycle capabilities.

See Also

- Configure Receivers
Converter Measurements

Edit / Undo  Pulse Application settings revert to those when Apply was last pressed.

Desired PRF and IFBW  Enter the DESIRED values. When Calculate is pressed, one or both of these values may change.

Pulse Repetition Frequency (PRF): Frequency of the pulses from the Pulse Generator.

Pulse Repetition Interval: 1/ PRF  Changes to either PRF or this setting changes both.

Receiver IF Bandwidth: IF Bandwidth of the PNA. Choose a setting from 1 Hz to 10 KHz.

Fixed PRF  When checked, (default setting) the Calculate algorithm will NOT adjust the PRF, but only change the IF Bandwidth.

Modulation/Gates  The Source Modulation and four PNA receiver gates can each have their own, or share, Pulse Generator outputs. Shared outputs have identical Width and Delay values. To configure and enable outputs, click Configure, then Pulse Generators to launch the Pulsed Generator Configuration dialog box.

Note: Option 036 and 037 limits the Source Modulation width to 117 ns.

Width  Pulse Width.

Delay  The delay that occurs before the pulse.

Duty Cycle  Calculated Duty Cycle of the source and each of the selected receivers. Updated when Calculate is pressed.

Pulse Mode On  When this box is checked, the PNA is enabled for Pulsed measurements. The PNA Status Bar annotation indicates the following:

- G  Internal IF gates enabled.
- F  Filtering for Pulsed Measurements enabled.

Apply  All selections are sent to the pulse generator and the active channel of the PNA.

Calculate  All selections are calculated and valid PRF and IFBW values are entered in their fields. If these settings are not acceptable, try changing the values you previously entered and click Calculate again. When acceptable values are attained, click Apply to send these values to the pulse generator and PNA.

Pulse Profile  Launches the Pulse Profile dialog box. Same as clicking View / Pulse Profile. If not available, check Pulse Mode ON, click Calculate, then Apply.

Minimize  Click to minimize the dialog box to make changes in the PNA application. To see the dialog again, select Macro, Pulse, or turn the Status Bar ON.

Save  All settings from the Narrowband Pulsed Application are saved in a *.ppf file. These settings are NOT
Recall  Restore settings from the specified *.ppf file that were previously saved.
Close  Closes the dialog box without saving changes.

How to configure Pulse Generators / Modulators and Receivers

From the Pulse App main dialog box

Learn about...

- **Configure Receiver Gain**
- **Converter Measurements**
- **No Pulse Generators** When checked, the Narrowband Pulsed Application does NOT attempt to communicate with internal or external pulse generators. This setting is used for troubleshooting purposes.
- **No SW Gating** When checked, the improved SW gating sensitivity is turned OFF. This setting is used for troubleshooting purposes.
# Pulsed Generator Configuration dialog box help

## Notes:

- See Block Diagram of PNA-X IF Path / Pulse Generators / Source Modulation
- For PNA-X Opt 224 (2-port Dual Source) select the 2-port Dual Source Configuration in the Path Configuration dialog to provide Pulse Modulation on both port 1 and port 2.
- This dialog may look different depending on the PNA model and number of receivers available.

Configures either the internal pulse generators (PNA-X models with relevant options), or Agilent 81110A Pulse Generator outputs. You can configure each 81110A Pulse Generator with either one or two 81111A output modules.

The Source Mod and four PNA receiver gates can each have their own, or shared, pulsed generators allowing identical Width and Delay values which are selected on the Main dialog.

To share an external generator output between one or more PNA inputs, use the same GPIB address and output module for each PNA input.

### Internal Pulse Gen Output (available ONLY on the PNA-X opt 025)

Specify the Pulse Gen (1 through 4) to use to modulate each of the PNA receiver IF gates or Sources.

### External Pulse Generator settings

- **GPIB Addr:** The GPIB address of the 81110A.
- **Output:** The output module of the 81110A.
- **Master:** The 81110A that uses the 10 MHz reference signal from the PNA.
- **Enabled:** Turns the pulse output ON.

### External Gate/Modulator settings

- **High:** Specify a 'TTL-High' voltage level
- **Low:** Specify a 'TTL-Low' voltage level
- **Ext Impedance:** Impedance of the modulator used to create the pulse.
- **Complement:** When this box is cleared, TTL HIGH is the pulse. When checked, TTL LOW is the pulse.

### Using Internal Modulators

When this box is checked, the voltage, impedance, and complement values are forced to settings that prevent damage to the internal modulator.

### Using Internal Pulse Generators

Makes the appropriate settings on this dialog available.

### Using Internal PNA gates

When this box is checked, the voltage, impedance, and complement values are forced to settings that prevent damage to the internal gates.
Receiver Gain Configuration dialog box help

See Block Diagram of PNA-X IF Path / Pulse Generators / Source Modulation

This dialog may look different depending on the PNA model and number of receivers available.

Sets the gain of each PNA receiver manually or automatically.

- **Auto** - The PNA selects the best gain level to make pulsed measurements.

Use the following to manually set the gain for each receiver.

- **Low** - about 0 dB of gain
- **Medium** - about 17 dB of gain
- **High** - about 24 dB of gain

The **PNA-X** has the following attenuation settings:

- **Low** - 30 dB of attenuation
- **Medium** - 15 dB of attenuation
- **Hi** - 0 dB of attenuation

Calibration in Pulse Mode

To perform a calibration in pulse mode (option H08), first configure and apply the pulse parameters (PRF, Pulse Width, Delays, IF gating, and so forth) **before** calibrating the system. This will ensure the PNA is configured properly during the calibration and measurement.

When performing **Unknown Thru** or **TRL calibrations**, ALL receivers must be gated. Otherwise, the error terms will not be correct after the calibration has completed. This can be accomplished by either having a separate pulse generator output for each of the IF gates, or by connecting pairs of the IF gates together with BNC-T’s. For example, if the pulse generator does not have enough outputs, then connect the R1 and R2 IF gates to the same pulse generator output. Also, connect the A and B IF gates to either separate outputs (recommended) or one output (reduces flexibility). The error terms will then be valid after the calibration is complete.

Pulse Profiling

Pulse profiling provides a time domain view of the pulse envelope. Profiling is performed using a measurement technique that "walks" a narrow receiver "snapshot" across the width of the pulse. This is analogous to using a camera to take many small snapshots of a wide image, then piecing them together to form a single, panoramic image.
Pulse Profiling can be performed using ratioed or unratioed measurements.

Pulse Profiling is performed at a single CW frequency.

**How to perform Pulse Profiling**

From the [Pulse App main dialog box](#),

Click the **Pulse Profile** button. or:

If this setting is unavailable, check **Pulse Mode ON**, click **Calculate**, then **Apply**.

---

**Pulse Profile dialog box help**

- [Learn about Pulse Profiling (scroll up)](#)
- [See Block Diagram of PNA-X IF Path / Pulse Generators / Source Modulation](#)

**Modulation / Gates**

These setting duplicate those found on the main [Pulse App dialog box](#).

In Pulse Profile, the Gate Delay settings (highlighted in yellow) are significant only with certain **Measurement Parameter** and **Couple Gates** settings.

**Time Parameters**
**Start, Stop** These two combine to make the window of the assembled pulse profile. To view the entire pulse, the start and stop values must be at least as wide as the Source Modulation **Width** plus **Delay** value.

**Step** Each consecutive snapshot is incremented by this value until the stop value is reached. Therefore, the number of points for the pulse profile measurement can be calculated as: (Stop - Start) / Step. The higher the number of points, the longer it takes to make the measurement.

**Measurement Parameter**

**CW Freq.** Frequency of the PNA source.

**Source Port** The PNA port supplying the source power. Only required for single receiver (unratioed) measurements.

**Param(eter)** Only those receiver gates (and relevant measurements) that are configured in **Pulsed Generator Configuration** are available.

**Note:** When a single receiver (unratioed) is selected, **Gate Delay** Settings (highlighted in yellow on above dialog image) are ignored.

If the reference receiver gate is NOT configured, the average of the Source Modulation pulse is used as the reference. For example: With **S21** Selected, but ONLY **B** receiver gate is configured, then...

![Diagram](source.png)

**B Gate is walked across the Source Modulation pulse.**

**Source Modulation pulse average is used as reference (not gated).**

**Coupled Gates** Used when the appropriate receiver gates are configured for your S-parameter measurement ONLY. This setting is ignored when a single receiver (**Param**) is selected.

- **Uncoupled** (box cleared) The reference gate is FIXED at the delay setting as the test gate is walked across the Source Modulation pulse as dictated by the **Time Parameter** settings.

![Diagram](source.png)

**For example:**

**S21** Selected, **B** and **R1** receiver gates configured, Gates **Uncoupled**

**B Gate is walked across the Source Modulation pulse.**

**R1 gate is fixed at pulse width and delay setting.**

- **Coupled** (box checked) The reference gate is walked synchronously with the test gate as dictated by the **Time Parameter** settings. Only the **difference** between the test and reference gate delay values is significant; NOT the absolute values.
For example:

S21 Selected, B and R1 receiver gates configured, Gates Coupled

B gate delay = 3 microseconds,

R1 gate delay = 2 microseconds

Difference = 1 microsecond

- B Gate is walked across the Source Modulation pulse.
- R1 gate is fixed at pulse width and delay setting.
- B gate leads R1 gate by 1 microsecond.

Data Format  Log Magnitude, Linear Magnitude, or Phase (only available if S-parameter selected).

Buttons

Show Gates  Allows you to change the receiver gating width and delay while looking at the results.

Apply Gate Settings  Click after making changes to gate settings.

Continuous Sweep  Check, then click Measure, to continuously measure pulse profiling.

Measure  Click to start the pulse profile measurement. Becomes Stop when continuously sweeping.

Marker to Delay  After making a measurement, you can drag the display marker to any point along the trace. Click this button and the marker time is entered into the Receiver Delay field on the main dialog box.

Save Data  Saves time domain data to the PNA hard drive in any of the following formats:

- Touchstone (*.s1p)
- Comma delimited (*.prn)
- Citifile (*.cti)

Learn more about these data formats

Signal Reduction versus Gate Width

Signal Reduction versus Gate Width

PRF = 1 MHz

The following two figures show the performance of the internal IF gates as the width is narrowed.
The following is a zoomed image of the shaded area (above).

The straight line shows the theoretical loss in dynamic range due to duty cycle effects when using narrowband detection.

The curved (red) line shows the actual measured performance of the gates.

The minimum gate width for <1dB deviation from theoretical is approximately 20ns.

See the specifications for the option H11 and option H08.
Pulsed Frequency Converter Measurements

The Narrowband Pulsed Application works with both FCA (option 083) and standard Frequency Offset (opt 080) measurements. On the Configure menu, check Converter Measurements. When checked, this setting prevents the Narrowband Pulsed Application from overwriting frequency offset values. This may limit the number of PRF and IFBW solutions that are returned when Calculate is pressed on the main Pulsed Application dialog box.

Note: Pulse Profiling can NOT be performed with frequency converter measurements.

Writing your own Narrowband Pulsed Application

You can use the Narrowband Pulsed Application or use an example program as a template for making your own Narrowband Pulsed Application.

The Narrowband Pulsed Application uses a custom .dll to perform the calculations that are necessary to make pulsed measurements. Use the COM Method below to send and return values to agilentpnapulsed.dll. Then use SCPI or COM commands to control the PNA.

<table>
<thead>
<tr>
<th></th>
<th>E836xC</th>
<th>PNA-X</th>
</tr>
</thead>
<tbody>
<tr>
<td>COM Example Program</td>
<td>E836x Create</td>
<td>PNA-X Create</td>
</tr>
<tr>
<td>SCPI Example Programs</td>
<td>None</td>
<td>Point-in-Pulse Pulse Profile</td>
</tr>
<tr>
<td>COM Methods</td>
<td>ConfigNarrowBand3</td>
<td>ConfigEnhancedNB2 ConfigEnhancedNBIFAtten</td>
</tr>
<tr>
<td>SCPI commands</td>
<td>SCPI</td>
<td>SCPI</td>
</tr>
<tr>
<td>COM commands</td>
<td>COM</td>
<td>COM</td>
</tr>
</tbody>
</table>

Install and Register the Pulsed .dll on your PC

To create your own Narrowband Pulsed Application, or run the Narrowband Pulsed Application from a remote PC, you must do the following:

1. Copy the following files from the PNA C:/program files/agilent/network analyzer/ to a directory on your PC.
   - agilentpnapulsed.dll
   - OffsetList.txt
   - prfbw.txt
2. To register the ActiveX DLL in Microsoft Windows Operating System:

   - From a command prompt on your PC, navigate to the directory where you copied the DLL.
   - Type: `regsvr32 agilentpnapulsed.dll` and press Enter

For Operating Systems other than Windows, see their associated help files to learn how to register DLL files.

---

Last Modified:

- 29-Apr-2009  Added Opt 036 and 037 note (A.08.60)
- 4-Sep-2008   Removed legacy content
- 2-Jul-2008   Added links to SCPI examples
**Wideband Pulsed Application**

**Note:** The PNA-X with Opt 025 and Opt 008 has an integrated Pulse Measurement Setup that makes this application no longer necessary on the PNA-X.

The Wideband Pulsed Application configures the PNA-X internal pulse generators and modulators for measuring pulsed S-parameters using the wideband mode detection technique.

The Wideband Pulse Application is designed to be used with the PNA-X with Opts 021, 022, and 025.

**Note:** Wideband Pulse application is NOT supported on the E836x and PNA-L models.

**See Also**

- To learn more about wideband detection, see Application Note 1408-12.
- See a Visual Basic example: Create a Wideband Pulsed Measurement using the PNA-X.
- Learn about the Narrowband Pulsed Application.

**Download and Install the Wideband Pulsed Application**

This application is installed and run as a macro on the PNA-X. Learn more about macros.


2. Click the download link

3. Save the downloaded file to the PNA hard drive

4. Double-click the downloaded file to install the Wideband Pulsed Application on the PNA.

5. The application is saved on the PNA at C:/Program Files/Agilent/Network Analyzer/Applications/WB_Pulse/Wideband_pulse.exe

6. Configure the macro. Learn how.

To learn more about Wideband pulsed application, click Help in the application.

___

**Last Modified:**

- 17-Mar-2010   Added PNA-X note
- 22-Jun-2007   MX New topic
Connecting the PNA to a PC

This document describes how to temporarily connect a PC to a PNA using a LAN cable. This is not necessary if your PNA is already connected to a network. This type of connection is for conveniently transferring large files, such as firmware, that may have already been downloaded and stored on the PC.

The PC can have any version of Windows (Windows 95 or newer.)

You will need the following:

- RJ-45 LAN crossover cable (or two normal cables with a suitable hub).
- Must be logged on the PNA with an Administrator User name and password.

Note: If your PC is on a domain, do not leave that domain by changing to a workgroup. This may prevent you from later rejoining your domain unless you involve your IT systems administrator. The following procedure will work regardless of whether or not you are on a domain and will not change any domain settings on the PC.

Procedure for All Operating Systems

1. Disconnect the PC from any existing LAN and connect it to the PNA using a crossover cable or hub. There is no need to turn it off to do this.

2. Find the current IP address of your PC. Open a DOS prompt (command window) and type `ipconfig`. This should then show your current IP address. Note this number. For this example, we will assume a PC IP address of 10.0.0.100. If it shows 0.0.0.0, you will have to assign an IP address. See procedure below.

3. On the PNA, click **File**, then **Exit** to close the PNA application.

4. Right-click on **My Network Places** and select **Properties**.

5. Right-click on **Local Area Connection** and select **Properties**.

6. Select **Internet Protocol (TCP/IP)** then click **Properties**.

7. Click **Use the following IP address** then enter an IP address that is ONE more or ONE less than that noted for the PC. Do not use a number that ends in 000. For this example, one could use either 10.0.0.101 or 10.0.0.99. We’ll assume the use of 101.

8. For **Subnet mask**, enter 255.255.255.0. Click **OK**, then **OK** again.

For Windows 95/98 systems, go to **Win95/98 step 9** below.

9. On the PC, open Windows Explorer. Click **Tools**, then **Map Network Drive**. Note the drive letter shown (for this example we'll assume it is "X".)

10. In **Folder** or **Path** type the IP address that was just entered for the PNA followed by C$ in the following format://10.0.0.101/C$.

11. In **Connect As** dialog, enter the PNA User name and Password. Unless it has been changed, this is either Administrator with NO password or PNA-Admin, Password Agilent. Click **OK**.
12. In Windows Explorer on the PC, you should now see a new drive letter entry (X) with the description of C$ on 10.0.0.101. The entire contents of the PNA C drive will now be available for reading or writing. Files can now be transferred by simply dragging them from one location to the other.

13. When you are done transferring files, disconnect the LAN cable connection between the two and reconnect your PC to the network if needed. If the PNA will never be connected to any network, the current network settings could remain. However it may be safer to reset the TCP/IP settings (changed in step 7) back to Obtain an IP address automatically.

If your PC currently has no IP address assigned

Follow steps 4 through 8 to add an IP address for the PC. The actual steps may vary slightly depending upon your operating system (NT4 uses Network Neighborhood and you must click on Protocols.) Assign the PC an IP address of 10.0.0.100. The last digits can be anything between 1 and 255, but do not alter the first 3 numbers (10.0.0.) When complete, reset the PC network configuration back to its previous settings.

Win95/98 Procedure for Accessing/Transferring Files

Because Windows 95/98 does not have the security features of NT-based operating systems (NT, Win2k, XP), the PC cannot access the drive on the PNA. To get around this limitation, any files that need to be transferred to/from the PC must be placed in a shared folder. The PNA can then read or write within this folder. This procedure will work for all versions of Windows.

Steps 1 through 8 are identical to the above and must be performed first.

9. On the PC, open Windows Explorer. Create a directory under the C drive named Shared. Right click the Shared folder name and select Sharing. Share the directory with full read/write permissions. If Sharing does not appear as a choice, then file sharing is not enabled. To enable file sharing,

   1. Right click Network Neighborhood, then click Properties.
   2. Click File and Print Sharing and enable give others access to my files
   3. Click OK, then OK again.
   4. Repeat the beginning of this step.

10. Copy the files to be transferred to the PNA to this shared folder.

11. On the PNA, open Windows Explorer. Click on Tools, Map Network Drive. Note the drive letter shown (for this example we?ll assume it is "X"). Uncheck “Reconnect at Logon” if it is currently checked.

9. Under the Folder entry, enter the IP address of the PC and the shared folder name in the following format: //10.0.0.100/Shared

10. The PNA should immediately connect to this folder and display its contents as drive "X". Files can now be read from, or written to, this shared directory (shown as Drive X.) Files can be transferred by simply dragging them from one location to the other.

11. When you are done transferring files, disconnect the LAN cable connection between the two and reconnect your PC to the network if needed. If the PNA will never be connected to any network, its current network settings could remain, however it is probably safer to reset the TCP/IP settings (changed in step 7) back to
Obtain an IP address automatically.
Easy versus Secure Configuration

When upgrading Firmware on the PNA, you encounter a Choose Configuration dialog box. This is used to determine the level of security set for the DCOM interface on the PNA. For more detailed information on the security settings for the DCOM interface, including a procedure for making these settings manually, see Configure for COM-DCOM Programming.

Comparison of the "Easy and More Secure" settings are as follows:

**Easy Connection:**

- No configuration of the PNA required for remote access to connect.
- Anyone on the local subnet can access the PNA remotely.
- People from other NT domains can connect to the PNA.

**More Secure:**

- Requires creating users on the PNA or adding the PNA to a domain
- An administrator of the PNA can specify users or groups that are allowed remote access to the PNA application
Changing Network Client

If your PC network uses Novell NetWare servers, a change must be made to the PNA setup before it can operate on your network. If you are unsure, ask your local IT department.

**Note:** Do NOT Uninstall "Client for Microsoft Networks". This will prevent proper operation of the PNA.

To remove "Client for Microsoft Networks" (Remove is different from Uninstall):

1. From the PNA Desktop, right-click **My Network Places**
2. Click **Properties**
3. Right-click **Local Area Connection**
4. Click **Properties**
5. Click (remove the check from) **Client for Microsoft Networks**

To install "Client Service for NetWare".

1. Click **Install**
2. In **Select Network Component Type**, make sure **Client** is selected
3. Click **Add**
4. In **Select Network Client**, make sure **Client Service for NetWare** is selected
5. Click **OK**.
Troubleshooting the PNA

By running a few checks, you can identify if the analyzer is at fault. Before calling Agilent Technologies or returning the instrument for service, please make the following checks.

- **Check the Basics**
- **PNA Application Terminates Unexpectedly**
- **Check Error Terms**
- **Check the Service Guide**

## Other Support Topics

### Check the Basics

A problem can often be solved by repeating the procedure you were following when the problem occurred. Before calling Agilent Technologies or returning the instrument for service, please make the following checks:

**Note:** Problems with the PNA application (slow or terminates unexpectedly) can be caused by a faulty Hard Disk Drive (HDD). For more information, see [Preventing PNA Hard Drive Problems](#) and [The PNA HDD Recovery Process](#).

1. Is there power at the power socket? Is the instrument plugged in?
2. Is the instrument turned on? Check to see if the front panel line switch and at least one of the LED rings around the test ports glows green. This indicates the power supply is on.
3. If you are experiencing difficulty with the front-panel keypad or peripherals, the USB bus may be overloaded. Remove the USB devices, restart the PNA, and reconnect the USB devices. See [Power-up](#).
4. If other equipment, cables, and connectors are being used with the instrument, make sure they are connected properly and operating correctly.
5. Review the procedure for the measurement being performed when the problem appeared. Are all the settings correct?
6. If the instrument is not functioning as expected, return the unit to a known state by pressing the [Preset](#) key.
7. Is the measurement being performed, and the results that are expected, within the [ specifications](#) and capabilities of the instrument?
8. If the problem is thought to be due to firmware, check to see if the instrument has the [latest firmware](#) before starting the troubleshooting procedure.
9. Check that the measurement calibration is valid. See [Accurate Measurement Calibrations](#) for more information.
10. If the necessary test equipment is available, perform the operator’s check and system verification in Chapter...
11. **Phase lock lost message** - This usually occurs when there is not enough source power to phase lock the PNA. It can occur during an errant FCA setup or Source Power Calibration. It can also occur if one of the front panel reference channel loops is not connected. Otherwise, this indicates a hardware problem.

**PNA Application Terminates Unexpectedly**

If an unexpected and irrecoverable error occurs, Agilent would like to know about it. The PNA attempts to save pertinent information about the state of the system. The **PNA does NOT send this information to Agilent**.

We respect the privacy of our customers. However, access to information that helps us improve the PNA is a benefit to both Agilent and you. Please take the time to contact us or email the saved information to **na_support@agilent.com**.

The following procedure shows how to do this:

1. A message box immediately appears on the screen containing the location of a directory. Please record this message. If you miss the message, you can find the directory location using the Windows Event Log: On the PNA, click Start, Settings, Control Panel, Administrative Tools, Event Viewer. Double-click the top line (most recent event). The location of the directory is seen in the Description.

2. A dialog box may appear on the screen (shown below) allowing you to add comments to help us replicate the crash.

3. Find the directory (described in Step 1) which contains the following files:

   - 835x.dmp which is the 835x.exe capturing the context in which the program crashed.
   - 835x.xml which reports some very basic information (exception code, OS version, and the list of modules loaded at the time of the crash and their respective version numbers).
   - 835xCrashLog.txt: The text file with your comments (described in Step 2), if submitted.

4. If your PNA is not connected to LAN or is not configured to send email, copy the files to a PC. Then, please email the files to **na_support@agilent.com**.
Check Error Terms

If you print the error terms at set intervals (weekly, monthly, and so forth), you can compare current error terms to these records. A stable, repeatable system should generate repeatable error terms over long time intervals, for example, six months. If a subtle failure or mild performance problem is suspected, the magnitude of the error terms should be compared against values generated previously with the same instrument and calibration kit. See the procedure for monitoring error terms.

- A long-term trend often reflects drift, connector and cable wear, or gradual degradation, indicating the need for further investigation and preventative maintenance. Yet, the system may still conform to specifications. The cure is often as simple as cleaning and gaging connectors or inspecting cables.

- A sudden shift in error terms reflects a sudden shift in systematic errors, and may indicate the need for further troubleshooting.

Consider the following while troubleshooting:

- All parts of the system, including cables and calibration devices, can contribute to systematic errors and impact the error terms.

- Connectors must be clean and gauged, and within specification for error term analysis to be meaningful. See the Chapter 2 in the PNA Service Guide for information on cleaning and gaging connectors.
  
  - Avoid unnecessary bending and flexing of the cables following measurement calibration, thus minimizing cable instability errors.
  
  - Use good connection techniques during the measurement calibration. The connector interface must be repeatable. See the PNA Service Guide for information on connection techniques.

- It is often worthwhile to perform the procedure twice (using two distinct measurement calibrations) to establish the degree of repeatability. If the results do not seem repeatable, check all connectors and cables.
Use error-term analysis to troubleshoot minor, subtle performance problems. See Chapter 3, "Troubleshooting," in the PNA Service Guide if a blatant failure or gross measurement error is evident.

Check the Service Guide


Last modified:

10/16/06    Added phase lock lost
PNA Error Messages

- 500 - 750 Calibrate
- 770 - 1000 Hardware
- 1000 - 1200 Measure
- 1281 - 1535 Parser
- 1536 - 1650 Display
- 1700 - 2000 Channel
- 2048 - 2200 General
- Standard SCPI Errors

Note: The EventID's listed below are provided for COM programming. For more information, see Working with PNA Events.

For more information on PNA error messages (see Error Messages).

Memory Overflow Error

Memory overflow. Trigger state set to Hold. Lower the IF bandwidth, or increase dwell or sweep time.

Severity: Informational

Further explanation: The measurement that you are currently making requires that data be stored faster than it can be processed. Very few customers will experience this situation.

Suggestions: To limit the amount of data to be stored, try lowering the IF Bandwidth, slow the sweep time, increase the dwell time, or limit the number of data points. There are many other settings that can be adjusted to solve this problem.

EventID:

Cal Errors

Message: 512

"A secondary parameter (power, IFBW, sweep time, step mode) of the calibrated state has changed."

Severity: Informational

Further explanation: The calibration is questionable when any of these secondary parameters change after the calibration is performed.

Suggestions: If you require an accurate measurement with the new settings, repeat the calibration.

EventID: 68020200 (hex)

Message: 513

"Calibration cannot be completed until you have measured all the necessary standards for your selected Cal Type."
Severity: Informational

Further explanation: You probably received this message because you attempted to turn correction on without first measuring all of the calibration standards

Suggestions: Finish measuring the cal standards

EventID: 68020201 (hex)

Message: 514
"Calibration set has been recalled using a file previously saved on an analyzer that had a different hardware configuration."

Severity: Informational

Further explanation:

Suggestions:

EventID: 68020202 (hex)

Message: 515
"Calibration is required before correction can be turned on. Channel number is <x>, Measurement is <x>.

Severity: Informational

Further explanation: There are no error correction terms to apply for the specified channel and measurement.

Suggestions: Perform or recall a calibration

EventID: 68020203 (hex)

Message: 516
"Critical parameters in your current instrument state do not match the parameters for the calibration set, therefore correction has been turned off. The critical instrument state parameters are sweep type, start frequency, frequency span, and number of points."

Severity: Informational

Further explanation: None

Suggestions: You can either recalibrate using the new settings or change back to the original setting that was used when the calibration was performed.

EventID: 68020204 (hex)

Message: 517
"Interpolation is turned off and you have changed the stimulus settings of the original calibration, so correction has been turned off."

Severity: Informational

Further explanation: The most accurate calibration is maintained only when the original stimulus settings are used.

Suggestions: If reduced accuracy is OK, set interpolation ON to allow stimulus setting changes.

EventID: 68020205 (hex)

Message: 518
"Interpolation is turned off and you have selected correction ON. Correction has been restored with the previous stimulus settings."

**Severity:** Informational

**Further explanation:** None

**Suggestions:** None

**EventID:** 68020206 (hex)

**Message:** 519

"Stimulus settings for your current instrument state exceeded the parameters of the original calibration, so correction has been turned off."

**Severity:** Informational

**Further explanation:** Correction data outside the stimulus settings does not exist.

**Suggestions:** Perform a broadband calibration, with increased numbers of points with interpolation ON, to maintain calibration over the widest possible stimulus frequency settings.

**EventID:** 68020207(hex)

**Message:** 520

"Cal Type is set to NONE for Channel <x>, Measurement <x>; please select Calibration menu or press Cal hard key."

**Severity:** Informational

**Further explanation:** A cal operation can not proceed until a calibration exists or the cal type is selected. This error can occur if the calibration can not be found. Also this error can happen if a calibration type is not specified before attempting to programatically execute cal acquisitions.

**Suggestions:** To find a calibration, select a Cal Set that contains the calibration needed for the current measurements. OR specify the cal type before beginning a calibration procedure.

**EventID:** 68020208 (hex)

**Message:** 521

"The measurement you set up does not have a corresponding calibration type, so correction has been turned off or is not permitted."

**Severity:** Informational

**Further explanation:** The calibration for the channel may apply only to certain S-Parameters. For example, a 1-Port calibration for S11 can not be applied to a 1-Port calibration applied to S22.

**Suggestions:** Select a calibration type, such as full 2-Port cal, that can be applied to all the measurements to be selected.

**EventID:** 68020209 (hex)

**Message:** 522

"The calibration type you selected cannot be set up."

**Severity:** Informational

**Further explanation:** "Please use the SCPI command ROUTe:PATH:DEFine:PORT <num>,<num> for full 2 port type port assignment."
Suggestions:
EventID: 6802020A (hex)

Message: 523
"The calibration path you selected cannot be set up because it is not valid for the current measurement."
Severity: Informational
Further explanation: "Please use the SCPI command ROUTe:PATH:DEFine:PORT <num>,<num> for full 2 port type port assignment related to your current measurement."
Suggestions:
EventID: 6802020B (hex)

Message: 524
"The source power calibration is complete."
Severity: Informational
Further explanation:

Suggestions:
EventID: 6802020C (hex)

Message: 525
"You have specified more than 7 standards for one or more calibration classes."
Severity: Informational
Further explanation: These have been truncated to 7 selections.
EventID: 6802020D (hex)

Message: 526
"No user calibration found for this channel."
Severity: Informational
Further explanation: A cal operation can not proceed until a calibration exists.
Suggestions: To find a calibration, you can select a Cal Set that contains the calibration needed for the current measurement.
EventID: 6802020E (hex)

Message: 527
"You do not need to acquire this standard for this calibration type."
Severity: Informational
Further explanation: This error can happen as a result of PROGRAMMATICALLY requesting the measurement of an un-needed calibration standard during a calibration procedure.
Suggestions: Check the specified cal type or eliminate the request for the measurement of the standard.
EventID: 6802020F (hex)
Message: 528
"Could not configure the Electronic Calibration system. Check to see if the module is plugged into the proper connector."
Severity: Informational
Further explanation: During an ECal operation, communication could not be established with the ECal module. The calibration will not be initiated until the presence of the ECal module is verified.
Suggestions: Verify the USB cable is connected properly. Disconnect and re-connect the cable to ensure the analyzer recognizes the module.
EventID: 68020210 (hex)

Message: 529
"DATA OUT OF RANGE: Design Limits Exceeded"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020211 (hex)

Message: 530
"EXECUTION ERROR: Could not open ECal module memory backup file"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020212 (hex)

Message: 531
"EXECUTION ERROR: Access to ECal module memory backup file was denied"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020213 (hex)

Message: 532
"EXECUTION ERROR: Failure in writing to ECal module memory backup file"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020214 (hex)

Message: 533
"EXECUTION ERROR: Failure in reading from ECal module memory backup file"
Severity: Error
Further explanation:
Suggestions:
**EventID:** E8020215 (hex)

**Message: 534**
"EXECUTION ERROR: Array index out of range"

Severity: Error
Further explanation:
Suggestions:
**EventID:** E8020216 (hex)

**Message: 535**
"EXECUTION ERROR: Arrays wrong rank"

Severity: Error
Further explanation:
Suggestions:
**EventID:** E8020217 (hex)

**Message: 536**
"EXECUTION ERROR: CPU"

Severity: Error
Further explanation:
Suggestions:
**EventID:** E8020218 (hex)

**Message: 537**
"EXECUTION ERROR: Cannot ERASE module"

Severity: Error
Further explanation:
Suggestions:
**EventID:** E8020219 (hex)

**Message: 538**
"EXECUTION ERROR: Cannot WRITE module"

Severity: Error
Further explanation:
Suggestions:
**EventID:** E802021A (hex)
Message: 539
"EXECUTION ERROR: Entry Not Found"
Severity: Error
Further explanation:
Suggestions:
EventID: E802021B (hex)

Message: 540
"EXECUTION ERROR: Invalid command while system is busy"
Severity: Error
Further explanation:
Suggestions:
EventID: E802021C (hex)

Message: 541
"Electronic Cal: Unable to orient ECal module. Please ensure the module is connected to the necessary measurement ports."
Severity: Error
Further explanation: There is no RF connection to the ECal module during a calibration step. An ECal orientation measurement has been attempted but the signal was not found.
Suggestions: Connect the ECal module RF connections to ports specified for the calibration step. The ECal module typically requires at least -18dBm for measurements. If your measurement requires the power level to be less than that, clear the Do orientation checkbox to bypass the automatic detection step.
EventID: E802021D (hex)

Message: 542
"EXECUTION ERROR: NO SPACE for NEW CAL, DELETE A CAL"
Severity: Error
Further explanation:
Suggestions:
EventID: E802021E (hex)

Message: 543
"EXECUTION ERROR: No More Room"
Severity: Error
Further explanation:
Suggestions:
EventID: E802021F (hex)
Message: 544
"EXECUTION ERROR: Other array error"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020220 (hex)

Message: 545
"EXECUTION ERROR: Ranks not equal"
Severity: Error
Further explanation:
Suggestions:
EventID: E8020221 (hex)

Message: 546
"EXECUTION ERROR: Too few CONSTANT ranks"
Severity: Error
EventID: E8020222 (hex)

Message: 547
"EXECUTION ERROR: Too few VARYing ranks"
Severity: Error
EventID: E8020223 (hex)

Message: 548
"EXECUTION ERROR: Unknown error"
Severity: Error
EventID: E8020224 (hex)

Message: 549
"EXECUTION ERROR: ecaldrv.dll bug or invalid module #"
Severity: Error
EventID: E8020225 (hex)

Message: 550
"EXECUTION ERROR: unexpected error code from ecal driver"
Severity: Error
EventID: E8020226 (hex)

Message: 551
"EXECUTION ERROR: unexpected internal driver error"
Severity: Error
EventID: E8020227 (hex)

**Message: 552**
"HARDWARE ERROR: Can't access ECal Interface Module"
Severity: Error
EventID: E8020228 (hex)

**Message: 553**
"HARDWARE ERROR: Can't release LPT port, reboot"
Severity: Error
EventID: E8020229 (hex)

**Message: 554**
"HARDWARE ERROR: VNA Error"
Severity: Error
EventID: E802022A (hex)

**Message: 555**
"HARDWARE ERROR: not enough data read from ECal module"
Severity: Error
EventID: E802022B (hex)

**Message: 556**
"OPERATION ABORTED BY HOST COMPUTER"
Severity: Error
EventID: E802022C (hex)

**Message: 557**
"OPERATION ABORTED BY USER"
Severity: Error
EventID: E802022D (hex)

**Message: 558**
"OUT OF MEMORY"
Severity: Error
EventID: E802022E (hex)

**Message: 559**
"QUERY INTERRUPTED: Message(s Abandoned"

Severity: Error
EventID: E802022F (hex)

Message: 560
"QUERY UNTERMINATED: INCOMPLETE PROGRAM Message"

Severity: Error
Further explanation:
Suggestions:
EventID: E8020230 (hex)

Message: 561
"QUERY UNTERMINATED: NOTHING TO SAY"

Severity: Error
Further explanation:
Suggestions:
EventID: E8020231 (hex)

Message: 562
"QUEUE OVERFLOW"

Severity: Error
EventID: E8020232 (hex)

Message: 563
"SETTINGS CONFLICT: ADDITIONAL STANDARDS ARE NEEDED"

Severity: Error
EventID: E8020233 (hex)

Message: 564
"SETTINGS CONFLICT: Adapter Cal is NOT possible"

Severity: Error
EventID: E8020234 (hex)

Message: 565
"SETTINGS CONFLICT: COMMAND OUT OF SEQUENCE"

Severity: Error
EventID: E8020235 (hex)

Message: 566
"SETTINGS CONFLICT: Cal STOPPED - VNA SETUP CHANGED"
Severity: Error
EventID: E8020236 (hex)

Message: 567
"SETTINGS CONFLICT: Calibration is NOT in progress"
Severity: Error
EventID: E8020237 (hex)

Message: 568
"SETTINGS CONFLICT: Can't find specified GPIB board"
Severity: Error
EventID: E8020238 (hex)

Message: 569
"SETTINGS CONFLICT: Can't find/load gpib32.dll"
Severity: Error
EventID: E8020239 (hex)

Message: 570
"SETTINGS CONFLICT: Can't find/load sicl32.dll"
Severity: Error
EventID: E802023A (hex)

Message: 571
"SETTINGS CONFLICT: Can't initialize VNA (bad address?)"
Severity: Error
EventID: E802023B (hex)

Message: 572
"SETTINGS CONFLICT: Can't load LPT port driver or USB driver DLL"
Severity: Error
EventID: E802023C (hex)

Message: 573
"SETTINGS CONFLICT: Invalid Calibration Sweep Mode."
Severity: Error
EventID: E802023D (hex)

Message: 574
"SETTINGS CONFLICT: Invalid Calibration Type"
Severity: Error
EventID: E802023E (hex)

Message: 575
"SETTINGS CONFLICT: Invalid Calibration"

Severity: Error
EventID: E802023F (hex)

Message: 576
"SETTINGS CONFLICT: Invalid GPIB board number specified"

Severity: Error
EventID: E8020240 (hex)

Message: 577
"SETTINGS CONFLICT: Invalid GPIB board type specified"

Severity: Error
EventID: E8020241 (hex)

Message: 578
"SETTINGS CONFLICT: Invalid Module Status"

Severity: Error
EventID: E8020242 (hex)

Message: 579
"SETTINGS CONFLICT: Invalid States"

Severity: Error
EventID: E8020243 (hex)

Message: 580
"SETTINGS CONFLICT: LPT port must be between 1 and 4"

Severity: Error
EventID: E8020244 (hex)

Message: 581
"Could not configure the Electronic Calibration system. Check to see if the module is properly connected."

Severity: Error
EventID: E8020245 (hex)

Message: 582
"SETTINGS CONFLICT: Specified LPT port does not exist"
Severity: Error
EventID: E8020246 (hex)

Message: 583
"SETTINGS CONFLICT: Use frequency domain for cal"
Severity: Error
EventID: E8020247 (hex)

Message: 584
"SETTINGS CONFLICT: Use step sweep type for cal."
Severity: Error
EventID: E8020248 (hex)

Message: 585
"SETTINGS CONFLICT: VNA address must be between 0 and 30"
Severity: Error
EventID: E8020249 (hex)

Message: 586
"SETTINGS CONFLICT: Wrong LPT port driver or USB driver DLL"
Severity: Error
EventID: E802024A (hex)

Message: 587
"SYNTAX ERROR: ECAL:DELAY command must have 2 numbers"
Severity: Error
EventID: E802024B (hex)

Message: 588
"SYNTAX ERROR: INCORRECT SYNTAX"
Severity: Error
EventID: E802024C (hex)

Message: 589
"SYNTAX ERROR: UNKNOWN COMMAND"
Severity: Error
EventID: E802024D (hex)

Message: 590
"Wrong port of module in RF path"
Severity: Error
EventID: E802024E (hex)

Message: 591
"User characterization not found in module"
Severity: Error
EventID: E802024F (hex)

Message: 592
Severity: Informational
"No source power calibration found for the channel and source port of the current measurement."
Further explanation: You tried to turn on source power cal but there is no source power cal data.
Suggestions: Perform a source power calibration
EventID: 68020250 (hex)

Message: 593
Severity: Informational
"A source power calibration sweep was not performed, so there is no correction for the channel and source port of the current measurement."
Further explanation: You tried to turn on source power cal but there is incomplete source cal data.
Suggestions: Perform a complete source power calibration
EventID: 68020251 (hex)

Message: 594
Severity: Informational
"A new trace could not be added to the active window for viewing the source power cal sweep, because it would have exceeded the limit on number of traces/window. Please remove a trace from the window before proceeding with source power cal."
Further explanation: The source power cal attempts to add a data trace to the active window. The active window already contains four traces.
Suggestions: Make the active window contain less than four traces.
EventID: 68020252 (hex)

Message: 595
Severity: Informational
"A new measurement could not be added for performing the source power cal sweep, because the limit on number of measurements has been reached. Please remove a measurement before proceeding with source power cal."
Further explanation: The source power cal attempts to add a measurement. The PNA already has the maximum number of measurements.
Suggestions: Delete a measurement.
EventID: 68020253 (hex)
**Message: 596**  
**Severity:** Informational  
"The calibration power value associated with the source power calibration of Port %1 on Channel %2 was changed with the calibration on. The calibration was not turned off, but the power value might no longer represent the calibration."

**Further explanation:** The source power cal accuracy is questionable.  
**Suggestions:** If high accuracy is required, perform another source power calibration.  
**EventID:** 68020254 (hex)

**Message: 597**  
**Severity:** Informational  
- Message that is passed from the power meter driver for a source power calibration. -

**Further explanation:** This error is generated by the power meter driver and passed through the PNA.  
**EventID:** 68020255 (hex)

**Message: 598**  
"During the acquisition of the sliding load standard, the slide was not properly moved to perform a circle fit. The standard's raw impedance was used to determine the directivity for one or more points."

**Severity:** Informational  
**Further Explanation:** To accurately characterize the standard, the sliding load must be move sufficiently to ensure enough samples around the complex circle or Smith Chart. Under-sampling will cause an inaccurate result.  
**Suggestions:** For best results when using a sliding load, be sure to use multiple slide positions that cover the full range of movement from front to back of the slot.  
**EventID:** 68020256 (hex)

**Message: 599**  
"This feature requires an unused channel, but could not find one. Please free up a channel and try again."

**Severity:** Informational  
**Further Explanation:** You attempted to view an item within a calset. However, the calset viewer requires that the result be displayed in a channel that is not currently in use. All the channels are currently used. The view can not display the requested item.  
**Suggestions:** You must delete at least one channel that is currently in use.  
**EventID:** 68020257 (hex)

**Message: 600**  
"Interpolation of the original calibration is not allowed since it was performed using Segment Sweep. Correction has been turned off."

**Severity:** Informational  
**EventID:** 68020258 (hex)

**Message: 601**
"Cal preferences saved. Cal preference settings can be changed from the 'Cal Preferences' drop down Cal menu."

Severity: Informational

Further explanation: See Save Preference

EventID: 68020259 (hex)

Message: 608

"CalType not set."

Severity: Error

Further explanation: A cal operation can not proceed until a calibration exists or the proper cal type is selected.

Suggestions: This error can happen if the calibration can't be found. To find a calibration, you can select a Cal Set that contains the calibration needed for the current measurements. This error can also happen if a calibration type is not specified before attempting to programmatically execute cal acquisitions. Specify the cal type before beginning a calibration procedure.

EventID: E8020260 (hex)

Message: 609

"The Calibration feature requested is not implemented."

Further explanation: The specified cal type can be one of many choices. For example, response calibrations require single standards, 1-Port calibrations require 3 standards, and 2-Port calibrations require up to 12 standards.

Suggestions: Be sure to measure only the standards needed for the specified cal type.

EventID: E8020261 (hex)

Message: 610

"The Calibration Class Acquisition requested is not valid for the selected Calibration Type. Please select a different acquisition or a different Calibration Type."

EventID: E8020262 (hex)

Message: 611

"The Calibration Standard data required for the selected caltype was not found."

Severity: Error

Further explanation: An unsuccessful attempt was made to retrieve a specified standard from the raw measurement buffer. The buffer should contain the raw measurements of cal standards stored during a calibration procedure.

Suggestions: Be sure the requested standard is required for the current cal type. Not all standards are needed for all cal types.

EventID: E8020263 (hex)

Message: 612

"The Error Term data required for the selected caltype was not found."

Severity: Error

Further explanation: An unsuccessful attempt was made to retrieve a specified error term from the error
correction buffer. The buffer should contain the error correction arrays for the current calibration.

Suggestions: Be sure the requested error term is required for the current cal type. Not all error terms are needed for all cal types.

EventID: E8020264 (hex)

Message: 613
The Calibration data set was not found.

Severity: Error

Further explanation: An unsuccessful attempt to access a cal set has been made. This may indicate a calset has been deleted or has been corrupted.

Suggestions: Try again or select another cal set. If the cal set appears in the cal set list, it may need to be deleted.

EventID: E8020265 (hex)

Message: 614
"The specified measurement does not have a calibration valid for Confidence Check. Please select a different measurement, or recall or perform a different Calibration Type."

Severity: Error

Further explanation: The measurement choice is prevented so that calibration will not be turned off. Not all cal types support all measurements. For example, an 1-Port cal on S11 can not be used to calibrate an S12 measurement. When a measurement is selected that does not have a calibration which can be applied, an informational message is displayed and calibration is turned off.

Suggestions: Use a full 2-Port calibration to be compatible with any S-Parameter.

EventID: E8020266 (hex)

Message: 615
" New calset created."

Severity: Informational message.

Further explanation: The newly created cal set will be automatically named and time stamped. If this is the beginning of a calibration procedure, the cal set will not be stored to memory until the calibration has completed successfully. The new cal set will be deleted if the calibration is canceled or does not otherwise complete successfully.

Suggestions: Informational

EventID: 68020267

Message: 617
The calset file: &lt;x&gt; appears to be corrupted and cannot be removed. Exit the application, remove the file, and restart.

Severity: Error

Suggestions: The cal set file is stored in the application home directory C:/Program Files/Agilent/Network Analyzer/PNACalSets.dat. Remove this file, then restart the application.

EventID: E8020269 (hex)
**Message: 634**

"The calset file: <x> load failed."

**Severity:** Error

**Further explanation:** The calset file contains a collection of calsets. The file resides on the hard drive.

**Suggestions:** Try restarting the application. If the failure persists, you may have to delete the cal set data file and restart the application. The cal set file is stored in the application home directory. C:/Program Files/Agilent/Network Analyzer/PNACalSets.dat. Remove this file, then restart the application.

**EventID:** E802027A (hex)

**Message: 635**

"The calset file: <x> save failed."

**Severity:** Error

**Further explanation:** The file operation detected an error. The save operation was aborted.

**Suggestions:** Retry.

**EventID:** E802027B (hex)

**Message: 636**

"A calset was deleted."

**Severity:** Informational

**Further explanation:** One of the calsets has been successfully deleted from the collection of calsets available. This can happen as the result of a user request or intentional operation.

**Suggestions:** None

**EventID:** 6802027C (hex)

**Message: 637**

"The version of the calset file: <x> is not compatible with the current instrument."

**Severity:** Error

**Further explanation:** A versioning error can prevent a calset from being used. This can happen as a result of instrument firmware upgrades.

**Suggestions:** If the versioning error is the result of firmware upgrade, you will have to re-install the old version of firmware to re-use the calset file. Or you can re-create the calsets with the current version of firmware.

The cal set file is stored in the application home directory C:/Program Files/Agilent/Network Analyzer/PNACalSets.dat. Remove this file, then restart the application.

**EventID:** E802027D (hex)

**Message: 638**

"Incompatible CalSets found: <x> of <y> stored calsets have been loaded."

**Severity:** Error

**Further explanation:** Errors were found on some of the calsets stored in the calset file. The errors may have been caused by versioning issues that may have corrupted the various calset keys.
Suggestions: Use the calset viewer to look at the contents of calset files. Delete the files that are corrupted.

EventID: 6802027E (hex)

**Message: 639**
"The Calset file: <x> was not found. A new file has been created."

**Severity:** Informational

**Further explanation:** The calset file should be stored on the hard drive. When the application is started, a search is done and the file is loaded if it can be found. If the file is not found, the analyzer will create a new file and display this message.

**Suggestions:** None

EventID: 6802027F (hex)

**Message: 640**
"The Calset specified is currently in use."

**Severity:** Error

**Further explanation:** This may indicate a conflict between multiple calset users attempting calibration tasks.

**Suggestions:** Save the instrument state. Preset the analyzer and recall the instrument state. This may abort any processes that may be in progress.

EventID: E8020280 (hex)

**Message: 641**
"The calset specified has not been opened."

**Severity:** Error

**Further explanation:** Multiple users may be attempting to access the calset.

**Suggestions:** Close multiple calset users so that only one user will access the calset.

EventID: E8020281 (hex)

**Message: 642**
"The maximum number of cal sets has been reached. Delete old or unused cal sets before attempting to create new ones."

**Severity:** Error

**Suggestions:** You may also delete the calsets data file.

The cal set file is stored in the application home directory. C:/Program Files/Agilent/Network_Analyzer/PNACalSets.dat. Remove this file, then restart the application.

EventID: E8020282 (hex)

**Message: 643**
The requested power loss table segment was not found.

**Severity:** Error

EventID: E8020283 (hex)
Message: 644
"A valid calibration is required before correction can be turned on."

Severity: Error

Further explanation: This usually indicates a calibration procedure has not run to completion or that the selected measurement does not have a valid calibration available from within the currently selected cal set.

Suggestions: To find a calibration, you can select a Cal Set that contains the calibration needed for the current measurements. This error can happen if a calibration type is not specified before attempting to programmatically execute cal acquisitions. Specify the cal type before beginning a calibration procedure.

EventID: E8020284 (hex)

Message: 645
The cal data for <x> is incompatible and was not restored. Please recalibrate."

Severity: Warning

Further explanation: None

Suggestions: None

EventID: A8020285 (hex)

Message: 646
"CalSet not loaded, version is too new."

Severity: Error

Further explanation: An old version of firmware is attempting to run with a new calset version. The version is incompatible.

Suggestions: The calset can be removed. You may also delete the calsets data file if you are migrating between various firmware revisions often and you would like to avoid this error. The cal set file is stored in the application home directory. C:/Program Files/Agilent/Network Analyzer/PNACalSets.dat. Remove this file, then restart the application.

EventID: E8020286 (hex)

Message: 647
"Custom cal type not found."

Severity: Error

Further explanation: None

Suggestions: None

EventID: E8020287 (hex)

Message: 648
"Custom correction algorithm defers to the client for interpolation."

Severity: Informational

EventID: 68020288 (hex)

Message: 649
"Custom cal dll threw an exception."

Severity: Error
EventID: E8020289 (hex)

Message: 650
"Could not load the ecal.dll library"

Severity: Error
EventID: E802028A (hex)

Message: 656
"The argument specified is not a valid cal type."

Severity: Error
EventID: E8020290 (hex)

Message: 657
"The function found existing interpolated data"

Severity: Informational
EventID: 68020291 (hex)

Message: 658
"The function computed new interpolation values."

Severity: Informational
EventID: 68020292 (hex)

Message: 659
"The source power measurement failed."

Severity: Error
Suggestions: Please check GPIB, power meter settings and sensor connections.
EventID: E8020293 (hex)

Message: 660
"Duplicate session found. Close session and retry."

Severity: Error
EventID: E8020294 (hex)

Message: 661
"The session does not exist. Open the session and try again."

Severity: Error
Further explanation:
EventID: E8020295 (hex)
Message: 662
"Attempt to launch a custom calibration failed."
Severity: Error
Further explanation:
EventID: E8020296 (hex)

Message: 663
"Request to measure a cal standard failed."
Severity: Error
Further explanation: Please ensure you are requesting to measure standards which are defined for this calibration.
EventID: E8020297 (hex)

Message: 664
"Since Electronic Calibration Kit is selected, Mechanical Cal Kit parameter cannot be changed."
Severity: Error
Further explanation:
EventID: E8020298 (hex)

Message: 665
"Frequencies of the active channel are below minimum or above maximum frequencies of the ECal module factory characterization."
Suggestions: Change the channel frequencies, or select another ECal module.
Severity: Error
EventID: E8020299 (hex)

Message: 666
"Calset chosen for characterizing the ECal Module Ports %1 does not contain a calibration for PNA Ports %2."
Severity: Error
Suggestions: Go back to select another calset or to perform another cal.
EventID: E802029A (hex)

Message: 667
"ECal module only has sufficient memory remaining to store a maximum of %1 points in User Characterization %2."
Severity: Error
Suggestions: Decrease your number of points, or choose to overwrite another user characterization.
EventID: E802029B (hex)

Message: 668
Input values are non-monotonic. Cannot interpolate.
Severity: Error
EventID: E802029C (hex)

Message: 669
Interpolation target is out of range. Cannot interpolate.

Severity: Error
EventID: E802029D (hex)

Message: 670
Guided Calibration Error: <>

Severity: Error
EventID: E802029E (hex)

Message: 671
The first call to the guided calibration interface must be Initialize.

Severity: Error
EventID: E802029F (hex)

Message: 672
The selected thru cal method was not recognized.

Severity: Error
EventID: E80202A0 (hex)

Message: 673
Could not generate the error terms.

Severity: Error
EventID: E80202A1 (hex)

Message: 674
Guided calibration must be performed on the active channel

Severity: Error
EventID: E80202A2 (hex)

Message: 675
You can not start using calibration steps until you have successfully called generate steps.

Severity: Error
EventID: E80202A3 (hex)

Message: 676
The step number given is out of range. Step numbers should be between 1 and the number of steps. 0 is not a
valid step number.

**Severity:** Error

**EventID:** E80202A4 (hex)

**Message:** 677
A calset was selected for channel: <n> without restoring stimulus.

**Severity:** Informational

**EventID:** 680202A5 (hex)

**Message:** 678
A calset was selected for channel: <n> restoring stimulus.

**Severity:** Informational

**EventID:** 680202A6 (hex)

**Message:** 679
The selected calset stimulus could not be applied to the channel.

**Severity:** Informational

**EventID:** 680202A7 (hex)

**Message:** 680
You attempted to measure power at a frequency outside the frequency range defined for the specified power sensor. Select another sensor or adjust the range for this sensor.

**Severity:** Error

**EventID:** E80202A8 (hex)

**Message:** 681
Specified frequency is outside the frequency ranges currently defined for the power meter's sensors.

**Severity:** Error

**EventID:** E80202A9 (hex)

**Message:** 682
Additional Calibration Standards need to be acquired in order to calibrate over the entire frequency range currently being measured.

**Severity:** Informational

**EventID:** 680202AA (hex)

**Message:** 683
The PNA failed to convert cal kits for use by unguided calibrations. The recommended action is to restore Cal Kit defaults.

**Severity:** Error

**EventID:** E80202AB (hex)
Message: 684
The PNA failed to convert cal kits for use by unguided calibrations. CalKit defaults have been restored.
Severity: Error
EventID: E80202AC (hex)

Message: 685
Power meter is reserved by a source power cal acquisition already in progress.
Severity: Error
EventID: E80202AD (hex)

Message: 686
Source power calibration has not been performed or uploaded for the specified channel and source port.
Severity: Error
EventID: E80202AE (hex)

Message: 687
Source power calibration data array size for the specified channel and source port does not match its associated stimulus number of points.
Severity: Error
EventID: E80202AF (hex)

Message: 688
Source power calibration of Port <n> on Channel <n> was turned off because the correction array no longer exists.
Severity: Error
EventID: E80202B0 (hex)

Message: 689
This command can only be used on a measurement created with a specified calibration loadport.
Severity: Error
EventID: E80202B1 (hex)

Message: 690
Interpolation is turned off and you have changed the stimulus settings of the original calibration, so correction has been turned off.
Severity: Error
EventID: E80202B2 (hex)

Message: 691
Stimulus settings for your current instrument state exceeded the parameters of the original calibration, so correction has been turned off.
Severity: Error
EventID: E80202B3 (hex)

Message: 692
Fixturing: the requested S2P file cannot be read. Possible formatting problem.
Severity: Error
EventID: E80202B4 (hex)

Message: 693
Fixturing: the requested S2P file cannot be opened.
Severity: Error
EventID: E80202B5 (hex)

Message: 694
Fixturing: the requested S2P file cannot be interpolated. This is usually because the frequency range in the file is a subset of the current channel frequency range.
Severity: Error
EventID: E80202B6 (hex)

Message: 695
Cal Registers can only be used by one channel: the channel conveyed in the name of the cal register. The name cannot be changed.
Severity: Error
Further explanation: See Cal Registers
EventID: E80202B7 (hex)

Message: 696
Fixturing: cannot be enabled with Response Calibrations and has been turned off.
Severity: Error
EventID: E80202B8 (hex)

Message: 697
The selected calibration cannot be performed for this measurement.
Severity: Error
EventID: E80202B9 (hex)

Message: 698
Fitting: RemoveAllConnectors() should be called prior to calling AddConnector after a fit has been attempted.
Severity: Error
EventID: E80202BA (hex)
Message: 699
An attempt was made to acquire calibration data before the system was properly initialized.

Severity: Error
EventID: E80202BB (hex)

Message: 700
Use IGuidedCalibration for multiport calibration types.

Severity: Error
EventID: E80202BC (hex)

Message: 701
Guided calibration requires number of thru measurement paths be at least equal to the number of calibration ports minus 1.

Severity: Error
EventID: E80202BD (hex)

Message: 702
A thru path was specified that includes a port which the calibration was not specified to include.

Severity: Error
EventID: E80202BE (hex)

Message: 703
One or more of the ports to be calibrated was not found in the set of specified thru paths.

Severity: Error
EventID: E80202BF (hex)

______________________________

Hardware Errors

______________________________

Message: 770
Input power too high. Source power is off.

Severity: Warning
EventID: A8030302 (hex)

Message: 771
Source power restored.

Severity: Informational
EventID: 68030303 (hex)

Message: 772
"The spampnp.sys driver is not working. Check system hardware. ! Data will be simulated. !"
Severity: Error
Further explanation: The Network Analyzer application cannot locate the DSP board. Hardware or a driver may be malfunctioning. This is also common when attempting to run the Network Analyzer on a workstation.
EventID: E8030304 (hex)

Message: 773
"Instrument Serial Bus Not Working."
Severity: Error
Further explanation: The instrument EEPROM appears to contain either all ones or all zeros. A serial bus hardware failure prevents reading the EEPROM.
EventID: E8030305 (hex)

Message: 784
Unleveled, source <n>, out <n>.
Severity: Error
Further explanation: The PNA was unable to set the power on port <n> to the desired level

Message: 848
"Phase lock lost"
Severity: Error
Further explanation: The instrument source was not able to lock properly. This can be the result of broken hardware, poor calibration, or bad EEPROM values.
Suggestions: Perform source calibration. Click System / Service / Adjustments / Source Calibration
EventID: E8030350 (hex)

Message: 849
Phaselock restored.
Severity: Success
EventID: 0x28030351 (hex)

Message: 850
"Unknown hardware error."
Severity: Error
Further explanation: Hardware malfunctioned prevents communication with the DSP.
EventID: E8030352 (hex)

Message: 851
DSP communication lost.
Severity: Error
EventID: E8030353 (hex)
Message: 852
RF power off.
Severity: Error
EventID: E8030354 (hex)

Message: 853
RF power on.
Severity: Success
EventID: 28030355 (hex)

Message: 854
Hardware OK.
Severity: Success
EventID: 28030356 (hex)

Message: 855
"Source unleveled."
Severity: Error
Further explanation: The source was unable to properly level at the requested power. The indicated power may not be accurate.
Suggestions: Try a different power level. Recalibrate source, if problem persists.
EventID: E8030357 (hex)

Message: 856
Source leveled.
Severity: Success
EventID: 28030358 (hex)

Message: 857
Input overloaded.
Severity: Error
EventID: E8030359 (hex)

Message: 858
Input no longer overloaded.
Severity: Success
EventID: 2803035A (hex)

Message: 859
"Yig calibration failed."
Severity: Error
Further explanation: Internal self-calibration of YIG oscillator tuning failed.
EventID: E803035B (hex)

Message: 860
Yig calibrated.
Severity: Success
EventID: 2803035C (hex)

Message: 861
"Analog ramp calibration failed."
Severity: Error
Further explanation: Internal analog sweep ramp calibration has failed.
EventID: E803035D (hex)

Message: 862
Analog ramp calibrated.
Severity: Success
EventID: 2803035E (hex)

Message: 863
Source temperature high.
Severity: Error
EventID: E803035F (hex)

Message: 864
Source temperature OK.
Severity: Success
EventID: 28030360 (hex)

Message: 865
"EEPROM write failed."
Severity: Error
Further explanation: Attempt to store calibration data to EEPROM has failed. There is a possible hardware failure.
EventID: E8030361 (hex)

Message: 866
EEPROM write succeeded.
Severity: Success
EventID: 28030362 (hex)
**Message: 867**
Attempted I/O write while port set to read only.

**Severity:** Error

**Further explanation:** Attempt to write to an I/O data port while the port set to input/read only.

**Suggestions:** Set data port to write/output before attempting to write to port.

**EventID:** E8030363 (hex)

**Message: 868**
" Attempted I/O read from write only port.

**Severity:** Error

**Further explanation:** Attempt to read from an I/O data port while the port set to output/write only.

**Suggestions:** Set data port to read/input before attempting to read from port.

**EventID:** E8030364 (hex)

**Message: 869**
Invalid hardware element identifier.

**Severity:** Error

**EventID:** E8030365 (hex)

**Message: 870**
Invalid gain level setting.

**Severity:** Error

**EventID:** E8030366 (hex)

**Message: 871**
Device driver was unable to allocate enough memory. Please try rebooting.

**Severity:** Error

**EventID:** E8030367 (hex)

**Message: 872**
DSP Error. Please Contact Agilent Support. Technical Information: DSP Type 1

**Severity:** Error

**EventID:** E8030368 (hex)

**Message: 873**
DSP Error. Please Contact Agilent Support. Technical Information: DSP Type 2

**Severity:** Error

**EventID:** E8030369 (hex)
**Message: 874**
DSP Error. Please Contact Agilent Support. Technical Information: DSP Type 3
Severity: Error
EventID: E803036A (hex)

**Message: 875**
DSP Error. Please Contact Agilent Support. Technical Information: DSP Type 4
Severity: Error
EventID: E803036B (hex)

**Message: 876**
DSP Error. Please Contact Agilent Support. Technical Information: DSP Type 5
Severity: Error
EventID: E803036C (hex)

**Message: 910**
The trigger connection argument was not recognized as valid by the firmware.
Severity: Error
EventID: 0xE803038E (hex)

**Message: 911**
The trigger connection specified does not support this trigger behavior
Severity: Error
EventID: E803038F (hex)

**Message: 912**
The trigger behavior specified was not recognized as valid by the firmware.
Severity: Error
EventID: E8030390 (hex)

**Message: 913**
The trigger connection specified does not physically exist on this network analyzer
Severity: Error
EventID: E8030391 (hex)

**Message: 914**
Cannot set "Accept Trigger Before Armed", since this hardware configuration does not support edge triggering.
Severity: Error
EventID: E8030392 (hex)
Message: 915
Cannot set "Trigger Output Enabled", since this hardware configuration does not support BNC2.
Severity: Error
EventID: E8030393 (hex)

Message: 916
Exceeded maximum trigger delay.
Severity: Error
EventID: E8030394 (hex)

Message: 917
Exceeded minimum trigger delay.
Severity: Error
EventID: E8030395 (hex)

Measure Errors

Message: 1024
If you are going to display or otherwise use a memory trace, you must first store a data trace to memory.
Severity: Warning
EventID: A8040400 (hex)

Message: 1025
"The measurement failed to shut down properly. The application is in a corrupt state and should be shut down and
restarted."
Severity: Error
Further explanation: This message is displayed if the PNA application becomes corrupt. If you continue to get this
error, please call customer service
EventID: E8040401 (hex)

Message: 1026
The measurement failed to shut down properly. The update thread failed to exit properly.
Severity: Warning
EventID: A8040402 (hex)

Message: 1027
"Group Delay format with CW Time or Power sweeps produces invalid data."
Severity: Warning
Further explanation: Group Delay format is incompatible with single-frequency sweeps. Invalid data is produced.
Suggestions: Ignore the data or choose a different format or sweep type.
EventID: A8040403 (hex)

**Message: 1028**
Severity: Informational
"MSG_LIMIT_FAILED"
Further explanation: Limit line test failed.
EventID: 68040404 (hex)

**Message: 1029**
Severity: Informational
"MSG_LIMIT_PASSED"
Further explanation: Limit line test passed.
EventID: 68040405 (hex)

**Message: 1030**
"Exceeded the maximum number of measurements allowed."
Severity: Warning
Further explanation: See [Traces, Channels, and Windows on the PNA](#) for learn about maximum measurements.
EventID: A8040406 (hex)

**Message: 1031**
"Network Analyzer Internal Error. Unexpected error in AddNewMeasurement."
Severity: Warning
Further explanation: If you continue to get this message, contact product support.
EventID: A8040407 (hex)

**Message: 1032**
"No measurement was found to perform the selected operation. Operation not completed."
Severity: Warning
Further explanation: None
Suggestions: Create a measurement before performing this operation.
EventID: A8040408 (hex)

**Message: 1033**
The Markers All Off command failed.
Severity: Warning
EventID: A8040409 (hex)

**Message: 1034**
"A memory trace has not been saved for the selected trace. Save a memory trace before attempting trace math operations."

**Severity:** Warning

**Further explanation:** Must have a memory trace when trying to do Trace Math,

**EventID:** A804040A (hex)

**Message:** 1035

"MSG_SET_AVERAGE_COMPLETE"

**Severity:** Informational

**Further explanation:** Informational for COM programming. Averaging factor has been reached.

**EventID:** 6804040B (hex)

**Message:** 1036

"MSG_CLEAR_AVERAGE_COMPLETE"

**Further explanation:** Informational for COM programming. Averaging factor has NOT been reached.

**EventID:** 6804040C (hex)

**Message:** 1037

"Time Domain transform requires at least 3 input points. The transform has been deactivated."

**Severity:** Informational

**Further explanation:** None

**Suggestions:** Increase the number of points.

**EventID:** 6804040D (hex)

**Message:** 1038

Smoothing requires a scalar format, and has been deactivated.

**Severity:** Informational

**EventID:** 6804040E (hex)

**Message:** 1039

A receiver power calibration in this instrument state file cannot be recalled into this firmware version.

**Severity:** Warning

**EventID:** A804040F (hex)

**Message:** 1047

Could not achieve target power.

**Severity:** Error

**Further explanation:** This indicates that the PNA was unable to find a source power during the THRU step of the cal sufficiently high to boost the measured noise power on port 2 to 6 dB above the noise floor.
**Message: 1104**
"Exceeded limit on number of measurements."
**Severity:** Error
**Further explanation:** See Traces, Channels, and Windows on the PNA for measurement limits.
**EventID:** E8040450 (hex)

**Message: 1105**
"Parameter not valid."
**Severity:** Error
**Further explanation:** A measurement parameter that was entered programmatically is not valid.
**EventID:** E8040451 (hex)

**Message: 1106**
"Measurement not found."
**Severity:** Error
**Further explanation:** Any of these could be the cause:
- Trying to calibrate but already have maximum measurements.
- Trying to do a confidence check but there is not a measurement.
- Trying to create, activate, or alter a measurement through COM that has been deleted through the front panel.
- Trying to use a trace name through programming that is not unique.
**EventID:** E8040452 (hex)

**Message: 1107**
"No valid memory trace."
**Severity:** Error
**Further explanation:** Must have a memory trace when trying to do Trace Math,
**Suggestions:** Store a memory trace.
**EventID:** E8040453 (hex)

**Message: 1108**
"The reference marker was not found."
**Severity:** Error
**Further explanation:** Attempted to create a delta marker without first creating a reference marker (COM only).
**EventID:** E8040454 (hex)

**Message: 1109**
"Data and Memory traces are no longer compatible. Trace Math has been turned off."
**Severity:** Error
**Further explanation:** Warning - channel setting has changed while doing trace math.
Suggestions: Store another memory trace and turn trace math back on.

EventID: A8040455 (hex)

Message: 1110
"Data and Memory traces are not compatible. For valid trace math operations, memory and data traces must have similar measurement conditions."

Severity: Error

Further explanation: Tried to do trace math without compatible data and memory traces.

Suggestions: Store another memory trace.

EventID: E8040456 (hex)

Message: 1111
"Marker Bandwidth not found."

Severity: Error

Further explanation: Could not find a portion of trace that meets the specified bandwidth criteria.

EventID: E8040457 (hex)

Message: 1112
"The peak was not found."

Severity: Error

Further explanation: Could not find portion of trace that meets peak criteria.

Suggestions: See Marker Peak criteria.

EventID: E8040458 (hex)

Message: 1113
"The target search value was not found."

Severity: Error

Further explanation: Could not find interpolated data point that meets search value.

EventID: E8040459 (hex)

Message: 1114
"Reflection measurement, such as S11, must supply an auxiliary port to disambiguate 2-port measurements on multiport instruments."

Severity: Error

Further explanation:

EventID: E804045A (hex)

Message: 1115
"The receiver power calibration has been turned off because the type of measurement or source port has changed, so the calibration is no longer valid."
Severity: Warning
Further explanation:
EventID: A804045B (hex)

Message: 1116
"Receiver power cal requires the active measurement to be of unratioed power."

Severity: Warning
Further explanation:
EventID: A804045C (hex)

Message: 1117
"There is currently no source power calibration associated with the channel and source port of the active measurement. A source power cal should be performed or recalled before performing a receiver power calibration."

Severity: Warning
Further explanation:
EventID: A804045D (hex)

Message: 1118
"The attempted operation can only be performed on a standard measurement type."

Severity: Error
Further explanation:
EventID: E804045E (hex)

Message: 1119
"The custom measurement cannot be loaded because it is not compatible with the Network Analyzer hardware."

Severity: Error
Further explanation:
Suggestions:
EventID: E804045F (hex)

Message: 1120
"The custom measurement cannot be loaded because it is not compatible with the Network Analyzer software."

Severity: Error
Further explanation:
EventID: E8040460 (hex)

Message: 1121
"The custom measurement load operation failed for an unspecified reason."

Severity: Error
Further explanation:
EventID: E8040461 (hex)

**Message: 1122**
"The custom measurement data processing has generated an unhandled exception, and will be terminated. The PNA software may be in an unstable state and it is recommended that the PNA software be shutdown and restarted."
**Severity:** Error

Further explanation:
EventID: E8040462 (hex)

**Message: 1123**
"The attempted operation can only be performed on a custom measurement type."
**Severity:** Error

Further explanation:
EventID: E8040463 (hex)

**Message: 1124**
"The requested custom measurement is not available."
**Severity:** Error

Further explanation:
EventID: E8040464 (hex)

**Message: 1125**
"The requested custom algorithm was not found."
**Severity:** Error

Further explanation:
EventID: E8040465 (hex)

**Message: 1126**
"Normalization cannot be turned on because the measurement does not have a valid divisor buffer."
**Severity:** Error

Further explanation:
EventID: E8040466 (hex)

**Message: 1127**
"The Raw Data requested by the measurement could not be provided."
**Severity:** Warning

Further explanation:
EventID: A8040467 (hex)
**Message: 1128**
"The selected Sweep Type does not allow Transform and Gating. Transform and Gating disabled."
**Severity:** Error
**Further explanation:**
**EventID:** E8040468 (hex)

**Message: 1129**
Memory trace can not be applied to this measurement
**Severity:** Error
**EventID:** E8040469 (hex)

**Message: 1130**
Normalization can not be applied to this measurement
**Severity:** Error
**EventID:** E804046A (hex)

**Message: 1131**
The data provided has an invalid number of points. It could not be stored
**Severity:** Error
**EventID:** E804046B (hex)

**Message: 1132**
The measurement stored in the save/recall state has an invalid version. It could not be loaded
**Severity:** Error
**EventID:** E804046C (hex)

**Message: 1133**
This data format argument for this operation must be "naDataFormat_Polar"
**Severity:** Error
**EventID:** E804046D (hex)

**Message: 1134**
This data format argument for this operation must be a scalar data format
**Severity:** Error
**EventID:** E804046E (hex)

**Message: 1135**
The memory trace is not valid for the current measurement setup.
**Severity:** Error
**EventID:** E804046F (hex)
**Message: 1136**
This measurement is incompatible with existing measurements in this channel. Choose another channel.

*Severity: Error*
*EventID: E8040470 (hex)*

**Message: 1137**
Port extension correction is not available for offset frequency measurements. Port extension correction has been disabled.

*Severity: Error*
*EventID: E8040471 (hex)*

**Message: 1138**
Physical port number assignments for logical port mappings must be unique.

*Severity: Error*
*EventID: E8040472 (hex)*

---

**Parser Errors**

**Message: 1281**
"You have sent a read command to the analyzer without first requesting data with an appropriate output command. The analyzer has no data in the output queue to satisfy the request."

*Severity: Error*
*EventID: 68050501 (hex)*

**Message: 1282**
"You must remove the active controller from the bus or the controller must relinquish the bus before the analyzer can assume the system controller mode."

*Severity: Error*
*EventID: E8050502 (hex)*

**Message: 1283**
"The analyzer did not receive a complete data transmission. This is usually caused by an interruption of the bus transaction."

*Severity: Error*
*EventID: E8050503 (hex)*

**Message: 1284**
"The instrument status byte has changed."

*Severity: Informational*
*EventID: 68050504 (hex)*
Message: 1285
"The SCPI command received has caused error number %1: "%2"."
Severity: Informational
EventID: 68050505 (hex)

Message: 1286
"The INET LAN server has been started as process number %1."
Severity: Informational
EventID: 68050506 (hex)

Message: 1360
"Execution of the SCPI command has failed"
Severity: Error
EventID: E8050550 (hex)

Message: 1361
"The INET/LAN device is not accessible."
Severity: Error
EventID: E8050551 (hex)

Message: 1362
"The INET/LAN driver was not found."
Severity: Error
EventID: E8050552 (hex)

Message: 1363
"The INET/LAN driver was not found."
Severity: Error
EventID: E8050553 (hex)

Message: 1364
"The INET/LAN device is unable to acquire the necessary resources."
Severity: Error
EventID: E8050554 (hex)

Message: 1365
"The INET/LAN device generated a generic system error."
Severity: Error
EventID: E8050555 (hex)
**Message: 1366**
"Invalid address for the INET/LAN device."

**Severity:** Error

**EventID:** E8050556 (hex)

**Message: 1367**
"The INET I/O library was not found."

**Severity:** Error

**EventID:** E8050557 (hex)

**Message: 1368**
"An error occurred in the INET system."

**Severity:** Error

**EventID:** E8050558 (hex)

**Message: 1369**
"Access to the INET/LAN driver was denied."

**Severity:** Error

**EventID:** E8050559 (hex)

**Message: 1370**
"Could not load error system message dll."

**Severity:** Error

**EventID:** E805055A (hex)

**Message: 1371**
"ErrorSystemMessage.dll does not export the right function."

**Severity:** Error

**EventID:** E805055B (hex)

**Message: 1372**
"Custom scpi library was not able to be knitted"

**Severity:** Error

**EventID:** E805055C (hex)

**Message: 1373**
"Could not knit the scpi error messages from the ErrorSystemMessage lib"

**Severity:** Error

**EventID:** E805055D (hex)
Message: 1374
Command is obsolete with this software version.
Severity: Error
EventID: E808055E (hex)

Message: 1375
CALC measurement selection set to none. Use Calc:Par:Sel
Severity: Error
EventID: E808055F (hex)

Message: 1535
"Parser got command: %1."
Severity: Informational
EventID: 680505FF (hex)

Display Errors 1536 - 1621

Message: 1536
"Exceeded the maximum of 4 traces in each window. The trace for <x> will not be added to window <x>."
Severity: Warning
Further explanation: None
Suggestions: Create the trace in another window. See the PNA window limits.
EventID: A8060600 (hex)

Message: 1537
"Exceeded the maximum of 16 data windows. New window will not be created."
Severity: Warning
Further explanation: None
Suggestions: Create the trace in an existing window. See the PNA window limits.
EventID: A8060601 (hex)

Message: 1538
"No Data Windows are present. Unable to complete operation."
Severity: Warning
Further explanation: Your remote SCPI operation tried to create a new measurement while there were no windows present
Suggestions: Create a new window before creating the measurement. See example Create a measurement using SCPI
EventID: A8060602 (hex)
Message: 1539
"No data traces are present in the selected window. Operation not completed."

Severity: Warning
Further explanation: None
EventID: A8060603 (hex)

Message: 1540
"Cannot complete request to arrange existing measurements in <x> windows due to the limit of <x> traces per window."

Severity: Informational
Further explanation: The arrange window feature cannot put the existing traces into the number of windows you requested because only 4 traces per window are allowed. See Arranging Existing Measurements
Suggestions: Either create more windows or delete some traces.
EventID: 68060604 (hex)

Message: 1541
"Unable to establish a connection with the specified printer."

Severity: Warning
Further explanation: None
Suggestions: Refer to Printer Help
EventID: A8060605 (hex)

Message: 1542
"Printout canceled."

Severity: Informational
EventID: 68060606 (hex)

Message: 1616
"Window not found."

Severity: Error
Further explanation: A window was specified in your program which does not exist.
Suggestions: Query the name of your window before specifying.
EventID: E8060650 (hex)

Message: 1617
"Duplicate window ID specified."

Severity: Error
Further explanation: None
EventID: E8060651 (hex)
Message: 1618
"Exceeded limit on number of windows."
Severity: Error
Further explanation: There is a limit of 4 windows per screen.
EventID: E8060652 (hex)

Message: 1619
"Exceeded limit on number of traces/window."
Severity: Error
Further explanation: There is a limit of 4 traces per window. See the Traces, Channels, and Windows on the PNA.
Suggestions: Create the trace in another window
EventID: E8060653 (hex)

Message: 1620
"Trace not found."
Severity: Error
Further explanation: Your program tried to communicate with a non-existing trace.
Suggestions: Query the trace ID before writing to it.
EventID: E8060654 (hex)

Message: 1621
"The operating system does not recognize this printer."
Severity: Warning
EventID: A8060655 (hex)

Message: 1622
Duplicate trace ID specified.
Severity: Error
EventID: E8060656 (hex)

Channel Errors 1792 - 1878

Message: 1792
"Sweep Complete."
Severity: Informational
Further explanation: None
Suggestions: None
EventID: 68070700 (hex)
**Message: 1793**

"All triggerable acquisitions have completed."

**Severity:** Informational

**Further explanation:**

**EventID:** 68070701 (hex)

**Message: 1794**

"The last trigger produced an aborted sweep."

**Severity:** Informational

**Further explanation:**

**EventID:** 68070702 (hex)

**Message: 1795**

"The segment list must be adjusted to have at least one active segment with more than 0 points to use segment sweep."

**Severity:** Informational

**Further explanation:** You attempted to change **Sweep type** to Segment sweep, but there is either no segments defined or no sweep points in the defined segments

**Suggestions:** Define at least one segment with at least one measurement point. See Segment sweep for more information

**EventID:** 68070703 (hex)

**Message: 1796**

"MSG_SET_CHANNEL_DIRTY"

**Severity:** Informational

**Further explanation:** This informational message occurs when a channel setting has changed but the channel still has data that was taken with the previous setting. The following CLEAR message occurs when new channel data is taken.

**EventID:** 68070704 (hex)

**Message: 1797**

"MSG_CLEAR_CHANNEL_DIRTY"

**Severity:** Informational

**Further explanation:** The previous SET message occurs when a channel setting has changed but the channel still has data that was taken with the previous setting. This CLEAR message occurs when new channel data is taken.

**EventID:** 68070705 (hex)

**Message: 1798**

Sweep time has changed from Auto to Manual mode. If desired to return to Auto mode, enter sweep time value of 0.
Severity: Informational
EventID: 68070706 (hex)

Message: 1799
"Set Sweep Completed"

Severity: Informational

Further explanation: This event occurs when a sweep and its associated sweep calculations finish. This is typically when all sweeps on a channel complete.

EventID: 68070707 (hex)

Message: 1800
"Clear Sweep Completed"

Severity: Informational

Further explanation: This event occurs immediately after the SET SWEEP COMPLETED event. These two events set and clear the "Sweep Completed" bit (bit 4) on the SCPI Device Status register.

EventID: 68070708 (hex)

Message: 1801
"All Sweeps Completed and Processed"

Severity: Informational

Further explanation: This event occurs when all of the sweeps and sweep calculations are complete for a channel.

EventID: 68070709 (hex)

Message: 1802
Low Pass: Frequency limits have been changed.

Severity: Informational

EventID: 6807070A (hex)

Message: 1803
Low Pass: Number of points have been changed.

Severity: Informational

EventID: 6807070B (hex)

Message: 1804
Low Pass: Frequency limits and number of points have been changed.

Severity: Informational

EventID: 6807070C (hex)

Message: 1805
"Channel created"
Severity: Informational
EventID: 6807070D (hex)

Message: 1806
"Channel deleted"

Severity: Informational
EventID: 6807070E (hex)

Message: 1872
"Channel not found."

Severity: Error
Further explanation: A non-existent channel is being referenced under program control.
Suggestions: Query the channel number, then refer to it by number.
EventID: E8070750 (hex)

Message: 1873
"The requested sweep segment was not found."

Severity: Error
Further explanation: A non-existent sweep segment is being referenced under program control.
EventID: E8070751 (hex)

Message: 1874
"The sweep segment list is empty."

Severity: Error
Further explanation: Segment Sweep cannot be specified unless there is at least one defined segment. This error will only occur under remote control.
EventID: E8070752 (hex)

Message: 1875
"The number of points in active sweep segment list segments is 0."

Severity: Error
Further explanation: Segment Sweep cannot be specified unless there is at least data point specified in a segment. This error will only occur under remote control.
EventID: E8070753 (hex)

Message: 1876
"The specified source attenuator is not valid."

Severity: Error
Further explanation: You tried to set the Attenuator property on the Channel object on a PNA that doesn't have a source attenuator.
**Message: 1877**
"Log Frequency sweep cannot be selected with the current Number of Points. Please reduce Number of Points."
**Severity:** Error
**Further explanation:** The maximum number of points that can be used for Log sweep is 401.
**EventID:** E8070754 (hex)

**Message: 1878**
"The requested Number of Points is greater than can be selected for Log Frequency sweep."
**Severity:** Error
**Further explanation:** The maximum number of points that can be used for Log sweep is 401.
**EventID:** E8070755 (hex)

**Message: 1879**
"Response frequencies exceeded instrument range so Frequency Offset has been turned off."
**Severity:** Error
**Further explanation:** This error is returned whenever the instrument detects that the stimulus sweep setup and Frequency Offset settings result in computed response frequencies that exceed instrument limits. When this occurs, the instrument automatically turns off Frequency Offset to avoid the out-of-range conditions.
**Suggestions:** When this condition has occurred, change settings for either the stimulus frequencies or Frequency Offset so that the Response frequencies are within instrument bounds. Once this is done, Frequency Offset can once again be turned on.
**EventID:** E8070756 (hex)

**Message: 1880**
The total number of points for all the given segments exceeds the maximum number of points supported. The segments were not changed.
**Severity:** Error
**EventID:** E8070757 (hex)

**Message: 1881**
This instance of the Channels object was not used to place the channels in Hold, so no channels were resumed.
**Severity:** Error
**EventID:** E8070758 (hex)

**Message: 1882**
The port number was outside the range of allowed port numbers.
**Severity:** Error
**EventID:** E8070759 (hex)
Message: 1883
More ports than are present are required for this operation.
Severity: Error
EventID: E807075B (hex)

General Errors

Message: 2048
"The function you requested requires a capability provided by an option to the standard analyzer. That option is not currently installed."
Severity: Error
Further explanation: None
Suggestions: To view the options on your analyzer, click Help / About Network Analyzer. For more information see PNA Options
EventID: 68080800 (hex)

Message: 2049
"The feature you requested is not available on the current instrument."
Severity: Error
Further explanation: None
EventID: 68080801 (hex)

Message: 2050
"The feature you requested is incompatible with the current instrument state."
Severity: Error
Further explanation: None
Suggestions: None
EventID: 68080802 (hex)

Message: 2051
"File<x> has been saved."
Severity: Informational
Further explanation: None
EventID: 68080803 (hex)

Message: 2052
"Attempt to save <x> failed."
Severity: Error
Further explanation: None
Suggestions: If using a floppy disk, ensure it is inside the drive and the disk is not full. Check the filename for special characters.

EventID: E8080804 (hex)

**Message: 2053**
"Attempt to recall file failed because <x> was not found."

Severity: Error
Further explanation: None
EventID: E8080805 (hex)

**Message: 2054**
"<x> has a bad header."

Severity: Error
Further explanation: None
Suggestions: Recopy the file and / or delete the file.
EventID: E8080806 (hex)

**Message: 2056**
"Request to enter hibernate state."

Further explanation: None
EventID: 68080808 (hex)

**Message: 2057**
"Power up from automatic hibernate state. Program received PBT_APMRESUMEAUTOMATIC Message."

Further explanation: None
EventID: 68080809 (hex)

**Message: 2058**
"Power up from suspend hibernate state. Program received PBT_APMRESUMESUSPEND Message."

Further explanation: None
EventID: 6808080A (hex)

**Message: 2059**
"Power up from suspend hibernate state. Program received PBT_APMRESUMECRITICAL Message."

Severity: Warning
Further explanation: None
EventID: A808080B (hex)

**Message: 2060**
"Power up from unknown hibernate state UI recovery called. Program received no PBT_Message within the time
Message: 2061
"<x> already exists. File is being overwritten."
Further explanation: Used only for remote applications
EventID: 6808080D (hex)

Message: 2062
"File has not been saved."
Severity: Error
Further explanation: Used only for remote applications
EventID: E808080E (hex)

Message: 2063
"File <x> has been recalled."
Further explanation: Used only for remote applications
EventID: 6808080F (hex)

Message: 2064
"State version in <x> is considered obsolete by this version of this code."
Severity: Error
Further explanation: You attempted to recall a file that is no longer valid.
Suggestions: You must recreate the file manually.
EventID: E8080810 (hex)

Message: 2065
"State version in <x> is newer than the latest version supported by this code."
Severity: Error
Further explanation: You attempted to recall a file that was created by a later version of the PNA application.
Suggestions: You must recreate the file manually.
EventID: E8080811 (hex)

Message: 2066
"Error occurred while reading file <x>"
Severity: Error
Further explanation: The file may be corrupt.
Suggestions: Try to recreate the file.
EventID: E8080812 (hex)

Message: 2067
"Windows shell error: <x>"
Severity: Error
Further explanation: None
EventID: E8080813 (hex)

Message: 2068
Send message timed out returning: <x>.
Severity: Error
Further explanation: None
EventID: E8080814 (hex)

Message: 2069
"Changing GPIB mode to System Controller."
Severity: Informational
Further explanation: None
EventID: 68080815 (hex)

Message: 2070
"Changing GPIB mode to Talker Listener."
Severity: Informational
Further explanation: None
EventID: 68080816 (hex)

Message: 2071
"The Network Analyzer can not be put in GPIB System Controller mode until the GPIB status is Local. Stop any remote GPIB programs which may be using the Network analyzer, press the Macro/Local key and try again."
Severity: Informational
Further explanation: See LCL and RMT Operation
Suggestions: Press the Macro/Local key and try again.
EventID: 68080817 (hex)

Message: 2120
"This method can not be invoked through a late-bound COM call."
Severity: Error
Further explanation: None
Suggestions: Use the alternate method described in the COM programming documentation
EventID: E8080878 (hex)

**Message:** 2128
"The specified format is invalid."
**Severity:** Error
**Further explanation:** None

EventID: E8080850 (hex)

**Message:** 2129
"WINNT exception caught by Automation layer."
**Severity:** Error
**Further explanation:** None

EventID: E8080851 (hex)

**Message:** 2130
"Bad port specification."
**Severity:** Error
**Further explanation:** None

EventID: E8080852 (hex)

**Message:** 2131
"Failed to find a printer."
**Severity:** Error
**Further explanation:** None

**Suggestions:** See [Connecting to a Printer]

EventID: E8080853 (hex)

**Message:** 2132
"Manual trigger ignored."
**Severity:** Error
**Further explanation:** None

EventID: E8080854 (hex)

**Message:** 2133
"Attempt to set trigger failed."
**Severity:** Error
**Further explanation:** None

EventID: E8080855 (hex)

**Message:** 2134
"Macro execution failed."

Severity: Error
Further explanation: None
EventID: E8080856 (hex)

Message: 2135
"Specified macro definition is incomplete."
Severity: Error
Further explanation: None
EventID: E8080857 (hex)

Message: 2137
"Block data length error."
Severity: Error
Further explanation: See Getting Data from the Analyzer
EventID: E8080859 (hex)

Message: 2139
"Requested data not found."
Severity: Error
Further explanation: None
EventID: E808085B (hex)

Message: 2142
"The parameter supplied was out of range, so was limited to a value in range before being applied to the instrument."
Severity: Success
Further explanation: None
Suggestions: View range limits before sending programming commands.
EventID: 2808085E (hex)

Message: 2143
The parameter supplied was out of range, so was limited to a value in range before being applied to the instrument.
Severity: Error
EventID: E808085F (hex)

Message: 2144
"Request failed. The required license was not found."
Severity: Error
Further explanation: None
**EventID:** E8080860 (hex)

**Message:** 2145
"A remote call to the front panel has returned hresult <x>"

**Severity:** Error

**Further explanation:** This may indicate a problem with the front panel

**Suggestions:** Contact Technical support

**EventID:** E8080861 (hex)

**Message:** 2146
The recall operation failed.

**Severity:** Error

**Further explanation:**

**EventID:** E8080862 (hex)

**Message:** 2147
Attempt to save file failed.

**Severity:** Error

**Further explanation:**

**EventID:** E8080863 (hex)

**Message:** 2148
Recall attempt failed because file was not found.

**Severity:** Error

**Further explanation:**

**EventID:** E8080864 (hex)

**Message:** 2149
Recall file has a bad header.

**Severity:** Error

**Further explanation:**

**EventID:** E8080865 (hex)

**Message:** 2150
Recall file version is obsolete and no longer compatible with this instrument.

**Severity:** Error

**Further explanation:**

**EventID:** E8080866 (hex)

**Message:** 2151
The recall file contains an istate version newer than this instrument. A remote call to the front panel has returned hresult %1

**Severity**: Error

**Further explanation**:

**EventID**: E8080867 (hex)

**Message 2152**

"Front Panel <x>

**Severity**: Error

**Further explanation**: None

**EventID**: E8080868 (hex)

**Message 2153**

"Front Panel message"

**Severity**: Informational

**Further explanation**: None

**EventID**: 68080869 (hex)

**Message 2154**

"Power Service <x>

**Severity**: Error

**Further explanation**: There is more than 1 instance of powerservice running. There should only be one running. This might happen after running install shield - especially when upgrading the CPU board.

**Suggestions**: Try rebooting. If this persists, please call Customer Support.

**EventID**: E808086A (hex)

**Message 2155**

"Power Service <x>

**Severity**: Informational

**Further explanation**: None

**EventID**: 6808086B (hex)

**Message 2156**

"The Agilent Technologies GPIB driver can not be loaded or unloaded."

**Severity**: Error

**Further explanation**: None

**Suggestions**: If the problem persists, from the PNA desktop, right-click on My Computer. Click Properties, Click Hardware Tab. Click Device Manager Button. Expand GPIB Devices. Right-click and click Uninstall all GPIB interfaces devices. Reboot the PNA.

**EventID**: E808086C (hex)
**Message 2157**
"The National Instruments GPIB driver can not be loaded or unloaded."

**Severity:** Error

**Further explanation:** None

**Suggestions:** If the problem persists, from the PNA desktop, right-click on My Computer. Click Properties, Click Hardware Tab. Click Device Manager Button. Expand GPIB Devices. Right-click and click Uninstall all GPIB interfaces devices. Reboot the PNA.

**EventID:** E808086D (hex)

**Message 2158**
"The Agilent GPIB driver is loaded but it can not start its parser."

**Severity:** Error

**Further explanation:** None

**EventID:** E808086E (hex)

**Message: 2159**
The front panel is in remote mode.

**Severity:** Warning

**EventID:** A808086F (hex)

**Message: 2160**
The Registry Key specified could not be found.

**Severity:** Error

**EventID:** E8080870 (hex)

**Message: 2161**
An overcurrent condition has been detected on a probe plugged into the front panel.

**Severity:** Warning

**EventID:** A8080871 (hex)

**Message: 2162**
The operation timed out.

**Severity:** Error

**EventID:** E8080872 (hex)

**Message 2163**
"The Network Analyzer executed a preset."

**Severity:** Informational

**Further explanation:** None

**EventID:** 68080873 (hex)
Message 2164
"Access to file denied."

Severity: Error

Further explanation: This means that the system can not open an output file for writing. Most likely because the file is write protected.

Suggestions: Pick another file name or file directory, check floppy disk hard disk write access.

EventID: E8080874 (hex)

Message 2165
"File type is structured storage."

Severity: Informational

Further explanation: None

EventID: 68080875 (hex)

Message 2166
"The trigger operation failed."

Severity: Error

Further explanation: None

EventID: E8080876 (hex)

Message 2167
"Argument out of range error."

Severity: Error

Further explanation: None

Suggestions: None

EventID: E8080877 (hex)

Message: 2169
The given COM object is not a custom application

Severity: Error

EventID: E8080879 (hex)

Message: 2170
The eventID supplied was not recognized as a valid PNA eventID

Severity: Error

EventID: E808087A (hex)

Message: 2171
The operation was canceled.
Severity: Error
EventID: E808087B (hex)

Message: 2172
High security level cannot be disabled directly. Only an instrument preset or recall of lower security instrument state will reset this security level.

Severity: Error
EventID: E808087C (hex)

Message: 2173
Local lockout mode is on. The PNA application will not accept input from front panel, keyboard or mouse until this mode is turned off from a remote interface.

Severity: Error
EventID: E808087D (hex)

Message: 2174
The SnP request is not valid for the selected measurement.

Severity: Error
EventID: E808087E (hex)

Message: 2175
Preset is not supported while this dialog or wizard is open. Close the dialog or wizard and then try again.

Severity: Error
EventID: E808087F (hex)

Message: 2176
The function you requested requires a capability provided by an option to the standard analyzer. That option is not currently installed.

Severity: Error
EventID: E8080880 (hex)

Message: 2177
Catastrophic error. Crash dump recorded at <n>

Severity: Error
EventID: E8080881 (hex)

Message: 2178
In the context of a noise calibration, this would occur if the PNA was unable to set the state of the tuner Ecal module.

Severity: Error
EventID: E8080882 (hex)
**Message: 2179**
Failed to open gen.lic.

**Severity:** Error

**EventID:** E8080883 (hex)

Last modified:

Nov. 29, 2006
Updated bookmarks
About Error Messages

PNA errors and Operating System errors are displayed and logged in an error file. You can choose how to display PNA errors, or choose to not display PNA errors at all.

- Error Display
- View Error Log
- List of PNA Errors
- SCPI Errors

Other System topics

Error Display

By default, error messages appear on the PNA screen for a brief period. You can choose to have them stay on the screen until you click an OK button, or have them not appear at all. When they stay on the screen, a Help button is available to provide further assistance.

How to select the display of Error Messages

Using front-panel HARDKEY [softkey] buttons

1. Press **SYSTEM**
2. then **[Help]**
3. then **[Error Messages]**
4. then **[Error Display]**

Using a mouse with PNA Menus

1. Click **Help**
2. then **Error Messages**
3. then **Error Display**
Error Display dialog box help

On Preset, these settings revert to their defaults (enabled, timed popups).

Enable Messages Check to display all PNA error messages as they occur. Clear to suppress the display of PNA error messages. You can still view them in the error log.

Calibration Error Message Windows

Timed Popups Displays error messages on the screen for a duration of time proportional to the length of the message. You can then view the message in the error log and get further assistance.

Confirmation Dialog boxes Displays error messages in a standard dialog box. You then choose OK or Cancel to close the dialog box, or press Help to get further information on the error message.

View Error Log

The PNA Error Log is a list of all events that have occurred. (Events are used in programming the PNA using COM.) PNA errors is a subset of PNA events. Only events with severity codes of ERROR are displayed on the PNA screen as they occur. From the error log, you can access further help with an error by selecting the error and clicking Help.

How to view the Error Log

Using front-panel HARDKEY [softkey] buttons

1. Press SYSTEM
2. then [Help]
3. then [Error Messages]
4. then [View Error Log]

Using a mouse with PNA Menus

1. Click Help
2. then Error Messages
3. then View Error Log

No programming commands
**Error Log** dialog box help

**Network analyzer errors only** Select to view only PNA errors. Clear to view all errors that occur on all applications of the computer.

**Description** Error message that appears on the PNA screen.

- **A** - Event ID Error message number
- **B** - Date the Error occurred
- **C** - Time the Error occurred
- **D** - Severity Code - All events have one of the following severity codes:
  - SUccess - the operation completed successfully
  - INFormation - events that occur without impact on the measurement integrity
  - WARening - events that occur with potential impact on measurement integrity
  - ERRor - events that occur with serious impact on measurement integrity
- **E** - Application in which the error occurred.

**OK** Closes the Dialog box

**Help** Provides further information on the selected Error message

To clear the Error Log:

1. From the **File** menu click **Minimize Application**
2. On the desktop, select **Start, Settings, Control Panel**
3. On the Control Panel, click **Administrative Tools**
4. On the Administrative Tools window, click **Event Viewer**

5. On the Event Viewer window, right-click **Application**

6. Select **Clear all Events**

7. If you want to save a file with the contents of the Event Log, click **Yes**. Otherwise, click **No**

To restore the PNA application, click on the PNA Analyzer taskbar button at the bottom of the screen.
Analyzer Accessories

- Coax Mechanical Calibration Kits
- Waveguide Mechanical Calibration Kits
- Electronic Calibration (ECal)
- Mechanical Verification Kits
- Adapter and Accessory Kits
- Test Port Cables
- USB Peripherals
- Connector Care and Cleaning Supplies

Other Support topics

For product and order information:

- Visit [www.agilent.com/find/accessories](http://www.agilent.com/find/accessories)
- Use the search function to locate information about a particular accessory or view the entire RF and Microwave Test Accessories Catalog.

Accessories are available in these connector types:

- 50 ohm Type-N
- 75 ohm Type-N
- 3.5 mm
- 7 mm (APC-7)
- 7-16
- 2.92 mm
- 2.4 mm
- 1.85 mm
- 1 mm

Test port cables and a calibration kit are necessary for a complete measurement system.
A verification kit is used to verify corrected system performance.

See the connector type for each PNA model

### Coax Mechanical Calibration Kits

<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Upper Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>85032B</td>
<td>Type-N (50 Ohm)</td>
<td>6 GHz</td>
</tr>
<tr>
<td>85032F</td>
<td>Type-N (50 Ohm)</td>
<td>9 GHz</td>
</tr>
<tr>
<td>85054B</td>
<td>Type-N (50 Ohm)</td>
<td>18 GHz</td>
</tr>
<tr>
<td>85036E</td>
<td>Type-N (75 Ohm)</td>
<td>3 GHz</td>
</tr>
<tr>
<td>85050B</td>
<td>7 mm</td>
<td>18 GHz</td>
</tr>
<tr>
<td>85033D</td>
<td>3.5 mm</td>
<td>6 GHz</td>
</tr>
<tr>
<td>85038A</td>
<td>7-16</td>
<td>7.5 GHz</td>
</tr>
<tr>
<td>85033E</td>
<td>3.5 mm</td>
<td>9 GHz</td>
</tr>
<tr>
<td>85052B</td>
<td>3.5 mm</td>
<td>26.5 GHz</td>
</tr>
<tr>
<td>85052C</td>
<td>3.5 mm TRL</td>
<td>26.5 GHz</td>
</tr>
<tr>
<td>85056K</td>
<td>2.92 mm</td>
<td>50 GHz</td>
</tr>
<tr>
<td>85056A</td>
<td>2.4 mm</td>
<td>50 GHz</td>
</tr>
<tr>
<td>85058B/E</td>
<td>1.85 mm (data-based)</td>
<td>67 GHz</td>
</tr>
<tr>
<td>85059A</td>
<td>1.00 mm (data-based)</td>
<td>110 GHz</td>
</tr>
</tbody>
</table>

### Waveguide Mechanical Calibration Kits
<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>X11644A</td>
<td>WR-90</td>
<td>8.2-12.4 GHz</td>
</tr>
<tr>
<td>P11644A</td>
<td>WR-62</td>
<td>12.4-18 GHz</td>
</tr>
<tr>
<td>K11644A</td>
<td>WR-42</td>
<td>18-26.5 GHz</td>
</tr>
<tr>
<td>R11644A</td>
<td>WR-28</td>
<td>26.5-40 GHz</td>
</tr>
<tr>
<td>Q11644A</td>
<td>WR-22</td>
<td>33-50 GHz</td>
</tr>
<tr>
<td>U11644A</td>
<td>WR-19</td>
<td>40-60 GHz</td>
</tr>
<tr>
<td>V11644A</td>
<td>WR-15</td>
<td>50-75 GHz</td>
</tr>
</tbody>
</table>

**Electronic Calibration (ECal)**

<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>RF Two-Port</td>
<td></td>
<td></td>
</tr>
<tr>
<td>85091C</td>
<td>7 mm (APC-7)</td>
<td>300 kHz-9 GHz</td>
</tr>
<tr>
<td>85092C</td>
<td>Type-N (50 ohm)</td>
<td>300 kHz-9 GHz</td>
</tr>
<tr>
<td></td>
<td>Port B available with 3.5 mm or 7-16a</td>
<td></td>
</tr>
<tr>
<td>85093C</td>
<td>3.5 mm</td>
<td>300 kHz-9 GHz</td>
</tr>
<tr>
<td></td>
<td>Port B available with Type-N (50 ohm) or 7-16a</td>
<td></td>
</tr>
<tr>
<td>85096C</td>
<td>Type-N (75 ohm)</td>
<td>300 kHz-3 GHz</td>
</tr>
<tr>
<td>85098C</td>
<td>7-16a</td>
<td>300 kHz-7.5 GHz</td>
</tr>
<tr>
<td></td>
<td>Port B available with Type-N (50 ohm) or 3.5 mm</td>
<td></td>
</tr>
<tr>
<td>85099C</td>
<td>Type-F</td>
<td>300 kHz-3 GHz</td>
</tr>
<tr>
<td>RF Four-Port</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N4431B</td>
<td>3.5mm (f) (four-port), Type-N (f) (four-port), Mixed connector types</td>
<td>9 kHzb-13.5 GHz</td>
</tr>
</tbody>
</table>

---

*a, b: For use at 9 kHz only. +For use at 13.5 GHz only.
<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4432A</td>
<td>Type-N (f) (four-port)</td>
<td>300 kHz-18 GHz</td>
</tr>
<tr>
<td>Option 020</td>
<td></td>
<td>(available Feb. 2006)</td>
</tr>
<tr>
<td>N4432A</td>
<td>APC 7 (four-port)</td>
<td>300 kHz-18 GHz</td>
</tr>
<tr>
<td>Option 030</td>
<td></td>
<td>(available Feb. 2006)</td>
</tr>
<tr>
<td>N4433A</td>
<td>3.5mm (f) (four-port)</td>
<td>300 kHz-20 GHz</td>
</tr>
<tr>
<td>Option 010</td>
<td></td>
<td>(available Feb. 2006)</td>
</tr>
</tbody>
</table>

**Microwave Two-Port**

<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4690B</td>
<td>Type-N (50 ohm)</td>
<td>300 kHz-18 GHz</td>
</tr>
<tr>
<td>N4691B</td>
<td>3.5 mm</td>
<td>300 kHz-26.5 GHz</td>
</tr>
<tr>
<td>N4692A</td>
<td>2.92 mm</td>
<td>10 MHz-40 GHz</td>
</tr>
<tr>
<td>N4693A</td>
<td>2.4 mm</td>
<td>10 MHz-50 GHz</td>
</tr>
<tr>
<td>N4694A</td>
<td>1.85 mm</td>
<td>10 MHz-67 GHz</td>
</tr>
<tr>
<td>N4696BA</td>
<td>7 mm</td>
<td>300 kHz-18 GHz</td>
</tr>
</tbody>
</table>

- Limits ECAL module high frequency to 7.5 GHz.
- Performance from 9 kHz to 300 kHz is valid only for the E5071C ENA Network analyzer with firmware version A.09.10 or higher.

**Verification Kits**

<table>
<thead>
<tr>
<th>Model</th>
<th>Connector Type</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>85055A</td>
<td>Type-N (50 Ohm)</td>
<td>300 kHz-9 GHz</td>
</tr>
<tr>
<td>85053B</td>
<td>3.5 mm</td>
<td>300 kHz-26.5 GHz</td>
</tr>
<tr>
<td>85057B</td>
<td>2.4 mm</td>
<td>.045-50 GHz</td>
</tr>
<tr>
<td>R11645A</td>
<td>WR-28</td>
<td>26.5-40 GHz</td>
</tr>
<tr>
<td>Q11645A</td>
<td>WR-22</td>
<td>33-50 GHz</td>
</tr>
</tbody>
</table>

**Adapters and Accessory Kits**
<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11878A</td>
<td>Type-N to 3.5 mm Adapter Kit</td>
</tr>
<tr>
<td>11525A</td>
<td>Type-N (m) to 7 mm (APC-7)</td>
</tr>
<tr>
<td>11853A</td>
<td>Type-N Accessory Kit</td>
</tr>
<tr>
<td>11900B</td>
<td>2.4 mm (f) to 2.4 mm (f)</td>
</tr>
<tr>
<td>11900C</td>
<td>2.4 mm (f) to 2.4 mm (m)</td>
</tr>
<tr>
<td>85130G</td>
<td>Test Port Adapter Set, 2.4 mm (f) to 2.4 mm (m,f)</td>
</tr>
<tr>
<td>11901B</td>
<td>2.4 mm (f) to 3.5 mm (f)</td>
</tr>
<tr>
<td>11901D</td>
<td>2.4 mm (f) to 3.5 mm (m)</td>
</tr>
<tr>
<td>85130F</td>
<td>Test Port Adapter Set, 2.4 mm (f) to 3.5 mm (m,f)</td>
</tr>
<tr>
<td>11902B</td>
<td>2.4 mm (f) to 7 mm (APC-7)</td>
</tr>
<tr>
<td>11920A</td>
<td>1 mm (m) to 1 mm (m)</td>
</tr>
<tr>
<td>11920B</td>
<td>1 mm (f) to 1 mm (f)</td>
</tr>
<tr>
<td>11920C</td>
<td>1 mm (m) to 1 mm (f)</td>
</tr>
<tr>
<td>11921A</td>
<td>1 mm (m) to 1.85 mm (m)</td>
</tr>
<tr>
<td>11921B</td>
<td>1 mm (f) to 1.85 mm (f)</td>
</tr>
<tr>
<td>11921C</td>
<td>1 mm (m) to 1.85 mm (f)</td>
</tr>
<tr>
<td>11921D</td>
<td>1 mm (f) to 1.85 mm (m)</td>
</tr>
<tr>
<td>11922A</td>
<td>1 mm (m) to 2.4 mm (m)</td>
</tr>
<tr>
<td>11922B</td>
<td>1 mm (f) to 2.4 mm (f)</td>
</tr>
<tr>
<td>11922C</td>
<td>1 mm (m) to 2.4 mm (f)</td>
</tr>
<tr>
<td>11922D</td>
<td>1 mm (f) to 2.4 mm (m)</td>
</tr>
</tbody>
</table>
### Test Port Cables

<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4697E</td>
<td>1.85 mm (f) to 1.85 mm (rugged f) flexible (single)</td>
</tr>
<tr>
<td>N4697F</td>
<td>1.85 mm (rugged f, f) to 1.85 mm (rugged m, rugged f) flexible (set)</td>
</tr>
<tr>
<td>N6315A</td>
<td>Type-N (m) to Type-N (f), 16 in. (single)</td>
</tr>
<tr>
<td>N6314A</td>
<td>Type-N (m) to Type-N (m), 24 in. (single)</td>
</tr>
<tr>
<td>85133D</td>
<td>2.4 mm (f) to 2.4 mm (m,f) semi-rigid (set)</td>
</tr>
<tr>
<td>85133F</td>
<td>2.4 mm (f) to 2.4 mm (m,f) flexible (set)</td>
</tr>
<tr>
<td>85134D</td>
<td>2.4 mm (f) to 3.5 mm (m,f) semi-rigid (set)</td>
</tr>
<tr>
<td>85134F</td>
<td>2.4 mm (f) to 3.5 mm (m,f) flexible (set)</td>
</tr>
</tbody>
</table>

### USB Peripherals

<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4688A</td>
<td>CD RW drive - with USB cable.</td>
</tr>
<tr>
<td>N4689A</td>
<td>USB 4-port hub - for connecting additional USB peripherals.</td>
</tr>
<tr>
<td>82357A</td>
<td>USB/GPIB Interface - for controlling GPIB devices through USB. Learn more about using the 82357A with the PNA</td>
</tr>
</tbody>
</table>

### Connector Care and Cleaning Supplies

See Connector Care

<table>
<thead>
<tr>
<th>Part Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1401-0225</td>
<td>Standard End-Cap, Type-N (m)</td>
</tr>
<tr>
<td>1401-0248</td>
<td>ESD Safe End-Cap, Type-N (m)</td>
</tr>
<tr>
<td>1401-0225</td>
<td>Standard End-Cap, Type-N (f)</td>
</tr>
<tr>
<td>1401-0247</td>
<td>ESD Safe End-Cap, Type-N (f)</td>
</tr>
</tbody>
</table>
## ESD Supplies

*See ESD topic.*

<table>
<thead>
<tr>
<th>Part Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>9300-1367</td>
<td>Adjustable antistatic wrist strap</td>
</tr>
<tr>
<td>9300-0980</td>
<td>Antistatic wrist strap grounding cord (5 foot)</td>
</tr>
<tr>
<td>9300-0797</td>
<td>Static control table mat (2 foot x 4 foot) with earth ground wire</td>
</tr>
<tr>
<td>9300-1126</td>
<td>ESD heel strap</td>
</tr>
</tbody>
</table>

---

**Last Modified:**

- 9-Jan-2009  Added footnote for N4431B
- 1-Dec-2008  Added link to connector care
82357A USB / GPIB Interface

The Agilent 82357A is an adapter that creates a GPIB Interface from one of your unused PNA USB ports.

- **Applications**
- **Installing**
- **Configuring**
- **Connecting**
- **Communicating with other Equipment**

### Applications

The 82357A can be used for the following PNA applications:

- **Frequency Converter Application** - The 82357A is included with the Frequency Converter Application (option 083). External sources MUST be connected to this Interface if controlling the PNA using an external PC. See connecting diagram below. To learn more, see Configure an external LO source.

**Note**: If the PNA is hibernated during an FCA measurement involving an external source under FCA control, and then the PNA is restarted, a VISA error message will appear stating “VI_ERROR_INV_OBJECT.” To correct this problem, the 82357 USB/GPIB interface must be reinitialized after hibernation. This is done by clicking on the Accept button in the interface initialization dialog box. The green READY light on the interface will illuminate.

- **PNA Controller** - The 82357A can be used by the PNA to control other GPIB devices. This frees the default GPIB interface to perform other GPIB operations, such as control the PNA from an external PC.

- **Source Power Cal** - The 82357A can be used to run a source power calibration.

### Installing the 82357A USB/GPIB Interface

1. [Download and install firmware](#) PNA revision 3.0 or greater. To check the revision of your PNA firmware, click Help then About Network Analyzer.

2. Upgrade to the latest Agilent IO libraries from the CDROM that was shipped with the 82357A. If not available, download them from [www.Agilent.com](http://www.Agilent.com) (search for 82357A)

### Configure the 82357A USB/GPIB Interface

When the 82357A is connected to the PNA USB, the following dialog box appears:
Normally, you do NOT need to edit these settings. The 82357A USB/GPIB Interface is configured automatically as the next unused VISA interface. This is usually **GPIB2** unless you have already configured it for another purpose.

If the VISA Interface Name appears as GPIB0 or GPIB1, these Interfaces must be returned to their default settings for the 82357A to work properly with the PNA. See Configure for VISA / SICL to learn how.

**Connecting the 82357A USB/GPIB Interface**

The following diagram illustrates how to connect GPIB test equipment using the USB/GPIB Interface.

- Plug the USB/GPIB Interface into any unused PNA USB port.
- The default GPIB Interface and USB/GPIB Interface should never be connected together.

![Diagram](image)

**Communicating with Equipment Connected to the USB/GPIB Interface**

- The Frequency Converter Application will automatically find and communicate with test equipment that is connected to the USB/GPIB Interface.
- Source power calibration will automatically find and communicate with the power meter that is connected to the USB/GPIB Interface.
To control other devices through your own program using the 82357A, you must include the new GPIB Interface number when addressing the devices.
Firmware Upgrade

PNA firmware upgrades are available to you at no cost in a self-extracting Install Shield file. The upgrade includes the PNA application, Online help, and Service Utilities. Note: The file is at least 50 MB.

Note: The CPU speed, amount of RAM, and Operating System in your PNA may limit your ability to upgrade firmware. See http://na.tm.agilent.com/pna/firmware/PNA_support_matrix.doc.

The following options are available for you to upgrade your PNA application:

- **Auto-Check** and **AgileUpdate** If your PNA is connected to the Internet, these utilities will automatically check for, download, and install, the new firmware and associated files when the PNA application is started. You will be prompted before this occurs.

- **Website Access** If your PNA is NOT connected to the Internet, but you have a PC that is, you can download the PNA firmware and associated files to a storage medium.

To manually check the version of firmware on the PNA, click Help, then About Network Analyzer.

**Note: After a firmware upgrade...**

- Custom Cal Kits must be imported. Learn more

- If a different desktop icon named "Network Analyzer" exists, the shortcut to the PNA application will assume the same icon. Right-click on the desktop, then click Refresh.

**Other Support Topics**

**Auto-Check**

With Internet access to your PNA, Auto-Check automatically and regularly checks the Internet for new PNA firmware revisions. If a new revision is found, a notification message prompts you to run the AgileUpdate utility, which then performs the actual download.

Without Internet access to your PNA, Auto-Check provides a reminder prompt at the selected intervals.

Auto-Check is run only when the PNA application is started. Once the PNA application is running, it will not check for updates again until it is restarted.

When Auto-Check runs, it checks the following conditions:

- Is there an active connection to the Internet?
- Is the Auto-Check utility enabled?
- Is it time to check for new firmware?
- Does new firmware exist?

If all of these conditions are true, Auto-Check shows the following dialog box.
If all of these conditions are NOT true, or to change these settings at any time, click **System, Service**, then **AgileUpdate**. From within AgileUpdate, click **AutoCheck**. These preferences are stored in the PNA registry. Future firmware upgrades will not change these settings.

**PNA Auto-Check** dialog box help

- **Enable** When the PNA application is started, Auto-Check will search the PNA website for firmware updates at the selected time interval.
- **Disable** When the PNA application is started, Auto-Check will NOT search the PNA website for firmware updates.
- **Time Interval** Select the time interval Auto-Check is to search for firmware updates.
- **Accept** Starts update process.
- **Ignore** No further action is taken until the selected time interval has elapsed.
- **Remind Me Later**: This window is displayed again after 1-20 days depending upon the time interval selected.

**AgileUpdate**

**Note:** You must have administrative privileges on the analyzer to run this utility. See [Set Up Analyzer Users](#).
How to start AgileUpdate

Connect the PNA to the Internet. A LAN connection is recommended because a firmware download can take many hours using a modem.

Using front-panel HARDKEY [softkey] buttons

1. Press **SYSTEM**
2. then **[Service]**
3. then **[AgileUpdate]**

Using a mouse with PNA Menus

1. Click **Utility**
2. then **System**
3. then **Service**
4. then **AgileUpdate**

1. Click **Check for Updates**.
2. If updates exist, click **Download & Install**.

No programming commands are available for this feature.
**AgileUpdate dialog box help**

**Note:** Your privacy is important to Agilent. AgileUpdate does NOT send ANY information from the PNA to the server. It only downloads from the server to the PNA.

**Restart**  
Click to restart from the beginning.

**Configure**  
Click to launch the [Configure dialog box](#).

**Clean-up**  
Click to delete all but the two most recent install shield packages from the PNA hard drive.

**Firmware History**  
Available after clicking [Check for Updates](#).

**Auto-Check**  
Launches the [Auto-Check](#) dialog box.

**Item / Application**  
Lists the items available for download at the firmware website.

- Click items with **i** to read more information about the download.
- Items in **RED** should be downloaded and installed individually.
- Multi-language help includes all help files except English.

**Note:** The firmware includes the help file. Therefore, only the firmware checkbox will be selected if a new version for both the firmware and the help file are available.

**Select Source**

**Default Website**  
The Agilent site that contains upgrade FW.

**Other Specified URL**  
Click if you were instructed to get firmware from a different website.

**Check Customer FW Releases**  
Check this box to also check Customer Releases in addition to Production Releases. This setting provides you with the very latest PNA firmware. Customer Releases are fully supported but have not yet been tested in all production models. Customer Releases take precedence over Production Releases. This setting is remembered and applied the next time AgileUpdate is run.

**Special Access Code...**  
Type in the code if you were given one from Agilent Technical Support. Otherwise, leave blank.

**Make Latest Firmware Available...**  
Select this checkbox if you want to download the latest firmware, even if it is not new.

**Check for Updates**  
Click to look for firmware updates at the Agilent website. If there are newer versions, the files will be listed.

**Download and Install**  
When updates are found, this selection becomes available. Some files may be pre-checked. Be sure the corresponding boxes are checked for the files you want to download. Then click to download and install the update.

**Download Only**  
Click to download the files to the analyzer hard disk and install the files at a later time. At that time, click [Install from File](#).
Configuration dialog box help

**Note:** If AgileUpdate will not connect, try to access ANY Internet website. Contact your local IT department if necessary.

**Proxy Setting**
- **No Proxy or Default Proxy** Click if you use a LAN connection. AgileUpdate will automatically use the proxy specified in Internet Explorer.
- **Use specified Proxy / Port** Click to enter the proxy name and port. The format is: proxyName:portNumber. (The proxy port number is typically 8088).

**Internet timeout** If you are using an automatic dial-up Internet connection you may need to increase the timeout.

**Current Connection Status** Shows the current status of the PNA connection to the Internet.

**Note:** These settings are NOT saved; they must be re-entered each time AgileUpdate is run.

**Agilent Website Access**

If you cannot access the Internet directly with your PNA, you can use an external PC with Internet access to download the file from the Agilent website. You can then transfer the file from your PC to your analyzer over a LAN or other means.

2. Click on the firmware to be downloaded.
3. Save the program to disk (hard drive of your PC).
4. Transfer the file from your PC to your PNA using LAN, CD, or USB Pen drive.
5. Double-click the file on the PNA.
**Warning**: You can save the upgrade file to your PC, but do not attempt to install the PNA application on your PC. It will alter system settings and can result in system crashes.

Last Modified:

- 12-Oct-2009 Added customer release (9.2)
- 13-Mar-2008 Added limitations note
PNA Configurations and Options

Included with each PNA is a mouse and keyboard. This topic presents standard PNA models and the available options and upgrades.

- **PNA Models**
  - **PNA-X Series**
  - **PNA N522x Series** New
  - **PNA-L Series**
  - **Microwave Models**
  - **RF Models** (Discontinued)
  - **mmWave Model**
  - **Measurement Receiver** N5264A

- **Common Options and Upgrade Kits**
- **Warranty Period**

See Also

- **PNA Series Configuration Guide** (requires an Internet connection)
- Click **Help** then **About Network Analyzer** to view the options that are installed on your PNA.

### New PNA 'C' Models

In Spring 2008, new PNA ‘C’ models were introduced that replace the PNA-L and Microwave ‘A’ and ‘B’ models. The new ‘C’ models have the **PNA-X User Interface** capabilities, including touchscreen and softkeys. In addition:

- The ‘C’ models have a bigger 8.4” display screen.
- The ‘C’ models share the same **Specifications** as their ‘A’ and ‘B’ model counterparts.
- Upgrades from ‘A’ and ‘B’ to ‘C’ models are available.

**Note:** The term 'Legacy' that is used in this help file refers to the PNA models that were introduced before the PNA-X (N5230A, E836xA/B, and all discontinued models).
PNA-X Series
See Block Diagrams

- N5241A: 10 MHz to 13.5 GHz
- N5242A: 10 MHz to 26.5 GHz
- N5244A: 10 MHz to 43.5 GHz
- N5245A: 10 MHz to 50.0 GHz
- N5247A: 10 MHz to 67 GHz
- N5264A: Measurement Receiver Learn more

The N5244A, N5245A, N5264A, and N5247A do NOT have a Hibernate mode. Learn more.

PNA-X Options
See options that are common to ALL models

<table>
<thead>
<tr>
<th>Option</th>
<th>Required Model/Options</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>None</td>
<td>2-port model base model. Includes six front-panel access loops.</td>
</tr>
<tr>
<td>219</td>
<td>200</td>
<td>To 2-port base model, add:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>● Bias-tees between each source and each test port</td>
</tr>
<tr>
<td></td>
<td></td>
<td>● Source and receiver attenuators:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>N5241A/42A</td>
</tr>
<tr>
<td></td>
<td></td>
<td>N5244A/45A</td>
</tr>
<tr>
<td></td>
<td></td>
<td>N5247A</td>
</tr>
<tr>
<td>224</td>
<td>200, one of 219 or H85, and 080</td>
<td>To 2-port model, add internal 2nd source, combiner, and mechanical switches.</td>
</tr>
<tr>
<td>400</td>
<td>None</td>
<td>4-port model base model. Includes twelve front-panel access loops.</td>
</tr>
<tr>
<td>Code</td>
<td>Description</td>
<td>Details</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>---------</td>
</tr>
</tbody>
</table>
| 419  | 400 (080 recommended) | To 4-port model, add:  
  - Source and receiver attenuators (see table above for values)  
  - Bias-tees between each source and each test port |
| 423  | 400, one of 419 or H85, and 080 | To 4-port model, add combiner and mechanical switches. |
| 008  | All PNA-X | Pulsed App. [Learn more](#) |
| 020  | All PNA-X | Add IF inputs on the rear panel for antenna and millimeter-wave. |
| 021  | All PNA-X | Add pulse modulator to internal Source1. [Learn more](#) |
| 022  | All PNA-X | Add pulse modulator to internal Source2. [Learn more](#) |
| 025  | All PNA-X | Add four internal pulse generators. [Learn more](#) |
| 028  | All PNA-X and 080 | Noise Figure measurements on amplifiers or converters using a standard PNA receiver. |
| 029  | N5241A or N5242A with 080 and one of the following: 219, 224, 419 or 423 | Noise Figure Application. Adds hardware and firmware for high-accuracy noise figure measurements on amplifiers or converters using internal low-noise receivers or a standard PNA receiver (Opt 028). |
| H29  | N5244A or N5245A with 423 and 080. | Same as Opt 029, but on a N5244A or N5245A. [Learn more](#) |
| 036  | N5244A or N5245A only. | No longer available. Add wide-pulse modulator to internal Source 1. Limits source modulation to 117 ns. |
| 037  | N5244A or N5245A only. | No longer available. Add wide-pulse modulator to internal Source 2. Limits source modulation to 117 ns. |
| H85  | N5242A | Allows high-power measurements up to 20 Watts (+43 dBm). from 10 MHz to 26.5 GHz. Similar to the PNA-X -219 or -419 but deletes the bias tees from the test set. [Learn more](#) |
| 086  | All PNA-X | Gain Compression Application. Adds firmware for fast and accurate gain compression measurements. [Learn more](#) |
| 087  | Opt 224 or 423 Opt 080 | Swept IMD and IM Spectrum. Adds firmware for fast and accurate IMD measurements on amplifiers and converters. [Learn more](#) |
| 088  | All 2-source PNA-X models. | Phase control provides coherent phase measurements. [Learn more](#) |
New PNA N522x Series

See Block Diagrams.

See Specs

- N5221A: 10 MHz to 13.5 GHz
- N5222A: 10 MHz to 26.5 GHz
- N5224A: 10 MHz to 43.5 GHz
- N5225A: 10 MHz to 50.0 GHz
- N5227A: 10 MHz to 67 GHz

The N522x models are identical to the PNA-X series except:

- N522x 2-port models are NOT available with 2 sources.
- N522x option numbering is slightly different.
- N522x models do NOT have internal RF switches or combiners (no RF Path Configuration). This has many measurement implications.
- N522x models do NOT have rear-panel access to RF Paths.
- N522x models do NOT offer a Noise Receiver (Opt 029).

PNA N522x-Options

See options that are common to ALL models

<table>
<thead>
<tr>
<th>Option</th>
<th>Required Model/Options</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>None</td>
<td>2-port model with single RF source.</td>
</tr>
<tr>
<td>400</td>
<td>None</td>
<td>4-port model with two sources.</td>
</tr>
<tr>
<td>201</td>
<td>All Models</td>
<td>Configurable test set. To base 2-port model, adds six front-panel jumpers and R1 reference receiver switch.</td>
</tr>
<tr>
<td>217</td>
<td>NOT available on</td>
<td>Extended power range and configurable test set. To Opt 201, adds</td>
</tr>
</tbody>
</table>
source and receiver attenuators.

<table>
<thead>
<tr>
<th>Model</th>
<th>Source Attn</th>
<th>Receiver Attn</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5221A/22A</td>
<td>0 to 65 dB in 5 dB steps</td>
<td>0 to 35 dB in 5 dB steps</td>
</tr>
<tr>
<td>N5224A/25A</td>
<td>0 to 60 dB in 10 dB steps</td>
<td>0 to 35 dB in 5 dB steps</td>
</tr>
<tr>
<td>N5227A</td>
<td>0 to 50 dB in 10 dB steps</td>
<td>0 to 50 dB in 10 dB steps</td>
</tr>
</tbody>
</table>

**Note:** For N5227A models, the extended power range is available ONLY with Opt 219 and 419.

<table>
<thead>
<tr>
<th>Opt</th>
<th>Model</th>
<th>Feature</th>
</tr>
</thead>
<tbody>
<tr>
<td>219</td>
<td>200</td>
<td>Extended power range, configurable test set, and Bias-tees. To Opt 217, adds bias-tees between each source and each test port.</td>
</tr>
<tr>
<td>401</td>
<td>400</td>
<td>Configurable test set. To base 4-port model, adds 12 front-panel jumpers and R1 reference receiver switch.</td>
</tr>
<tr>
<td>417</td>
<td>400</td>
<td>Extended power range and configurable test set. To Opt 401, adds source and receiver attenuators. See table above for values. <strong>Note:</strong> For N5227A models, the extended power range is available ONLY with Opt 219 and 419.</td>
</tr>
<tr>
<td>419</td>
<td>400 (080 recommended)</td>
<td>Extended power range, configurable test set, and Bias-tees. To Opt 417, adds bias-tees between each source and each test port.</td>
</tr>
<tr>
<td>008</td>
<td>All N522x models</td>
<td>Pulsed App. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td>020</td>
<td>All N522x models</td>
<td>Add <a href="#">IF inputs on the rear panel</a> for antenna and millimeter-wave.</td>
</tr>
<tr>
<td>021</td>
<td>All N522x models</td>
<td>Add pulse modulator to internal Source1. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td>022</td>
<td>All 4-port N522x models NOT available with 2-port models.</td>
<td>Add pulse modulator to internal Source2. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td>025</td>
<td>All N522x models</td>
<td>Add four internal pulse generators. <a href="#">Learn more.</a></td>
</tr>
<tr>
<td>028</td>
<td>All N522x models and 080</td>
<td>Noise Figure measurements on <a href="#">amplifiers</a> or <a href="#">converters</a> using a standard PNA receiver.</td>
</tr>
<tr>
<td>086</td>
<td>All N522x models</td>
<td>Gain Compression Application. Adds firmware for fast and accurate gain compression measurements. <a href="#">Learn more.</a></td>
</tr>
</tbody>
</table>
### Swept IMD and IM Spectrum

Swept IMD and IM Spectrum. Adds firmware for fast and accurate IMD measurements on amplifiers and converters. [Learn more](#).

### Phase Control

Phase control provides coherent phase measurements. [Learn more](#).

### Fast CW Mode

Fast CW mode enables 500 million point data buffer. [Learn more](#).

### iTMSA

iTMSA. Adds firmware for Integrated True Mode Balanced measurements. [Learn more](#).

---

#### PNA-L Series  Model N5230A and N5230C [See note](#)

**Note:** To see if your PNA has a Reference Receiver for each Test Port, scroll down to see the number of test ports and number of reference receivers. If they are equal, then there is a reference receiver for each test port. Block Diagrams are available at the end of each specifications document.

<table>
<thead>
<tr>
<th>Option</th>
<th>Frequency Range</th>
<th>Test Ports</th>
<th>Reference Receivers</th>
<th>Connector</th>
<th>Test Set</th>
</tr>
</thead>
<tbody>
<tr>
<td>020</td>
<td>300 KHz* to 6 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Standard</td>
</tr>
<tr>
<td>025</td>
<td>300 KHz* to 6 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Configurable</td>
</tr>
<tr>
<td>120</td>
<td>300 KHz* to 13 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Standard</td>
</tr>
<tr>
<td>125</td>
<td>300 KHz* to 13 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Configurable</td>
</tr>
<tr>
<td>140</td>
<td>300 KHz* to 13.5 GHz</td>
<td>4</td>
<td>1</td>
<td>3.5 mm Male</td>
<td>Standard</td>
</tr>
<tr>
<td>145</td>
<td>300 KHz* to 13.5 GHz</td>
<td>4</td>
<td>1</td>
<td>3.5 mm Male</td>
<td>Configurable²</td>
</tr>
<tr>
<td>146</td>
<td>300 KHz* to 13.5 GHz</td>
<td>4</td>
<td>1</td>
<td>3.5 mm Male</td>
<td>Configurable³ + Second Source</td>
</tr>
<tr>
<td>220</td>
<td>10 MHz* to 20 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Standard</td>
</tr>
<tr>
<td>225</td>
<td>10 MHz* to 20 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>Configurable</td>
</tr>
<tr>
<td>240</td>
<td>300 KHz* to 20 GHz</td>
<td>4</td>
<td>1</td>
<td>3.5 mm Male</td>
<td>Standard</td>
</tr>
<tr>
<td>245</td>
<td>300 KHz* to 20 GHz</td>
<td>4</td>
<td>1</td>
<td>3.5 mm Male</td>
<td>Configurable²</td>
</tr>
<tr>
<td>PNA Model</td>
<td>Frequency Range</td>
<td>Test Ports</td>
<td>Reference Receivers</td>
<td>Connector Type</td>
<td>Front Panel Jumpers</td>
</tr>
<tr>
<td>-----------</td>
<td>----------------------------------</td>
<td>------------</td>
<td>---------------------</td>
<td>----------------</td>
<td>---------------------</td>
</tr>
<tr>
<td>E8361A</td>
<td>10 MHz* to 67 GHz (tunable to 70 GHz)**</td>
<td>2</td>
<td>2</td>
<td>1.85 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8361C</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8362A</td>
<td>45 MHz to 20 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8363A</td>
<td>45 MHz to 40 GHz</td>
<td>2</td>
<td>2</td>
<td>2.4 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8364A</td>
<td>45 MHz to 50 GHz</td>
<td>2</td>
<td>2</td>
<td>2.4 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8362B</td>
<td>10 MHz* to 20 GHz</td>
<td>2</td>
<td>2</td>
<td>3.5 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8362C</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8363B</td>
<td>10 MHz* to 40 GHz</td>
<td>2</td>
<td>2</td>
<td>2.4 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8363C</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8364B</td>
<td>10 MHz* to 50 GHz</td>
<td>2</td>
<td>2</td>
<td>2.4 mm Male</td>
<td>0</td>
</tr>
<tr>
<td>E8364C</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* Typical specs apply

**Test Set**

**Standard** NO Configurable Test Set.

**Configurable (2-port)** - Adds six front panel access loops and two 60 dB step attenuators in 10 dB steps.

**Configurable (4-port)** - Adds nine front panel access loops and one 60 dB step attenuator in 10 dB steps.

**Configurable (4-port)** - Adds nine front panel access loops and two 60 dB step attenuators in 10 dB steps.

**Microwave Standard Models** *(see options)*

Block Diagrams are available at the end of each specifications document.
RF Standard Models (see options)
Block Diagrams are available at the end of each specifications document.

<table>
<thead>
<tr>
<th>PNA Model</th>
<th>Frequency Range</th>
<th>Ports</th>
<th>Connector Type</th>
<th>Reference Receivers</th>
<th>Front Panel Jumpers</th>
</tr>
</thead>
<tbody>
<tr>
<td>E8356A</td>
<td>300 kHz to 3 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>E8357A</td>
<td>300 kHz to 6 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>E8358A</td>
<td>300 kHz to 9 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>E8801A</td>
<td>300 kHz to 3 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>E8802A</td>
<td>300 kHz to 6 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>E8803A</td>
<td>300 kHz to 9 GHz</td>
<td>2</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>N3381A</td>
<td>300 kHz to 3 GHz</td>
<td>3</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>N3382A</td>
<td>300 kHz to 6 GHz</td>
<td>3</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>N3383A</td>
<td>300 kHz to 9 GHz</td>
<td>3</td>
<td>Type-N Female</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

All RF models are no longer produced (June 1, 2005).

Millimeter Wave PNA
Block Diagrams are available at the end of each specifications document.
### Measurement Receiver

Block Diagrams are available at the end of each specifications document.

<table>
<thead>
<tr>
<th>PNA Model</th>
<th>Frequency Range</th>
<th>Ports</th>
<th>Connector Type</th>
<th>Reference Receivers</th>
<th>Front Panel jumpers</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5264A</td>
<td>IF Frequencies only</td>
<td>0</td>
<td>N/A</td>
<td>1</td>
<td>N/A</td>
</tr>
</tbody>
</table>

#### Options

108 - Built-in 26.5 GHz LO source with +10 dBm output power.

118 - Fast-CW mode enables 500 million point data buffer.

### Common Options and Upgrade Kits for ALL models

The following options are installed at the time of purchase, and some are also available after the initial purchase of a PNA. To order an upgrade, contact your Agilent representative.

<table>
<thead>
<tr>
<th>Option</th>
<th>Supported Models</th>
<th>Description</th>
</tr>
</thead>
</table>

---

* Typical specs apply from 10 to 45 MHz
* Test heads to 325 GHz are also available

Upgrade your existing E836xB with the following:

- **H11 option**
- N5260A Test Set
- Millimeter-Wave VNA Frequency Extension Modules from Oleson Microwave Labs Extension Modules
<table>
<thead>
<tr>
<th>Code</th>
<th>Part Number</th>
<th>Description</th>
<th>Additional Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>006</td>
<td>E8356A</td>
<td>Upgrade to 6 GHz frequency range.(No longer available)</td>
<td>Adds 6 GHz operation. Includes installation at an Agilent service center. Instrument calibration (required after frequency upgrade) is available for an additional fee.</td>
</tr>
<tr>
<td>009</td>
<td>E8356/7A</td>
<td>Upgrade to 9 GHz frequency range.(No longer available)</td>
<td>Adds 9 GHz operation. Includes installation at an Agilent service center. Instrument calibration (required after frequency upgrade) is available for an additional fee.</td>
</tr>
<tr>
<td>010</td>
<td>All</td>
<td>Time-domain</td>
<td>Adds time-domain capability to analyzer. The serial number of the analyzer must be specified when ordering this kit. Software upgrade. <a href="#">Learn more about Time Domain</a> <a href="#">Learn how this option is enabled</a></td>
</tr>
<tr>
<td>014</td>
<td>E8361A/C</td>
<td>Configurable test set</td>
<td>Adds front panel access to the source output and coupler input on test ports 1 and 2. Adds front panel access directly to all receivers, including the reference receiver. Upgrade includes installation at an Agilent service center. All PNA-X models include front panel access.</td>
</tr>
<tr>
<td>015</td>
<td>E8356/57/58A</td>
<td>Configurable test set (No longer available)</td>
<td>Adds front panel access to the source output and coupler input on test ports 1 and 2. Adds 35 dB step attenuators between the couplers and receivers. Upgrade includes installation at an Agilent service center.</td>
</tr>
<tr>
<td>016</td>
<td>E8361A/C</td>
<td>Receiver Step Attenuators</td>
<td>Adds two step attenuators. (E8361A adds 50 dB step attenuators with 10 dB resolution; all other PNA models add 35 dB step attenuators with 5 dB resolution.) Each attenuator is inserted between a test port and its corresponding receiver. Requires option UNL.</td>
</tr>
<tr>
<td>022</td>
<td>E8361A/C</td>
<td>Extended Memory</td>
<td>Adds more RAM for a total of 512MB. <a href="#">Learn more about PNA RAM</a></td>
</tr>
<tr>
<td>080</td>
<td>All</td>
<td>Frequency Offset</td>
<td>Enables you to set the PNA source independently from where the receivers are tuned. This capability is important for measuring mixers, converters, and amplifiers. To control the additional hardware, a basic user interface, SCPI, and COM commands are provided. Requires option 014 or Active test set</td>
</tr>
<tr>
<td>Model</td>
<td>Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8361A/C</td>
<td><strong>Reference Switch</strong> Add an internal solid state switch in the R1 reference path for controlling an external reference mixer. Requires option 014 and opt 080.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8362B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8363B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8364B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>See specifications for a block diagram which includes the reference switch. All PNA-X models include this reference switch.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>082</td>
<td><strong>Scalar Mixer Measurements</strong> Allows Only the Scalar Mixer Converter (SMC) portion of the Frequency Converter Measurement Application. Provides the same intuitive user-interface, easy calibration, and external source control for making fixed and swept LO Scalar Mixer measurements. When used with a multiport PNA or external test set, SMC is only available on PNA ports 1 and 2. Requires Opt 080.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8361A/C</td>
<td><strong>Frequency Converter Measurement Application</strong> Provides an intuitive user-interface for making extremely accurate conversion loss and absolute group delay measurements on mixers and converters. Exceptional amplitude and phase accuracy is achieved through two new calibration techniques: Scalar Mixer Calibration and Vector Mixer Calibration. The application also provides automatic control of all of Agilent's major signal sources. Requires option 014, 080, and 081.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8362B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8363B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8364B/C</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PNA-X</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Embedded LO</strong> Provides the ability to measure frequency converters that have an embedded LO. Requires at least one Converter App option. Learn more</td>
<td></td>
<td></td>
</tr>
<tr>
<td>098</td>
<td><strong>CPU board upgrade (No longer available)</strong> Replaces the 266 MHz CPU board with a 500 MHz CPU board. Upgrade includes installation at an Agilent service center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8356/ 57/ 58A</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>108</td>
<td><strong>70 dB step attenuator (No longer available)</strong> Adds a single 70 dB step attenuator that is switched between the source and each output port to extend the output power to -85 dBm. Note: Two 70 dB step attenuators are standard equipment with PNA models E8356/ 57/ 58A. Upgrade includes installation at an Agilent service center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E8801/ 02/ 03A</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N3381/ 82/ 83A</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Code</td>
<td>Options</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>---------</td>
<td>-------------</td>
<td></td>
</tr>
</tbody>
</table>
| 1E5  | E8801/ 02/ 03A N3381/ 82/ 83A | High stability 10 MHz time base (No longer available)  
Replaces 10 ppm time base with a 1 ppm time base. High stability time base is standard with PNA models E8356/ 57/ 58A and E8362/ 63/ 64B. |
| 550  | E8361A/C E8362B/C E8363B/C E8364B/C N5230A/C | Add full 4 port capability and differential measurements - [Learn more.](#)  
Requires a PNA with configurable test set. [E836x Opt 014](#) or [N5230A Opt. x25](#) |
| 551  | All PNA-X, C models, and N522x models. | Add fully integrated measurements at all of the available test ports.  
Solutions and PNA requirements depend on the supported test set. See [http://www.agilent.com/find/multiport](http://www.agilent.com/find/multiport) |
| 897  | All PNA-X, C models, and N522x models. | Perpetual license for built-in performance test software for [Agilent inclusive cal.](#)  
Adds built-in performance testing and calibration software for self-maintainers. Requires additional equipment. See your PNA Service Guide for more information on equipment required. |
| 898  | All PNA-X, C models, and N522x models. | Perpetual license for built-in performance test software for [Standards compliant cal.](#)  
Adds built-in performance testing and calibration software for self-maintainers. Requires additional equipment. See your PNA Service Guide for more information on equipment required. |
| H08  | All PNA-X and C Models | Integrated Pulsed Measurement Setup  
Provides the Pulsed Application for average pulse and point-in-pulse measurements.  
PNA-X models use the [Integrated Pulse Setup](#)  
E836x models require options H11, UNL, 014, 080, and 081. [Learn more](#)  
See PNA-X options for more information. |
| H11  | E8361A/C E8362B/C E8363B/C E8364B/C | IF Access  
Provides rear panel access to the PNA IF paths for:  
- Extended frequency coverage to 325 GHz  
- Pulsed measurement capability  
- Advanced antenna measurements  
Requires options UNL, 014, 080, and 081. [Learn more](#) |
Extended power range and bias tees

Add two step attenuators and two bias tees. (E8361A adds 50 dB step attenuators; E8362/63/64A/B adds 60 dB step attenuators. All attenuators have a 10 dB resolution.) A step attenuator and bias-tee set is inserted between the source and test port 1 and another set between the source and test port 2.

Upgrade includes installation at an Agilent service center.

Certification Options

<table>
<thead>
<tr>
<th>Option</th>
<th>Supported Models</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>UK6</td>
<td>ALL except N5250A</td>
<td>Complete set of measurement data which was acquired from testing your PNA to published specifications. Includes calibration label, calibration certificate, and data report. Conforms to ISO 9001.</td>
</tr>
<tr>
<td>1A7</td>
<td>ALL except N5250A</td>
<td>Complete set of measurement data which was acquired from testing your PNA to published specifications. Includes calibration label, ISO 17025 calibration certificate, data report, measurement uncertainties and guard bands on all specifications. Conforms to ISO17025 and ISO 9001.</td>
</tr>
</tbody>
</table>

Documentation and Localization Options

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A documentation CD-ROM is no longer included with each PNA shipment (Feb.2005).</td>
</tr>
<tr>
<td>To download a service guide for your PNA, or the latest version of PNA Help, visit <a href="http://www.agilent.com/find/pna">www.agilent.com/find/pna</a>, search for your PNA model, then click Library.</td>
</tr>
</tbody>
</table>

PNA Warranty Period

The actual warranty on your instrument depends on the date it was ordered as well as whether or not any warranty options were purchased at that time. To determine the exact warranty on your instrument, contact Agilent Technologies with the model and serial number of your instrument.

For online information about Agilent's service and support products visit: [www.agilent.com/find/tm_services](http://www.agilent.com/find/tm_services).
21-Jun-2011  Updated link to new config guide
29-Apr-2011  Added new N522x models
2-Nov-2010   Added link to config guide, N5247A, and 088
3-Mar-2010   Added 028 and H29
17-Dec-2009  Corrected Certification Options
26-Oct-2009  Added 029 requirements
30-Apr-2009  Added new PNA-X models and
             PNA-X w/ Opts 081 and 014
8-Sep-2008   Added IMD
9-Jun-2008   Edits for C models and N5242A
25-Oct-2007  Added C models
5-Sep-2007   Added Embedded LO
9/26/06      MQ Added PNA-L 4port models
9/27/06      MX Added PNA-X
Option Enable Utility

The Option Enable utility allows you to perform the following activities on your PNA.

- Enable or remove software options and some hardware options.
- Recover option data if the hard drive or other data-containing assembly is replaced.
- Input or change a serial number.

The following items are discussed in this topic:

**Keywords**

- **Running the Program**
- **Removing an Option**
- **Installing an Option**
- **Repairing and Recovering Option Data**
- **Installing or Changing the Serial Number**

**See Also**

See PNA Configurations and Options

**Keywords**

To add certain options, you need a keyword that is provided by Agilent. There are two types of keywords:

- **Option Keywords** add a software option.
- **Model Keywords** may be required if you replace multiple assemblies.

Keywords are linked to the PNA **Host ID**, which is displayed on the Option Enable dialog box (below).

**Temporary and Permanent Options**

Any software option can also be installed on a temporary basis for a specified amount of time. This allows you to evaluate a specific feature or capability at no cost.

If the license key provided by Agilent has an expiration date, you must select the "temporary" option and enter the expiration date exactly as stated in the license statement. If you decide to make this option permanent, Agilent will provide a new keyword that converts the option to permanent status.

For either permanent or temporary software options, a provided keyword must be entered.

**Running the Program**
On the PNA, click System, point to Service, then click Option Enable.

1. To enable or remove an option, select it from the drop-down list of available options. If the desired option is not listed, select the last choice in the list, labeled Enter Unlisted Option.

2. Enter the 3-character option name and click Enter.

If a software option was chosen, the following occurs.

- The Remove button will be enabled.
- The keyword entry area becomes visible.
- The permanent/temporary selection is enabled.

If a hardware option is selected, the following occurs.

- With the hardware option already installed, the Remove button is enabled.
- With the hardware option not installed, the Enable button is enabled.

Removing an Option

1. To remove an option, click Remove.

2. After the option is removed, restart the network analyzer application for the changes to take effect.

Note: Removal of a licensed option (such as Option 010, Time Domain) will permanently remove the license.
keyword. If this option may be needed in the future, then record the license keyword before removing the option. Do this by copying the file "gen.lic" to another location (such as a floppy disk), or print it using notepad. The file, located at “C:/Program Files/Agilent/Network Analyzer” contains all the information needed to recreate the license.

Installing an Option

1. If the keyword entry area is visible, enter a keyword. (The keyword is not case sensitive.)
2. Click Enable.
3. After the option is installed, restart the network analyzer application for the changes to take effect.

Note: If a desired option is not visible, it may be because a prerequisite option has not yet been installed. For example, Option 083 will not be visible if Option 080 is not already present. See PNA options.

Repairing and Recovering Option Data

Use this part of the Option Enable Utility in the following situations:

- If the hard drive is replaced
- If the frequency reference assembly is replaced

This routine rebuilds the option information contained on the hard drive and frequency reference assembly (primary and backup).

1. Select Repair from the Option Enable menu bar.

Note: If you are unsure if this routine needs to be done, run it; no harm will result.

2. The model and serial number are displayed, along with four check boxes.
3. Select the boxes that apply.
4. Click Begin Repair. The routine checks all data files and performs any needed repairs. You may be asked to verify certain information and processes.
5. If the routine finds that the model number is incorrect or invalid, you will be asked to select the correct model number.
   - Along with this model number, a model keyword will be required. If this is not labeled on the analyzer, or is not otherwise known, contact Agilent
   - After you have entered the requested data, click Change Model. This process takes about 30 seconds.
6. When done, click Exit Repair.
7. If you do not need to install any other options, click Exit.
Installing or Changing the Serial Number

It may be necessary to install or change a serial number if certain assemblies are replaced.

1. To change the serial number, select Change Serial from the Option Enable menu bar. The current serial number will be displayed. If no serial number has previously been entered, the word "NONE" will be displayed.

2. Type the new serial number into the space provided, and click Change Serial. (The serial number is not case sensitive.)

**Note:** Use extreme care when entering the serial number; only one entry chance is allowed!

3. To change an incorrect serial number, a clear-code password is required. Contact Agilent to obtain this clear code and have the existing serial number available. Enter the clear code in the space provided, along with the new serial number, then click Change Serial.

---

Last Modified:

20-Sep-2007   Added Install note
**Instrument Calibration**

An instrument calibration is a process where the PNA performance is measured to ensure that it operates within specifications. If any performance parameter does not conform to the published specifications, adjustments are made to bring the performance into conformance.

**Why Should I Get an Instrument Calibrated?**

Over time, the active components in the analyzer age and the performance may degrade or drift.

To ensure that the analyzer is performing to the published specifications, you should have an instrument calibration performed periodically.

**How Often Should I Get an Instrument Calibrated?**

It is your responsibility to determine the calibration period which best meets your requirements. However, a 12 to 18 month calibration cycle is appropriate for most users.

There are two things to consider: performance drift and connector wear.

- The instrument specifications are set to consider the performance drift that may occur over a 24 month period. Therefore, getting the instrument calibrated at 24 month intervals ensures that the analyzer maintains performance within the operating specifications. If you need the analyzer to maintain more consistent operation, you may want to have the instrument calibrated more often than the recommended 24-month interval.

- Connector wear is a bigger factor and depends on the number of connections that are made. The test ports become noticeably worn after 500 to 700 connections. This could represent about 12 months with average use. With more frequent connections, the calibration cycle should be sooner. You can extend the time between calibrations and thereby save money by using connector savers and by performing proper Connector Care.

**How Do I Get an Instrument Calibrated?**

To get the instrument calibrated, send it to one of the Agilent Technologies service centers. See Technical Support.

The PNA must be fully functional when it is sent to the service center, or they will charge for their repair services. If the PNA is being used in a secure environment where the hard drive can not be sent with the PNA, a second hard drive must be purchased and configured for use with the PNA in an "unclassified" environment before the PNA is sent to the service center.

To perform the instrument calibration yourself, you must have the following required items:

- Instrument Calibration Test Equipment
- Performance Test Software

**What Are My Choices of Instrument Calibration?**

The following types of instrument calibration are available from Agilent Technologies at the time of initial order:
Standard

Includes a certificate of calibration stating the instrument has been calibrated and is operating within the published specifications.

Option UK6

Available ONLY at the initial shipment. Includes the test data from the calibration and the certificate of calibration stating the instrument has been calibrated and is operating within the published specifications.

Option A6J

Available ONLY at the initial shipment. Includes the test data and measurement uncertainties from the calibration and the certificate of calibration stating the instrument has been calibrated using a process in compliance with ANSI Z540 and is operating within the published specifications.

Option 1A7

Available ONLY at the initial shipment. Includes the test data and measurement uncertainties from the calibration and the certificate of calibration stating the instrument has been calibrated using a process in compliance with ISO 17025 and is operating within the published specifications.

The following types of instrument calibration are available from Agilent Technologies service center:

Agilent Calibration

Includes the test data from the calibration and the certificate of calibration, stating the instrument has been calibrated and is operating within the published specifications.

ANSI Z540 Calibration

Includes the test data from the calibration and the certificate of calibration, stating the instrument has been calibrated using a process in compliance with ANSI Z540.1 and is operating within the published specifications.

ISO 17025 Calibration

Includes the test data from the calibration and the certificate of calibration, stating the instrument has been calibrated using a process in compliance with ISO 17025 and is operating within the published specifications.

For more information on these options, visit www.agilent.com/find/calibration.

Last Modified:

24-Mar-2010 Updated options and cal cycle
Other Resources

The following network analysis resources are also available.

Document Resources

Application Notes
You can also access application notes at this URL:
http://www.agilent.com/find/PNA

Third-Party Resources

For information about test fixtures and part handlers, contact:

   Inter-Continental Microwave
   www.icmicrowave.com

For information about probing equipment and accessories, contact:

   Cascade Microtech, Inc.
   www.cascademicrotech.com
**SCPI Errors**

**SCPI Errors**

- 100 to -200 Command Errors
- 200 to -299 Execution Errors
- 300 to -399 SCPI Specified Device-Specific Errors
- 400 to -800 Query and System Errors
- 100 to 200 PNA-specific Errors

**See Also**

PNA Error messages.

---

**-100 to -200 Command Errors**

A command error indicates that the test set’s GPIB parser has detected an IEEE 488.2 syntax error. When one of these errors is generated, the command error bit in the event status register is set.

- **-100 std_command**
  Command - This event bit (Bit 5) indicates a syntax error, or a semantic error, or a GET command was entered, see IEEE 488.2, 11.5.1.1.4.

- **-101 std_invalidChar**
  Invalid character - Indicates a syntactic elements contains a character which is invalid for that type.

- **-102 std_syntax**
  Syntax - Indicates that an unrecognized command or data type was encountered. For example, a string was received when the device does not accept strings.

- **-103 std_invalidSeparator**
  Invalid separator - The parser was expecting a separator and encountered an illegal character. For example, the semicolon was omitted after a program message unit.

- **-104 std_wrongParamType**
  Data type - The parser recognized a data element different than one allowed. For example, numeric or string data was expected but block data was encountered.

- **-105 std_GETNotAllowed**
  GET not allowed - Indicates a Group Execute Trigger was received within a program message. Correct the program so that the GET does not occur within the program code.

- **-108 std_tooManyParameters**
  Parameter not allowed - Indicates that more parameters were received than expected for the header. For example, *ESE common command only accepts one parameter, so *ESE 0,1 is not allowed.

- **-109 std_tooFewParameters**
  Missing parameter - Indicates that less parameters were received than required for the header. For example, *ESE requires one parameter, *ESE is not allowed.
-110 std_cmdHeader Command header - Indicates an error was detected in the header. This error is used when the device cannot detect the more specific errors -111 through -119.

-111 std_headerSeparator Header separator - Indicates that a character that is not a legal header separator was encountered while parsing the header.

-112 std_IDTooLong Program mnemonic too long - Indicates that the header contains more that twelve characters, see IEEE 488.2, 7.6.1.4.1.

-113 std_undefinedHeader Undefined header - Indicates the header is syntactically correct, but it is undefined for this specific device. For example, "XYZ" is not defined for any device.

-114 std_suffixOutOfRange Header suffix out of range - Indicates the value of a header suffix attached to a program mnemonic makes the header invalid.

-120 std_numericData Numeric data - This error, as well as errors

-121 std_invalidCharInNumber Invalid character in number - Indicates an invalid character for the data type being parsed was encountered. For example, an alpha in a decimal numeric or a "9" in octal data.

-123 std_exponentTooLarge Exponent too large - Indicates the magnitude of an exponent was greater than 32000, see IEEE 488.2, 7.7.2.4.1.

-124 std_decimalTooLong Too many digits - Indicates the mantissa of a decimal numeric data element contained more than 255 digits excluding leading zeros, see IEEE 488.2, 7.7.2.4.1.

-128 std_numericNotAllowed Numeric data not allowed - Indicates that a legal numeric data element was received, but the device does not accept one in this position for the header.

-130 std_suffix Suffix - This error, as well as errors -131 through -139, are generated when parsing a suffix. This particular error message is used if the device cannot detect a more specific error.

-131 std_badSuffix Invalid suffix - Indicates the suffix does not follow the syntax described in IEEE 488.2, 7.7.3.2, or the suffix is inappropriate for this device.

-134 std_suffixTooLong Suffix too long - Indicates the suffix contain more than 12 characters, see IEEE 488.2, 7.7.3.4.

-138 std_suffixNotAllowed Suffix not allowed - Indicates that a suffix was encountered after a numeric element that does not allow suffixes.

-140 std_charData Character data - This error, as well as errors

-141 std_invalidCharData Invalid character data - Indicates that the character data element contains an invalid character or the particular element received is not valid for the header.

-144 std_charDataTooLong Character data too long - Indicates the character data element contains more than twelve characters, see IEEE 488.2, 7.7.1.4.
-148  std_charNotAllowed  Character data not allowed - Indicates a legal character data element was encountered where prohibited by the device.

-150  std_stringData  String data - This error, as well as errors

-151  std_stringInvalid  Invalid string data - Indicates that a string data element was expected, but was invalid, see IEEE 488.2, 7.7.5.2. For example, an END message was received before the terminal quote character.

-158  std_stringNotAllowed  String data not allowed - Indicates that a string data element was encountered but was not allowed by the device at this point in parsing.

-160  std_blockData  Block data - This error, as well as errors -161 through -169, are generated when parsing a block data element. This particular error message is used if the device cannot detect a more specific error.

-161  std_badBlock  Invalid block data - Indicates a block data element was expected, but was invalid, see IEEE 488.2, 7.7.6.2. For example, and END message was received before the end length was satisfied.

-168  std_blockNotAllowed  Block data not allowed - Indicates a legal block data element was encountered, but not allowed by the device at this point in parsing.

-170  std_expr  Expression - This error, as well as errors -171 through -179, are generated when parsing an expression data element. This particular error message is used if the device cannot detect a more specific error.

-171  std_invalidExpression  Invalid expression - Indicates the expression data element was invalid, see IEEE 488.2, 7.7.7.2. For example, unmatched parentheses or an illegal character.

-178  std_exprNotAllowed  Expression data not allowed - Indicates a legal expression data was encountered, but was not allowed by the device at this point in parsing.

-180  std_macro  Macro - This error, as well as error -181 through -189, are generated when defining a macro or execution a macro. This particular error message is used if the device cannot detect a more specific error.

-181  std_validOnlyInsideMacro  Invalid outside macro definition - Indicates that a macro parameter place holder was encountered outside of a macro definition.

-183  std_invalidWithinMacro  Invalid inside macro definition - Indicates that the program message unit sequence, sent with a *DDT or a *DMC command, is syntactically invalid, see IEEE 488.2, 10.7.6.3.

-184  std_macroParm  Macro parameter - Indicates that a command inside the macro definition had the wrong number or type of parameters.

**-200 to -299 Execution Errors**

These errors are generated when something occurs that is incorrect in the current state of the instrument. These errors may be generated by a user action from either the remote or the manual user interface.
Execution - This event bit (Bit 4) indicates a PROGRAM DATA element following a header was outside the legal input range or otherwise inconsistent with the device's capabilities, see IEEE 488.2, 11.5.1.1.5.

Invalid while in local

Settings lost due to rtl

Command protected - Indicates that a legal password-protected program command or query could not be executed because the command was disabled.

Trigger

Trigger ignored

Arm ignored

Init ignored

Trigger deadlock

Arm deadlock

Parameter - Indicates that a program data element related error occurred.

Settings conflict - Indicates that a legal program data element was parsed but could not be executed due to the current device state.

Data out of range - Indicates that a legal program data element was parsed but could not be executed because the interpreted value was outside the legal range defined by the devices.

Too much data - Indicates that a legal program data element of block, expression, or string type was received that contained more data than the device could handle due to memory or related device-specific requirements.

Illegal parameter value - Indicates that the value selected was not part of the list of values given.

Out of memory - The device has insufficient memory to perform the requested operation.

Lists not same length - Attempted to use LIST structure having individual LIST's of unequal lengths.

Data corrupt or stale - Indicates invalid data, a new reading started but not completed since the last access.

Data questionable - Indicates that measurement accuracy is suspect.
-232 std_invalidFormat  Invalid format

-233 std_invalidVersion  Invalid version - Indicates that a legal program data element was parsed but could not be executed because the version of the data is incorrect to the device. For example, a not supported file version, a not supported instrument version.

-240 std_hardware  Hardware - Indicates that a legal program command or query could not be executed because of a hardware problem in the device.

-241 std_hardwareMissing  Hardware missing - Indicates that a legal program command or query could not be executed because of missing device hardware. For example, an option was not installed.

-250 std_massStorage  Mass storage - Indicates that a mass storage error occurred. The device cannot detect the more specific errors described for errors -251 through -259.

-251 std_missingMassStorage  Missing mass storage - Indicates that a legal program command or query could not be executed because of missing mass storage.

-252 std_missingMedia  Missing media - Indicates that a legal program command or query could not be executed because of missing media. For example, no disk.

-253 std_corruptMedia  Corrupt media - Indicates that a legal program command or query could not be executed because of corrupt media. For example, bad disk or wrong format.

-254 std_mediaFull  Media full - Indicates that a legal program command or query could not be executed because the media is full. For example, there is no room left on the disk.

-255 std_directoryFull  Directory full - Indicates that a legal program command or query could not be executed because the media directory was full.

-256 std_fileNotFound  File name not found - Indicates that a legal program command or query could not be executed because the file name was not found on the media.

-257 std_fileName  File name - Indicates that a legal program command or query could not be executed because the file name on the device media was in error. For example, an attempt was made to read or copy a nonexistent file.

-258 std_mediaProtected  Media protected - Indicates that a legal program command or query could not be executed because the media was protected. For example, the write-protect switch on a memory card was set.

-260 std_expression  Expression

-261 std_math  Math in expression

-270 std_macroExecution  Macro - Indicates that a macro related execution error occurred.
-271 std_macroSyntax Macro syntax - Indicates that a syntactically legal macro program data sequence, according to IEEE 488.2, 10.7.2, could not be executed due to a syntax error within the macro definition.

-272 std_macroExec Macro execution - Indicates that a syntactically legal macro program data sequence could not be executed due to some error in the macro definition, see IEEE 488.2, 10.7.6.

-273 std_badMacroName Illegal macro label - Indicates that the macro label was not accepted, it did not agree with the definition in IEEE 488.2, 10.7.3

-274 std_macroPlaceholder Macro parameter - Indicates that the macro definition improperly used a macro parameter placeholder, see IEEE 4882, 10.7.3.

-275 std_macroTooLong Macro definition too long - Indicates that a syntactically legal macro program data sequence could not be executed because the string of block contents were too long for the device to handle, IEEE 488.2, 10.7.6.1.

-276 std_macroRecursion Macro recursion - Indicates that a syntactically legal macro program data sequence count not be executed because it would be recursive, see IEEE 488.2, 10.7.6.6.

-277 std_cantRedefineMacro Macro redefinition not allowed - Indicates that redefining an existing macro label, see IEEE 488.2, 10.7.6.4.

-278 std_macroNotFound Macro header not found - Indicates that a legal macro label in the *GMS?, see IEEE 488.2, 10.13, could not be executed because the header was not previously defined.

-280 std_program Program

-281 std_cantCreateProgram Cannot create program

-282 std_illegalProgramName Illegal program name

-283 std_illegalVarName Illegal variable name

-284 std_programRunning Program currently running

-285 std_programSyntax Program syntax

-286 std_programRuntime Program runtime

-290 std_memoryUse Memory use

-291 std_execOutOfMemory Out of memory

-292 std_nameNotFound Referenced name does not exist
**-300 to -399 SCPI Specified Device-Specific Errors**

A device-specific error indicates that the instrument has detected an error that occurred because some operations did not properly complete, possibly due to an abnormal hardware or firmware condition. For example, an attempt by the user to set an out of range value will generate a device specific error. When one of these errors is generated, the device specific error bit in the event status register is set.

- **-300 std_deviceSpecific**  
  Device specific - This event bit (Bit 3) indicates that a device operation did not properly complete due to some condition, such as overrange see IEEE 488.2, 11.5.1.1.6.

- **-310 std_system**  
  System

- **-311 std_memory**  
  Memory - Indicates some physical fault in the devices memory, such as a parity error.

- **-312 std_PUDmemoryLost**  
  PUD memory lost - Indicates protected user data saved by the *PUD command has been lost, see IEEE 488.2, 10.27.

- **-313 std_calMemoryLost**  
  Calibration memory lost - Indicates that nonvolatile calibration data used by the *CAL? command has been lost, see IEEE 488.2, 10.2.

- **-314 std_savRclMemoryLost**  
  Save/recall memory lost - Indicates that the nonvolatile data saved by the *SAV command has been lost, see IEEE 488.2, 10.33.

- **-315 std_configMemoryLost**  
  Configuration memory lost - Indicates that nonvolatile configuration data saved by the device has been lost.

- **-320 std_storageFault**  
  Storage fault - Indicates that the firmware detected a fault when using data storage. This is not an indication of physical damage or failure of any mass storage element.

- **-321 std_outOfMemory**  
  Out of memory - An internal operation needed more memory than was available

- **-330 std_selfTestFailed**  
  Self-test failed - Indicates a problem with the device that is not covered by a specific error message. The device may require service.

- **-340 std_calFailed**  
  Calibration failed - Indicates a problem during calibration of the device that is not covered by a specific error.

- **-350 std_queueOverflow**  
  Queue overflow - Indicates that there is no room in the queue and an error occurred but was not recorded. This code is entered into the queue in lieu of the code that caused the error.

- **-360 std_comm**  
  Communication - This is the generic communication error for devices that cannot detect the more specific errors described for error -361 through -363.

- **-361 std_parity**  
  Parity in program message - Parity bit not correct when data received for example, on a serial port.
-362 std_framing Framing in program message - A stop bit was not detected when data was received for example, on a serial port (for example, a baud rate mismatch).

-363 std_inputBufferOverrun Input buffer overrun - Software or hardware input buffer on serial port overflows with data caused by improper or nonexistent pacing.

**-400 to -800 Query and System Errors**

A Query error is generated either when data in the instrument's GPIB output queue has been lost, or when an attempt is being made to read data from the output queue when no output is present or pending.

-400 std_queryGen Query - This event bit (Bit 2) indicates that an attempt to read data from the Output Queues when no output is present or pending, to data in the Output Queue has been lost see IEEE488.2, 11.5.1.1.7.

-410 std_interrupted Query INTERRUPTED - Indicates the test set has been interrupted by a new program message before it finishes sending a RESPONSE MESSAGE see IEEE 488.2, 6.3.2.3.

-420 std_unterminated Query UNTERMINATED - Indicates an incomplete Query in the program see IEEE 488.2, 6.3.2.2.

-430 std_deadlocked Query DEADLOCKED - Indicates that the Input Buffer and Output Queue are full see IEEE 488.2, 6.3.1.7.

-440 std_responseNotAllowed Query UNTERMINATED after indefinite response - Indicates that a query was received in the same program message after a query requesting an indefinite response was executed see IEEE 488.2, 6.5.7.5.

-500 std_powerOn Power on

-600 std_userRequest User request

-700 std_requestControl Request control

-800 std_operationComplete Operation complete

**PNA Specific (Positive) SCPI Errors**

100 dupWindNum "Duplicate window number"

101 windNumNotFound "Window number not found"

102 failedWindCreate "Window creation failed"

103 noCalcParamSelection "CALC measurement selection set to none" 
   See CALC:PAR:SEL

104 dupMeasName "Duplicate measurement name"
105 dataNotFound  "Requested data not available"
106 measNotFound  "Requested measurement not found"
107 traceNotFound  "Requested trace not found"
108 notImplemented  "Mnemonic not yet implemented"
109 noDocument  "No measurement container found"
110 dupTraceNum  "Duplicate trace number"
111 titleStrTooLong  "Title string exceeds 50 characters"
112 memoryNotFound  "Requested memory not found"
113 exceedMaxTraces  "Exceeded the maximum number of traces per window"
114 SerNumNotFound  "The serial number was not found. Please store the serial number."
115 LoadFailed  "The state was not loaded. Please check the file name."
116 StoreFailed  "The state was not stored. Please check the file and path names."
117 File  "An in the File operation occurred. Please check file and path names."
118 measChanConflict  "Measurement does not belong to specified channel."
119 exceedMaxWindows  "Exceeded the maximum number of data windows"
120 markerNotFound  "The specified marker was not found."
121 diagnostic  "Diagnostic ."
122 channelNotFound  "The specified channel was not found."
123 exceedMaxMeasurements  "Exceeded the maximum number of allowed measurements."
124 parameterOutOfRange  "The specified value was out of range."
125 userRangeNotValid  "The currently selected user range is not valid."
126 referenceMarkerNotFound  "The reference marker is not active."
127 sweepSegmentNotFound  "The sweep segment was not found."
128 markerNotDelta  "The specified marker is not a delta marker."
129 printoutFailed  "Attempt to output to a printer failed."
130 memory_trace_not_compatible  "Memory not compatible. Trace Math not applied."
131 trace_math_reset
"Memory not compatible. Trace Math turned off."

132 hw_read_failed
"Hardware read failed."

133 hw_write_failed
"Hardware write failed."

134 dsp_active
"Failed because DSP was not halted."

135 secure_memory
"Attempt to access secure memory region."

136 snum_protected
"The serial number is protected."

137 snum_format_bad
"The serial number format is bad."

138 snum_already_set
"The serial number is already set."

139 hw_setting_failed
"Hardware setting failed."

140 cal_access_failed
"Calibration data access failed."

141 db_access_failed
"Database access failed."

142 memory_range_exceeded
"Command exceeds usable memory range."

143 lost_phase_lock
"Phase lock has been lost."

144 over_power
"Detected too much power at input."

145 ee_wrt_failed
"EEPROM write failed."

146 yig_cal_failed
"YTO calibration failed."

147 ramp_cal_failed
"Analog ramp calibration failed."

148 dspcom_bad
"DSP communication failed."

149 no_license_found
"Request failed. The required license was not found."

150 argLimited
"The argument was out of range

151 markerBWNotFound
"The Marker Bandwidth was not found."

153 peakNotFound
"The Peak was not found."

154 targetNotFound
"The Target search value was not found."

155 calNotImpl
"The Calibration feature requested is not implemented."

156 calClassNotValidForCalType
"SENS:CORR:CCH measurement selection set to none"
158 calNotValidForConfidenceCheck "Selected measurement does not have a calibration valid for Confidence Check"

159 invalidPort "Specified port is out of range"

160 invalidPortPath "ROUT:PATH:DEF:PORT x, y does not match measurement; setting to defaults"

161 ioInvalidWrite "Attempted I/O write while port set to read only."

162 ioInvalidRead "Attempted I/O read from write only port."

163 calsetNotFound "Requested Cal Set was not found in Cal Set Storage."

164 noCalSetSelected "There is no Cal Set currently selected for the specified channel."

165 cantDeleteCalSetInUse "Cannot delete a Cal Set while it is being used."

166 calsetStimChange "Channel stimulus settings changed to match selected Cal Set."

167 exceedMaxCalSets "Exceeded the maximum number of cal sets."

168 calCouldNotTurnOn "A valid calibration is required before correction can be turned on."

169 standardMeasurementRequired "The attempted operation can only be performed on a standard measurement type."

170 noDivisorBuffer "A valid divisor buffer is required before normalization can be turned on."

171 InvalidReceiverPowerCalParagraph "Receiver power cal requires the measurement to be of unratioed power."

172 ecalCouldNotConfigure "Could not configure the Electronic Calibration system. Check to see if the module is plugged into the proper connector."

173 measHasNoMemoryAlg "This measurement does not support memory operations"

174 measHasNoNormalizeAlg "This measurement does not support normalize operations."

175 userCharacterizationNotFound "User characterization was not found in the Electronic Calibration module."

176 measInvalidBufferSize "The data provided has an invalid number of points. It could not be stored."

Last Modified:

4-Aug-2009    Cosmetic mods
Technical Support

Click on the region of interest.

For more contact information, visit [http://www.agilent.com/find/contactus](http://www.agilent.com/find/contactus)

**Other Support Topics**

**United States:**
(tel) (+1) 800-829-4444
(alt) (+1) 303 662 3999
(fax) (+1) 888 900 8921

**Canada**
(tel) 1 877 894 4414
(fax) 1 (905) 206 4120

**Europe:**

**Austria**
(tel) 0820 87 44 11*
(fax) 0820 87 44 22

**Belgium**
(tel) (+32) (0)2 404 9340
(alt) (+32) (0)2 404 9000
(fax) (+32) (0)2 404 9395
Denmark
(tel) (+45) 7013 1515
(alt) (+45) 7013 7313
(fax) (+45) 7013 1555

Finland
(tel) 08 0052 4000
(alt) (+358) 10 855 2100
(fax) (+358) 92 536 0176

France
(tel) 0825 010 700*
(alt) (+33) (0)1 6453 5623
(fax) 0825 010 701*

Germany
(tel) 01805 24 6333*
(alt) 01805 24 6330*
(fax) 01805 24 6336*

Ireland
(tel) (+353) (0)1 890 924 204
(alt) (+353) (0)1 890 924 206
(fax) (+353) (0)1 890 924 024

Israel
(tel) (+972) 3 9288 500
(fax) (+972) 3 9288 501

Italy
(tel) (+39) (0)2 9260 8484
(fax) (+39) (0)2 9544 1175

Luxembourg
(tel) (+32) (0)2 404 9340
(alt) (+32) (0)2 404 9000
(fax) (+32) (0)2 404 9395

Netherlands
(tel) (+31) (0)20 547 2111
(alt) (+31) (0)20 547 2000
(fax) (+31) (0)20 547 2190

Russia
(tel) (+7) 095 797 3963
(alt) (+7) 095 797 3900
(fax) (+7) 095 797 3901

Spain
(tel) (+34) 91 631 3300
(alt) (+34) 91 631 3000
(fax) (+34) 91 631 3301

Sweden
(tel) 0200 88 22 55*
(alt) (+46) (0)8 5064 8686
(fax) 020 120 2266*

Switzerland (French)
(tel) 0800 80 5353 opt. 2*
(alt) (+33) (0)1 6453 5623
(fax) (+41) (0)22 567 5313

Switzerland (German)
(tel) 0800 80 5353 opt. 1*
(alt) (+49) (0)7031 464 6333
(fax) (+41) (0)1 272 7373

Switzerland (Italian)
(tel) 0800 80 5353 opt. 3*
(alt) (+39) (0)2 9260 8484
(fax) (+41) (0)22 567 5314

United Kingdom
(tel) (+44) (0)7004 666666
(alt) (+44) (0)7004 123123
(fax) (+44) (0)7004 444555

Japan:
(tel) 0120 421 345
Latin America:

Mexico
(tel) (+52) 55 5081 9469
(alt) 01800 5064 800
(fax) (+52) 55 5081 9467

Brazil
(tel) (+55) 11 4197 3600
(fax) (+55) 11 4197 3800

Australia:
((tel) 1800 629 485
(alt) 1800 143 243
(fax) 1800 142 134

New Zealand
(tel) 0 800 738 378
(fax) 64 4 495 8950

Asia Pacific:

China
(tel) 800 810 0189
(alt) (+86) 10800 650 0021
(fax) 800 820 2816

Hong Kong
(tel) 800 930 871
(alt) (+852) 3197 7889
(fax) (+852) 2 506 9233

India
(tel) 1600 112 929
(fax) 000800 650 1101
Malaysia
(tel) 1800 888 848
(alt) 1800 828 848
(fax) 1800 801 664

Singapore
(tel) 1800 375 8100
(fax) (+65) 6836 0252

South Korea
(tel) 080 769 0800
(alt) (+82) 2 2004 5004
(fax) (+82) 2 2004 5115

Taiwan
(tel) 0800 047 866
(alt) 00801 651 317
(fax) 0800 286 331

Thailand
(tel) 1800 226 008
(alt) (+66) 2 268 1345
(fax) (+66) 2 661 3714

Last Modified:
1-Dec-2008  Changed US number
3.8 GHz Frequency Adjustment

This routing adjusts the internal fixed-frequency YIG Oscillator to 3.8 GHz by changing a DAC value. This DAC value is stored in the analyzer's non-volatile memory. This adjustment is only needed on some PNA models. Typically, the oscillator can be set to within 12 kHz of 3.8GHz; it is not necessary for it to be exactly 3.8GHz.

Spectrum Analyzers Compatibility

This routine is compatible with Agilent 856x and 859x spectrum analyzers, and may also work on some other Agilent spectrum analyzers.

If no compatible analyzer is available, select "NONE" for the spectrum analyzer. You can then adjust the DAC manually by viewing the 3.8 GHz signal on another analyzer.

Procedure (For Compatible Spectrum Analyzers Only)

Note: The viewable 3.8 GHz signal level will be low; typically be around -70dBm. Do not use any attenuators in the adjustment, other than the default 10 dB attenuation used in most spectrum analyzers.

1. Connect spectrum analyzer input to the network analyzer's PORT 1 output.
2. Connect GPIB cable from analyzer to spectrum analyzer. Make sure no other controllers are active on the same connection.
3. Set the spectrum analyzer GPIB address to 18.
4. In the analyzer System menu, point to Service, Adjustments, and click 3.8 GHz Freq. Adjust.
5. Click Begin Adj, for the program to adjust the internal oscillator for minimal error and store the results. When the status area indicates the adjustment is complete, select Exit.

Procedure (For Non-Compatible Spectrum Analyzers Only)

Note: The viewable 3.8 GHz signal level will be low; typically be around -70dBm. Do not use any attenuators in the adjustment, other than the default 10 dB attenuation used in most spectrum analyzers.

1. Connect the spectrum analyzer input to the network analyzer's PORT 1 output.
2. Set the spectrum analyzer to the following settings:
   - Center frequency = 3.8 GHz
   - Span = 100 MHz
   - Bandwidth = 10 kHz
   - Scaling where a signal of -70 dBm will be clearly visible

3. In the analyzer **System** menu, point to **Service, Adjustments**, and click **3.8 GHz Freq. Adjust**.

4. Under **Spectrum Analyzer**, select **NONE** option for spectrum analyzer.

5. Click **Begin Adj**.

6. The application presets the DAC to an initial value equal to the current value stored. View the spectrum analyzer to see if the signal is above or below 3.800 GHz.
   - If the signal frequency is above 3.8 GHz, move the slider to adjust the DAC to a lower value (left).
   - If the frequency is below 3.8 GHz, move the slider to adjust the DAC to a higher value (right).

**Note:** The valid DAC values are from 0 to 4095. The oscillator will shift about 23 kHz per DAC value.

7. Set the DAC value to reach a frequency very close to 3.8 GHz. If you made large changes in DAC values, allow several seconds for thermal effects to stabilize.

8. Change the spectrum analyzer settings to better view the frequency signal:
   - Frequency span = 500 kHz
   - Bandwidth = 3 kHz

9. Change the DAC value to keep the signal centered at 3.8 GHz.

10. Once you have determined the correct DAC value, click **SAVE DAC** to permanently store that value into EEPROM. Click **Exit**.

**Note:** If large changes are made to the existing DAC value, then this test should be repeated again after 15-30 minutes. This allows the thermal effects to fully stabilize.
10 MHz Reference Frequency Adjustment

This routine adjusts the analyzer's internal time-base to exactly 10 MHz by changing a DAC value. This DAC value is stored in the analyzer's non-volatile memory. This routine should only be necessary in the following situations:

- The frequency reference assembly is replaced.
- The 10 MHz reference has drifted significantly from the factory adjusted value.

**WARNING:** The range of this adjustment is only about 20 Hz. It is highly recommended that a very accurate frequency standard be used to measure this 10 MHz signal.

Frequency Counter Compatibility

This procedure uses SCPI commands (over GPIB) to communicate with the frequency counter. It should work with the Agilent 5313xA, 5315xA, 53181A series of counters as well as the older 5350 series.

If no compatible counters are available, select the "Manual" mode of operation. If you do choose the manual mode, you must input the measured frequency manually.

Procedure for GPIB Counters Only

1. Connect the analyzer rear panel 10 MHz Reference output to the frequency counter.
2. Connect a GPIB cable from the analyzer to the counter. Make sure no other controllers are active on the same connection.
3. If applicable, connect the house frequency standard to the counter reference input.
4. Set the counter GPIB address to 03. Ensure that the counter is the only device at this address.
5. In the analyzer System menu, point to Service, Adjustments, and click 10 MHz Freq. Adjust.
6. Click Begin Adj. The application adjusts the internal reference for minimal error and stores the results.
7. Click Read Freq to trigger another reading of the 10 MHz signal.
8. Read the current DAC value stored in the analyzer's non-volatile memory (value = 0 - 4095).
9. When the status area indicates the adjustment is complete, click Exit.

Procedure for Non-GPIB Counters

1. Connect the counter input to the rear panel 10 MHz Reference Output.
2. Set the counter to at least 1 Hz resolution.
3. If applicable, connect the house-frequency standard to the counter reference input.
4. In the analyzer System menu, point to Service, Adjustments and click 10 MHz Freq. Adjust.
5. Under **Frequency Counter**, select **Manual**.

- Adjust the slider bar **arrows** until the frequency counter reads 10.0 MHz at your desired level of accuracy.
- Click **Exit** to save the results.

**Note:** If the counter is misreading the frequency, it may be necessary to attenuate the input, or set the input impedance to 50 ohms, or both.

---

Last Modified:

9-May-2008  Updated with new UI
Display Test

The PNA screen should be bright with all annotations and text readable. The display test allows you to check for non-functioning pixels and other problems.

**Note** If the display is dim or dark, refer to “Troubleshooting LCD Display Problems” in the PNA Service Guide.

What Is a Damaged Pixel?

A pixel is a picture element that combines to create the image on the display. They are about the size of a small pin point. Damaged pixels can be either “stuck on” or “dark.”

- **Stuck on pixel** - red, green, or blue; always displayed regardless of the display setting. It will be visible on a dark background.
- **Dark pixel** - always dark; displayed against a background of its own color.

How to Run the Display Test

On the **System** menu, point to **Service**, and then click **Display Test**.

A multi-color screen is displayed. Be prepared to look for the symptoms described below. Click the Start Test button. To continue to the next test, click the moving Next Test button. The button moves to allow you to see all of the display. After the test is completed, the display defaults to the network analyzer screen.

How to Identify a Faulty Display

One or more of the following indicate a bad display:

- Complete row or column of “stuck on” or “dark” pixels
- More than six “stuck on” pixels (but not more than three green)
- More than twelve “dark” pixels (but not more than seven of the same color)
- Two or more consecutive “stuck on” pixels or three or more consecutive “dark” pixels (but no more than one set of two consecutive dark pixels)
- “Stuck on or “dark” pixels less than 6.5 mm apart (excluding consecutive pixels)

If any of these symptoms occur, your display is considered faulty. See the Service Guide for your PNA model.
**LO Power Adjustment**

This procedure adjusts the receiver's LO input power to a specific level by changing DAC values. These DAC values are then stored in the analyzer's non-volatile memory. The procedure will vary depending upon the model number. This adjustment is only applicable to some PNA models.

**Power Meter Compatibility**

This routine is only compatible with the Agilent EPM series of power meters. Different sensors may be used. For 9 GHz analyzers and below, an 8482 or E4412A sensor can be used. For the higher frequency units (20 GHz or above), a sensor must be able to measure a maximum of 20 GHz. At no time during this test will a frequency higher than 20 GHz be measured, even if the PNA has a maximum frequency of 50 GHz.

If the older HP 84xx series of sensors are used, the correct calibration data should be entered into the appropriate cal table of the EPM series power meter, although for this adjustment, high accuracy is not required. Inaccuracies in the order of several tenths of a dB are acceptable.

**Procedure**

1. Allow the analyzer and power meter to warm up for 30 minutes.
2. Manually zero and calibrate the power sensor. (This allows you to skip this step later)
3. Connect a GPIB cable from the analyzer to the power meter. Make sure no other controllers are active on the same connection.
4. Set the power meter GPIB address to 13. (others can also be used; 13 is the default)
5. Remove the outer cover on the analyzer.
6. In the PNA System menu point to Service, then Adjustments, and click LO Power Adjust.
7. Connect the power sensor to the LO output, using adapters if needed.. The LO output location varies with model number. Click on the LO Power Adjust Setup menu selection to see a diagram of the exact location.
8. **For 9 GHz units and below:**
   Click Begin Adj to start the LO power cal routine. The routine adjusts the power level for each band (1 through 3) to fall within certain bounds. If any changes are made, it automatically stores them.
   - **For 20GHz units and above:**
     If using an 84xx power sensor, click Configure and select the proper sensor model number. Click Close.
     Click Calibrate to begin the adjustment. The entire calibration process takes about 5 minutes. Once completed, you can verify the current calibration accuracy by clicking Verify Cal.
     **Note:** Correction constants are defaulted at the beginning of calibration. Once the calibration process has started, it must be completed in order to regenerate proper data.
9. Click Read DAC to view the current DAC values (0-4095) stored in the PNA non-volatile memory for each band (0-7).
10. When the message/status area indicates the adjustment is complete, click Exit.

11. Reconnect the semi-rigid cable and replace the covers.
Offset LO Power Adjustment

Note: This adjustment is only performed on PNAs with Frequency Offset Mode (option 080), and only on certain models.

The Offset LO Adjustment sets the LO power for the offset mixer to a consistent value across all bands. It requires access to the internal components of the PNA so that the power sensor can be connected to the LO output. Because the LO frequency does not exceed 3 GHz, almost any power sensor can be used. The adjustment is relatively simple and only takes a couple of minutes.

When to perform
This adjustment should be performed when any of the following occur:

- the A13 Frequency Offset Receiver is significantly modified or replaced
- the A9 Synthesizer is replaced (that is where the correction data resides)

How to perform Offset LO Power adjustment

1. To start the Offset LO adjustment, click System, point to Service, Adjustments, then click Offset LO Adjust.

2. You will be prompted to zero and cal the power sensor. (You can do this before beginning.)

3. Connect the sensor to J3 of the A13 LO output by removing the existing cable (or simply disconnecting one end) as shown in the Set-up diagram in the program.

4. Connect the power meter to the PNA using a GPIB cable. Make sure the GPIB address shown in the program matches the actual power meter address (default is 13.)

5. Click Adjust to begin the test and follow the instructions.

The program automatically adjusts all bands; no user input is needed. The program repeats several times as this is an iterative process. The progress of the adjustment is shown on the screen.

The Configure menu selection is for factory personnel ONLY.

Once completed, to verify the actual results, click Verify.

Upon exiting, the PNA application will restart; this takes several seconds.
Overview

The Operator's Check should be performed when you first receive your PNA, and any time you wish to have confidence that the PNA is working properly.

Notes

- The Operator's Check does not verify performance to specifications. To verify PNA performance to specifications, run System Verification.
- Allow the PNA to warm up for 90 minutes before considering a failed test to be valid.
- The Operator’s Check can NOT be run with a Multiport test set enabled. However, you can run a performance check as described in the Test Set User's Guide. See the N44xx User's Guide.

The Pass/Fail criteria used in the Operator's Check identifies obvious failures in the following portions of the PNA hardware:

- Repeatability of the RF switch in the test set
- Attenuation ranges of the test port attenuators (if installed)
- Calibration of the receivers
- Frequency response of the receivers
- Phase lock and leveling
- Noise floor and trace noise
## How to Run the Operator's Check

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>For PNA-X and ‘C’ models</strong></td>
<td><strong>Programming Commands</strong></td>
</tr>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then <strong>[Service]</strong></td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then <strong>[Operator's Check]</strong></td>
<td>3. then <strong>Service</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>Operator's Check</strong></td>
</tr>
</tbody>
</table>

1. Connect one or more standards (see [Configure](#)).
2. Click **Begin** and **Continue** (if necessary) until "Operator's Check is complete!" appears.

![Operator's Check](image)

This dialog box will look slightly different, depending on PNA model number and installed options. Some of the tests are not run if the appropriate option is not installed.

To learn about how each test is performed, click one of the tests on the right of the dialog. For example, the following information dialog is launched when **Leveling** is clicked:
Operators Check dialog box help

**Note:** It is normal for a momentary un leveled condition to appear during portions of the Operators Check.

**Configure**

**Prompt for attachment of Short / Open**  If you do not have enough shorts or opens for all test ports, you will be prompted to move the standard to the next test port. Connect either a short or open to port 1, then click Begin.

**Shorts / Opens are attached to all ports**  Connect either a short or open for each test port, then click Begin. All ports are tested without interruption. You can mix shorts and opens on the test ports.

**PNA**  Shows information about the PNA that is being tested.

**Legend**  Shows the status icons used in the Operator's Check and their meaning. **Pending Pass** means that a portion of the testing has been completed successfully.

**Results**  Shows the current status of each test. Click on the test name to learn how that test is performed. This may help in troubleshooting failed tests. If any tests Fail, refer to Chapter 3 of the PNA service guide.

**Begin**  Starts the Operator's Check.

**View Results**  Shows all results in text format. Failed items are preceded by ==>>>.

This text file can be printed or saved with a unique file name to compare results with previous or subsequent testing.

**Exit**  Ends the program and closes the window.

---

Last Modified:

- 4-Sep-2008  Removed legacy content
- 17-Apr-2008  Added test description note
- 5-Feb-2008  Added multiport test set note
Option H11 Test Verification

The PNA Option H11 Test / Verification utility verifies the PNA's option H11 functionality. The associated **H11 inputs and outputs** are tested and the results are compared against **expected levels at the factory**. These tests are used in conjunction with the **Operator's Check** and **System Verification** programs to check PNA functionality.

### How to Run Option H11 Verification

**Using front-panel HARDKEY [softkey] buttons**

1. Press **SYSTEM**
2. then **[Service]**
3. then **[Rear Panel/H11 Test or H11 Verification]**

**Using a mouse with PNA Menus**

1. Click **System**
2. then **Service**
3. then **Rear Panel/H11 Test or H11 Verification**

---

**Option H11 Verification** dialog box help

The Option H11 Verification software leads you through each of the tests listed on the Configure Tab.

**Important:** Required Configuration of the PNA being tested:

- All connections to external devices and test sets must be removed from the Option H11 connectors and Test Set I/O connector on the PNA rear panel.
- On the **Millimeter Module Configuration** dialog box, make the following settings:
  - PNA Model E8361A - Select **Agilent N5250A**, and check **Use Standard PNA operation when N5260A is NOT connected**.
  - All other PNA models - Select **Standard PNA**.
Run Tab
The list of test equipment is the complete list that is required to run ALL of the tests (recommended).

Start  Click to run the tests that are selected on the Configure Tab.

Exit  Exits Option H11 Verification.

Configure Tab
All of the tests are selected by default. Tests can be run separately for troubleshooting purposes by checking or clearing the boxes.

Power Tests
Output power is measured at each of the selected RF and LO Test Set connectors on the rear panel. The results are displayed against limit lines that represent the expected levels at the factory.

Equipment Required:

- PNA supported power meter and 26.5 GHz power sensor (Recommended: Agilent E4418/19 Power Meter and E4413A Power Sensor). Set GPIB address = 13.
- GPIB and Sensor cables.
- On E8361A ONLY, a device is required to simulate an N5260A Test Set is connected to the Test Set I/O. All other PNA models do NOT require this device for this test.
  - Agilent Service personnel: Use the E8361-60063 Test Set I/O Tester.
  - All others: On a commercially available DB-25 male connector, connect a jumper between pins 12 and 22. See the Test Set I/O connector diagram. Insert the male connector into the PNA rear panel Test set I/O connector.

External IF Input and Crosstalk Tests
A 8.33 MHz signal is injected into each of the selected 8.33 MHz IF IN connectors on the rear panel. The signal is measured at each of the selected receivers. The results are displayed against limit lines that represent the expected levels at the factory.

Equipment Required:

- 33120A Function Generator. Set GPIB address = 10.
- 10 dB Attenuator (Agilent 8493B or equivalent) and BNC adapters.
- GPIB and BNC cables

External Pulse Input Test
A pulse train is injected into each of the selected Pulse IN connectors on the rear panel. The signal is measured at each of the selected receivers. The results are displayed against limit lines that represent the expected levels at the factory.

Note: A noisy pulse generator can cause a false failure of the "1% Duty Cycle Pulse Input" portion of the
External Pulse Input Test.

Equipment Required:

- 8110A or 81110A Pulse Generator. Set GPIB address = 11.
- GPIB and BNC cables

**Test Set I/O Connector Test**

Tests the ability of the PNA to detect and control an external test set through the PNA rear panel TEST SET I/O connector.

Equipment Required:

- Agilent Service personnel: Use the E8361-60063 Test Set I/O Tester.
- All others: Use a DVM to measure the TTL voltage level on the following pin numbers using pin 1 as ground:
  - Pins: 3, 4, 5, 6, 8, 9, 10, 11, 17, 19, 20, 21, 22, 23, 24
  - See [Test Set I/O connector diagram](#) to determine the pin number location.
  - The software toggles all pins HIGH, then LOW.
  - TTL HIGH should read approximately +5.0 volts.
  - TTL LOW should read approximately +0.1 volts.

**Help Tab**

Displays the Option H11 Verification program revision number and information about the PNA being tested.

**Advanced Help** Click to display this help topic.

**View Results** Appears after the test is run. Click to show all results in text format. Failed items are preceded by `>>>>> Fail`. This text file can be printed or saved with a unique file name to compare results with previous or subsequent testing.
Phase-Lock IF Gain Adjustment for E836x models only!

The E836x A/B PNA models have a variable gain control for the phase-lock loop IF signal. By dynamically changing the gain as a function of frequency and power, the phase-lock signal amplitude can be adjusted to a constant level for the entire operating range of the instrument. This constant level is important for phase-lock acquisition and stability.

**When to perform**

Phase-Lock IF Gain Adjustment should be performed when any of the following occur:

- A source calibration
- An assembly in the reference receiver path (R1,R2) is replaced.
- The Test Set Motherboard is replaced
- The Phase Lock board is replaced
- **Phase Lock Lost** error message appears after replacing a source or receiver assembly
- The external R path has changed. For example, when a multiport test set with R channel path has been added or removed.

**How to perform Phase-Lock IF Gain adjustment**

Ensure the Reference Channel paths are properly configured and the connections are properly torqued.

1. From the **System** menu, click **Service**, then **Adjustments**, then **IF Gain Adjustment**.
2. Under Select Specials, select **None**.
3. No connections to the test ports are required.
4. Click **Begin Adj.** The adjustment takes about a minute to complete.
5. The advanced screen is for factory personnel only.

---

IF Gain Adjustment for N524x models only!

The N524x PNA models have a variable gain control for the loop IF signal. By dynamically changing the gain as a function of frequency and power, the phase-lock signal amplitude can be adjusted to a constant level for the entire operating range of the instrument. This constant level is important for phase-lock acquisition and stability.

**When to perform**

IF Gain Adjustment should be performed when any of the following occur:
An assembly in the reference receiver path (R1,R2) is replaced.

The Test Set Motherboard is replaced

The IF Mux assembly or SPAM assembly is replaced

**How to perform IF Gain adjustment**

Ensure the Reference Channel paths are properly configured and the connections are properly torqued.

1. From the **System** menu, click **Service**, then **Adjustments**, then **IF Gain Adjustment**.

2. An appropriate power meter and sensor will be required. The sensor must cover the upper frequency range of the PNA. A good quality cable will also be needed. Adapters can be used as needed.

3. Connect the power meter to the GPIB port labeled System Controller.

4. Click **Begin Adj.**

5. Set the GPIB address in the program accordingly and select the sensor being used.

6. Follow the instructions displayed in the program.

The adjustment takes about 5 minutes to complete for a 26.5 GHz PNA. Higher frequency units may take longer.

The Default menu selection is for factory personnel only. This will preset all values to default levels for troubleshooting purposes only. If this is selected, a full IF gain adjustment will need to be performed.

**Last modified:**

3/7/07  Added N5242x information.
System Verification

The performance of the network analyzer is specified in two ways: system specifications, and instrument specifications. It is the end user’s responsibility to determine which set of specifications is applicable to their use of the PNA.

A network analyzer measurement “system” includes the analyzer, calibration kit, test cables, and any necessary adapters. The system verification software in the PNA is used to verify the system’s conformance to the “system” specifications. A “pass” result demonstrates that the analyzer, test cables, and adapters, perform correctly as a system. It DOES NOT demonstrate that any one component performs according to its individual specifications. A change to any part of this measurement system requires a re-verification of the system.

Instrument specifications specify the network analyzer’s uncorrected measurement port characteristics and its output and input behavior. The PNA performance tests are used to verify the analyzer’s conformance to “instrument” specifications.

The system verification utility verifies the PNA system specifications by automatically measuring the magnitude and phase for all four S-parameters for each verification device, and comparing the values against the following:

- Factory measured data from files on the verification disk
- Limit lines based on the measurement uncertainty

System Verification requires the use of a calibration kit and verification kit which has been certified within the past 12 months by Agilent. System Verification can NOT be used to perform this kit certification.

Operator’s Check should also be performed to verify the basic operation of the PNA.

Equipment Used in the System Verification

Precautions for Handling Airlines

Flow Diagram of Procedure

Procedure for System Verification

If the System Fails the Verification Test

Interpreting the Verification Results

Notes

- Although the performance for all S-parameters is measured, the S-parameter phase uncertainties are less important for verifying system performance. Therefore, the limit lines will not appear on the printouts.

- System Verification can NOT be run with a Multiport test set enabled. However, you can run a performance check as described in the Test Set User's Guide. See the N44xx User's Guide.

Equipment Used in the System Verification

For PNA models:
### E8356A, E8357A, E8358A
#### N3381A, N3382A, N3383A
#### E8801A, E8802A, E8803A
(Type-N test ports)

<table>
<thead>
<tr>
<th>Equipment Type</th>
<th>Type-N</th>
<th>3.5 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calibration kit</td>
<td>85032F</td>
<td>85033E</td>
</tr>
<tr>
<td>or</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ECAL Module</td>
<td>85092B</td>
<td>85093B</td>
</tr>
<tr>
<td>Verification kit</td>
<td>85055A</td>
<td>85053B</td>
</tr>
<tr>
<td>RF Cable</td>
<td>N6314A</td>
<td></td>
</tr>
</tbody>
</table>

See Cable substitution

### E8362A/B/C
#### N5230A (20 GHz)
(3.5 mm test ports)
#### N5241A and N5242A

<table>
<thead>
<tr>
<th>Equipment Type</th>
<th>3.5 mm</th>
<th>Type-N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calibration kit</td>
<td>85052B/C/D</td>
<td>85054B/D</td>
</tr>
<tr>
<td>or</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ECAL Module</td>
<td>N4691A</td>
<td>N4690A</td>
</tr>
<tr>
<td>Verification kit</td>
<td>85053B</td>
<td>85055A</td>
</tr>
<tr>
<td>RF Cable(s)</td>
<td>Single: 85131C/E</td>
<td>Single: 85132C/E</td>
</tr>
<tr>
<td></td>
<td>Pair: 85131D/F</td>
<td>Pair: 85132D/F</td>
</tr>
<tr>
<td>Adapters</td>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

Single: 85130C and one 7mm-to-Type-N from 85054B cal kit
Pair: Two 7mm-to-Type-N from 85054B cal kit

### E8363A/B, E8364A/B/C
#### N5230A (40 or 50 GHz)
(2.4 mm test ports)
<table>
<thead>
<tr>
<th>Equipment Type</th>
<th>2.4 mm</th>
<th>3.5 mm</th>
<th>Type-N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calibration kit or ECAL Module</td>
<td>85056A/D</td>
<td>85052B/C/D</td>
<td>85054B/D</td>
</tr>
<tr>
<td>Verification kit</td>
<td>85057B</td>
<td>85053B</td>
<td>85055A</td>
</tr>
<tr>
<td>RF Cable(s)</td>
<td>Single: 85133C/E Pair: 85133D/F</td>
<td>Single: 85134C/E Pair: 85134D/F</td>
<td>Single: 85135C/E Pair: 85135D/F</td>
</tr>
<tr>
<td>Adapters</td>
<td>None</td>
<td>Single: 85130F Pair: None</td>
<td>Single: 85130E and two 7mm-to-Type-N from 85054B cal kit Pair: Two 7mm-to-Type-N from 85054B cal kit</td>
</tr>
</tbody>
</table>

**E8361A/C**
(1.85 mm test ports)

<table>
<thead>
<tr>
<th>Equipment Type</th>
<th>1.85 mm</th>
<th>2.4 mm</th>
<th>3.5 mm</th>
<th>Type-N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calibration kit or ECAL Module</td>
<td>85058B</td>
<td>See 2.4 mm test port table above</td>
<td>See 2.4 mm test port table above</td>
<td></td>
</tr>
<tr>
<td>Verification kit</td>
<td>85058V</td>
<td>See 2.4 mm test port table above</td>
<td>See 2.4 mm test port table above</td>
<td></td>
</tr>
<tr>
<td>RF Cable(s)</td>
<td>Single: N4697E Pair: N4697F</td>
<td>See 2.4 mm test port table above</td>
<td>See 2.4 mm test port table above</td>
<td></td>
</tr>
<tr>
<td>Adapters</td>
<td>None</td>
<td>See 2.4 mm test port table above</td>
<td>See 2.4 mm test port table above</td>
<td></td>
</tr>
</tbody>
</table>

**Cable Substitution**

The test port cables specified for the PNA have been characterized for connector repeatability, magnitude and phase stability with flexing, return loss, insertion loss, and aging rate. Since test port cable performance is a significant contributor to the system performance, cables of lower performance will increase the uncertainty of your measurement. It is highly recommended that the test port cables be regularly tested.

If the system verification is performed with a non-Agilent cable, ensure that the cable meets or exceeds the operation of the specified cable. Refer to the cable User’s Guide for specifications.

**Cable Flex Factor**

Flex Factor determines how much of the cable phase uncertainty to include in determining the limit lines.

- Set to 0% (zero) if the cables are held down in a fixture and are not allowed to move during the calibration and verification.
• Set to 100% if the cables are allowed to move a lot.

**Calibration Kit Substitution**
Non-Agilent calibration kits are not recommended nor supported.

**Precautions for Handling Airlines**
When you are using the airlines in the verification kit, observe the following practices to ensure good measurement techniques.

• Be very careful not to drop the airline's center or outer conductor. Damage will result if these devices are dropped.
• Use proper Electro-Static Discharge (ESD) procedures.
• Clean your hands or wear gloves as skin oils will cause a change in electrical performance.
Flow Diagram of Procedure

The operational flow of the software is depicted by the flowchart shown below.
Procedure for System Verification

1. If you want printed test outputs, connect a printer to the analyzer. Let the analyzer warm up for at least 30 minutes.

2. Insert the PNA verification kit floppy disk into the analyzer disk drive.
3. On the **System** (or **Utility**) menu, point to **Service**, and click **System Verification**. The System Verification window similar to this will be displayed.

4. In the **Calibration Kit** box, select the calibration kit or ECal module that is being used. The corresponding verification kit to use appears in the **Verification Kit** box.

5. Under **Printer Output** click on any of the following options.

   - **Print Tabular Data**: Prints the verification data in tabular form which includes measured data and uncertainty limits. Refer to a tabular data example, later in this topic.
   - **Print Graphs**: Prints the verification data in graphical form. The graphic form includes the measured data trace, factory supplied data trace and uncertainty limits. Refer to a plot data example, later in this topic.
   - **File Tabular Data**: Writes the verification data in tabular form to a text file in the C:/Program Files/Agilent/Network Analyzer/Documents/ directory.
   - **File Graphs**: Saves a screen image in .PNG format in the C:/Program Files/Agilent/Network Analyzer/Documents/ directory.

**Note**: If you want printed output, it is assumed you have already installed the Windows driver for your particular printer, and have tested that you can print to the printer from the network analyzer. This software is designed to print to whichever printer is currently set as the Default printer (see Printers in the Windows Control Panel).

6. To modify the number of ports to be verified, to change the number of devices to measure, or to use a previously stored verification calibration, click on the **Configure** tab and make the desired selections.

   - For the system verification to be truly adequate, the software must measure all devices in the kit with a recent calibration applied. Removing and reattaching any test port cables or adapters invalidates all
previous calibrations.

7. Click **Run**.

8. Follow the instructions on the analyzer for performing the system verification, inserting the verification devices as prompted.

**Note for 3 Port PNA:**
The System Verification Procedure is **repeated three times**. The first time, **Ports 1 and 2** are measured as a pair; then **Ports 1 and 3** are measured; and lastly, **Ports 2 and 3** are measured.

**Note for 4 Port PNA:**
The System Verification Procedure is **repeated two times**. The first time, **Ports 1 and 2** are measured as a pair, then **Ports 3 and 4** are measured.

**Step-by-Step Process Description**

1. Depending upon the selected choice in the Calibration submenu of the Configure menu, the user is either prompted to recall a previous calibrated instrument state, or is guided through a full 2-port calibration using the selected calibration kit. For ECal, the ECal module is connected just once; a standby message is posted while the software is performing the calibration.

2. The user is prompted to connect the first verification device.

3. The software reads the factory measured data for that device and uncertainty values for that data (CITIfiles) from the floppy disk supplied with the verification kit.

4. The software sends the factory measured data, calibration kit and instrument state information to the uncertainty calculator DLL, which generates uncertainty values specific to the PNA.

5. The analyzer first sets up for magnitude measurements of all four S-parameters, each parameter in a separate window (lin mag for S11 and S22, log mag for S21 and S12). Each of the factory measured $S$-parameters are fed to the appropriate window as a memory trace. Limit line offsets are calculated as the sum of the factory measured data uncertainties and PNA uncertainties reported by the DLL. Upper and lower limits are displayed (factory measured data + uncertainty sum, factory measured data - uncertainty sum). The PNA takes a sweep, limit test is turned on and PASS/FALSE status is reported in each of the four windows.

6. The user clicks a button when ready to view phase measurements. The four windows get updated for phase format, phase memory traces, phase limits and PASS/FALSE result.

7. If the limit test of any of the four S-parameters (magnitude or phase) indicates a FAIL status, the software suggests troubleshooting tips and asks if the user would like to repeat measurement of that device or proceed to the next device. If proceeding to the next device, the factory measured data and uncertainties for the next device are read from floppy, the uncertainty DLL gets called with this next set of factory measured data, and the four measurement windows get updated for magnitude measurement of the next device.

8. The software follows this same process until all selected devices have been measured, at which point a summary window is displayed containing the set of PASS/FALSE results for all four parameters of each device.

**If the System Fails the Verification Test**
IMPORTANT: Inspect all connections. Do not remove the cable from the analyzer test port. This will invalidate the calibration that you have done earlier.

1. Repeat this verification test. Make good connections with correct torque specifications for each verification device.
2. Disconnect, clean and reconnect the device that failed the verification test. Then measure the device again.
3. If the analyzer still fails the test, check the measurement calibration by viewing the error terms as described in "Front Panel Access to Error Terms" on page 4-7 of the Service Guide.
4. Refer to the graphic below, for additional troubleshooting steps.

**Verification Fails Flowchart**

Interpreting the Verification Results

The graphic below shows an example of typical verification results with **Tabular Data** selected in the **Printer Output** area of the System Verification window. A graphic later in this topic shows an example of typical verification results with **Measurement Plots** selected in the **Printer Output** area of the System Verification windows. These printouts include a comparison of the data from your measurement results with the traceable data and corresponding uncertainty specifications. Use these printouts to determine whether your measured data falls within the total uncertainty limits at all frequencies.

The tabular data consists of:
- Frequency of the data points (in MHz).
- Lower limit line as defined by the total system uncertainty specification.
- Results of the measurement.
- Upper limit line as defined by the total system uncertainty specification.
- Test status (PASS or FAIL) of that measurement point.

**Printout of Tabular Verification Results**

The printed graphical results show:

- Upper limit points as defined by the total system uncertainty specifications.
- Lower limit points as defined by the total system uncertainty specifications.
- Data measured at the factory.
- Results of measurements.
- Measurement parameter names and formats (Lin Mag or Log Mag).
- Serial number of device (00810).
- Device being measured (Sys Ver 20 dB attenuator).

**Printout of Graphical Verification Results**
Last Modified:

- 6-Apr-2009   Updated for N5241A
- 8-Apr-2008   Updated for 'C' Models
- 11-Feb-2008  Added note about multiport
- 15-Jan-2008  Added Flex Factor and image
Source Adjustment

Source Adjustment is a **SERVICE** Routine which should be performed when a component in the source chain is replaced, or when the PNA fails an annual calibration. It adjusts the PNA source power for flatness across its full frequency range.

This topic does **NOT** discuss **Source Power Calibration**, which calibrates a PNA source over the current measurement range.

Required Equipment

Preferred Power Meter: E4419B
Alternate Power Meters: E4419A or EPM-442A

**Note:** The power sensor depends on the PNA frequency range. Depending on the PNA model, two power sensors may be required to test the full frequency range. The PNA front panel connector type will determine the cable used and if an adapter is required with the power sensor(s).

<table>
<thead>
<tr>
<th>PNA Model</th>
<th>Power Sensor(s)</th>
<th>Cable</th>
</tr>
</thead>
<tbody>
<tr>
<td>E8356A</td>
<td>8482A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8357A</td>
<td>8482A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8358A</td>
<td>8482A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8801A</td>
<td>8482A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8802A</td>
<td>8482A and E4412A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8803A</td>
<td>8482A and E4412A</td>
<td>N6314A</td>
</tr>
<tr>
<td>N3381A</td>
<td>8482A</td>
<td>N6314A</td>
</tr>
<tr>
<td>N3382A/N3383A</td>
<td>8482A and E4412A</td>
<td>N6314A</td>
</tr>
<tr>
<td>E8361A</td>
<td>8487A (use with adapter 11900B) and V8486A (use with adapter V281B) <strong>See E8361A procedure below</strong></td>
<td>N4697-60001</td>
</tr>
<tr>
<td>E8362A/B/C</td>
<td>E4413A</td>
<td>85131E</td>
</tr>
<tr>
<td>E8363A/B/C</td>
<td>8487A (use with adapter 11900B)</td>
<td>85133E</td>
</tr>
<tr>
<td>N5241A</td>
<td>E4413A</td>
<td>85131E</td>
</tr>
</tbody>
</table>

See PNA Accessories
Procedure

1. Refer to your power meter documentation to ensure the proper calibration factors for the power sensor have been entered into the table on the power meter.

2. Connect a GPIB cable between the power meter and network analyzer (use the System Controller GPIB port if applicable.)

3. Ensure the power sensor(s) are connected to the power meter.

4. In the analyzer System menu, point to Service, Adjustments, and click Source Adjustment.

5. There are 3 different versions of the Source Calibration software; all are slightly different. All have a button that is labeled "Calibrate" or "Adjust". This is the button that will begin the calibration process. Some versions will also have a button labeled "Verify" that will test the source calibration without making any changes. Other selections are for factory personnel use only.

6. Once begun, you must enter the power meter and sensor information. The software will verify the power meter and sensor. You are then prompted to connect the sensor(s) and cable as needed.

Connecting sensors to the PNA

Additional Information

All ports are tested on all PNAs. Source calibration takes approximately 10 to 45 minutes to complete depending on the frequency range and model number of the PNA. The E8361 models may take up to 90 minutes.

Troubleshooting

In the event there is a problem with Source Adjustment, please refer to the "Troubleshooting" chapter in the PNA.
E8361 Procedure

Source and Receiver adjustment requires the power meter to measure the source power over the full range of each of the PNA internal bands. Because the 8487A can not measure accurately above 50 GHz, it can only be used up to the next highest band switch frequency at 46.2 GHz. The V8486A sensor and V281B adapter are used from 46.2 GHz to 67 GHz.

For highest accuracy, the V8486A and V281B should be sent to Agilent for a custom calibration from 45 GHz to 70 GHz.

For the next highest accuracy level, the following procedure shows how to measure correction factors yourself from 46 to 50 GHz. This procedure assumes you have already loaded correction factors for both sensors into the power meter.

1. On your power meter, add 46 and 48 GHz to the Cal Factor Table.
2. Preset the PNA
3. Tune the PNA to 46 GHz (CW frequency)
4. Using the 8487A, measure power at port 1. Record this value.
5. Tune the PNA to 48 GHz (CW frequency)
6. Using the 8487A, measure power at port 1. Record this value.
7. Connect the V8486A, V281A, and 1.85 f-f adapter to the power meter.
8. Tune the PNA to 46 GHz (CW frequency)
9. Adjust the cal factor table 46 GHz setting until the power meter reading matches the power readings from step 4.
10. Tune the PNA to 48 GHz (CW frequency)
11. Adjust the cal factor table 48 GHz setting until the power meter reading matches the power readings from step 8.

Last Modified:

6-Apr-2009   Updated models
10-Mar-2009   Changed to Source Adjustment
**Receiver Calibration**

Receiver calibration adjusts the network analyzer receivers for a flat response across its full frequency range. This adjustment is for service only; not for measurement calibration.

**Required Equipment**

Preferred Power Meter: E4419B  
Alternate Power Meters: E4419A or EPM-442A

**Note:** The power sensor depends on the PNA frequency range. Depending on the PNA model, two power sensors may be required to test the full frequency range. The PNA front panel connector type will determine the cable used and if an adapter is required with the power sensor(s).

See [PNA Accessories](#)

**Procedure**

1. Refer to your power meter documentation to ensure the proper calibration factors for the power sensor have been entered into a table on the power meter.

2. Connect a GPIB cable between the power meter and network analyzer.

3. Ensure the power sensor(s) are connected to the power meter.

4. In the analyzer **System** menu, point to **Service, Adjustments**, and click **Receiver Calibration**.

5. The software presents you with two choices:

   a. Click **Inspect Flatness** to observe flatness of receiver response versus frequency. Although there is no explicit specification for receiver flatness, Receiver Calibration should improve Transmission and Reflection Tracking error terms which are specified.

   b. Click **Calibrate** to begin the receiver calibration process. The software prompts you to connect the sensor(s), cable and adapter as needed (see the following graphics).

   **Connecting sensor(s) to the PNA**
Connecting adapter and cable between sensor and PNA

Through connection using the specified cable
**Additional Information**

Receiver Calibration tests all PNA receivers, taking approximately 15 and 45 minutes. Length is dependent on frequency range and number of ports.

**Troubleshooting**

In the event there is a problem with Receiver Calibration, please refer to the "Troubleshooting" chapter in your PNA Service Guide.

---

Last Modified:

12-Jul-2007   Removed outdated table of supported power meters
## Receiver Display

- **The Receiver Display as a Troubleshooting Tool**
- **How to start the Receiver Display**

### Other Support Topics

### The Receiver Display as a Troubleshooting Tool

The Receiver Display is a Troubleshooting Tool. It enables the analyzer to isolate faulty functional groups within its own Measurement System. Traces for each Receiver are Displayed in individual windows. Identifying discrepancies of the traces in these windows can help isolate the faulty assembly.

For a thorough description of Receiver Display and the troubleshooting steps see Chapter 3 of the PNA Service Guide. You can download the Service Guide for your PNA model from our website: [http://na.tm.agilent.com/pna/](http://na.tm.agilent.com/pna/)

### How to Start the Receiver Display

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <code>[SYSTEM]</code></td>
<td>1. Click <code>[Utilities]</code></td>
</tr>
<tr>
<td>2. then <code>[Service]</code></td>
<td>2. then <code>[System]</code></td>
</tr>
<tr>
<td>3. then <code>[Utilities]</code></td>
<td>3. then <code>[Service]</code></td>
</tr>
<tr>
<td>4. then <code>[Receiver Display]</code></td>
<td>4. then <code>[Utilities]</code></td>
</tr>
<tr>
<td></td>
<td>5. then <code>[Receiver Display]</code></td>
</tr>
</tbody>
</table>

[Programming Commands]
IF Path Configuration Settings

This dialog is used to set many IF receiver settings for PNA-X and N522x models. See the IF Path settings for "C" models.

In this topic:

- PNA-X and N522x IF Frequencies
- How to Start the IF Path Configuration dialog
- IF (Receiver) Path Configuration dialog box
- Expanded Block Diagram and Descriptions
- IF Path Configuration using COM and SCPI

Other IF Access Topics

PNA-X and N522x Auto IF Frequencies

Wideband/Normal IF path:

**Note:** For the following discussion, **RF** = Receiver Frequency

**Wideband/Normal IF path:**

**With DSP Version 4:**

- RF < 53 MHz: IF = 2.535211 MHz  \[3 \times (60e6 / 71)\]
- RF >= 53 MHz: IF = 7.605634 MHz  \[9 \times (60e6 / 71)\]

**With DSP Version 5,** the IF frequency is dependent on the RF AND the current IFBW setting:

- All RF; IF Bandwidth >= 1MHz: (All Models)
<table>
<thead>
<tr>
<th>IFBW Setting</th>
<th>IF Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 MHz</td>
<td>7.692 MHz</td>
</tr>
<tr>
<td>1.5 MHz</td>
<td>7.368 MHz</td>
</tr>
<tr>
<td>2 MHz</td>
<td>8.450 MHz</td>
</tr>
<tr>
<td>3 MHz</td>
<td>8.163 MHz</td>
</tr>
<tr>
<td>5 MHz</td>
<td>6.897 MHz</td>
</tr>
<tr>
<td>7 MHz</td>
<td>10.53 MHz</td>
</tr>
<tr>
<td>10 MHz</td>
<td>15.38 MHz</td>
</tr>
<tr>
<td>15 MHz</td>
<td>22.22 MHz</td>
</tr>
</tbody>
</table>

- **IF Bandwidth <= 600 kHz:**
  - **RF >= 53 MHz; All models:** IF = 7.438017 MHz \([(9 \times (100e6 / 121))\]
  - **RF <= 53 MHz; PNA-X models:** IF = 2.479339 MHz \([(3 \times (100e6 / 121))\]
  - **RF <= 53 MHz; N522x models:** IF = 826.446 kHz \([1 \times (100e6 / 121)]\)

**Narrowband IF path:**
- **IF = 10.70 MHz**
- **Bandwidth = 30 kHz**

**Manually change the IF frequency**

The **IF frequency** can be changed to any value between +14.9999 MHz and -14.9999 MHz using **SENS:IF:FREQ** or **IFFrequency** commands.

- With DSP Version 4 - 34 and above, min and max IF frequencies up to +/- 20.1 MHz are available.
- With DSP Version 5, min and max IF frequencies up to +/- 38 MHz are available.
- Performance is degraded drastically above +/- 14.9999 MHz.

[Learn about DSP Version](#)
**How to start the IF Path Configuration dialog**

To provide quicker access, use the Setup softkey. Learn how.

**Using front-panel HARDKEY [softkey] buttons**

1. Press **CHANNEL**
2. then **[Hardware Setup]**
3. then **[IF Config]**

**Using a mouse with PNA Menus**

1. Click **Trace/Chan**
2. then **Channel**
3. then **Hardware Setup**
4. then **IF Config**

---

**IF (Receiver) Path Configuration** dialog box help

The IF path, represented in the block diagram at the top of the dialog, is duplicated for each of the receivers (A, B, C, D, R1, R2, R3, R4). In addition, each path can be configured differently for each channel.

**Switch** - Indicates element in the PNA-X and N522x expanded PNA-X and N522x block diagram.

**IF Input** (1) - Available on the PNA-X and N522x with Opt 020. Internal input is a test port or reference receiver input. External Input is through the PNA-X and N522x rear-panel connectors.

**IF Attenuator** (3) - Specify IF attenuation for the narrowband path of the selected receiver.

**IF Filter** (2) - Select Wideband or Narrowband (includes the ability to pulse gates).

**IF Gain** (10) - Set to **Auto** by default, the following are reasons to change the IF Gain:

- For millimeter systems that do NOT use the external millimeter test set, the millimeter head output IFs are routed directly to the RF receivers. You may want to change the IF gain to improve the noise figure.
of the receivers.

- External couplers are often used for high-power test setups. The PNA automatically adjusts the IF Gain depending on the frequency of operation in order to correct for several aspects of the hardware, including the large coupled-arm roll off at low frequencies (below 700MHz). If you replace the internal coupler with one having a different low-frequency roll off, then you may also want to change the IF gain to avoid overdriving the receiver.

- When using the rear panel direct IF inputs, the gain is set low when in standard operation and very high when in millimeter mode. You may want to control the gain of the direct IF inputs to improve noise figure or to avoid overdriving the receivers.

ADC Filter (5) - Select Auto, Narrow (9 MHz or 11 MHz), or Wide (16 MHz or 38 MHz) filter. Learn more.

Couple all IF paths - Check to make the same setting for all receivers.

Expanded PNA-X and N522x Block Diagram (IF / Pulse Generators / DSP)

Blue boxes are configurable elements. Click a blue box, or scroll down, to see how to make settings using SCPI and COM commands.

Receiver / IF Path (top block)

Scroll up for descriptions of the Receiver / IF Paths. Most of these elements can be set from the front-panel User Interface (UI).

Pulse Generators (middle block in above diagram)

See how to make these settings remotely.
Switch 6: Represents Internal or External triggering for the pulse generator. If set to Internal, the pulse generator is internally triggered and puts out a periodic pulse train with a period defined by the pulse generator SCPI or COM commands. If set to External, the period is ignored, and the pulse generator puts out one set of pulses (P0-P4) per external trigger. All five pulse outputs have unique delay and pulse width settings. See all Pulse Generator SCPI and COM commands.

- **External** trigger input is on the Pulse I/O connector pin 7 (PulseSyncIn). The PulseSyncIn line provides a configurable trigger signal into the Pulse Generators. If a level trigger is still valid when the first pulse set is finished, another set will be generated. Only one set of pulses is emitted when edge triggering is used. The length of time that it takes to emit one set of pulses is the end time of the last enabled pulse (largest of width + delay of all the pulses P0-P4.)

The pulse input polarity (positive or negative) and type (edge or level) is configurable only with DSP version: 4.0 FPGA: 34 or higher. Learn more. Otherwise, the pulse generators respond only to positive, level input trigger signals. Externally triggering is configured using the Pulse Generator ‘Polarity’ and ‘Type’ SCPI and COM commands.

**P0:** When P0 is enabled, it is hardwired to trigger the data acquisition ADCs. If the data acquisition system is not ready, the P0 trigger is ignored. If the pulse generator is internally triggered, then the data acquisition system receives periodic triggers. If the pulse generator is externally triggered, then the data acquisition system receives a trigger each time the pulse generator is triggered. In either case, data acquisition is synchronized to the pulse generator. Data acquisition begins on the rising edge of P0. The width of P0 does NOT directly matter as data acquisition does not stop when P0 goes false. The width of P0 DOES matter when determining the time for one set of pulses when externally triggered (see External trigger above). The following describes how the P0 generator triggers data acquisition:

- **Step mode sweeps** of any sweep type: By default, each P0 rising edge triggers a single data point. When point averaging is on, all of the measurements (subpoints) that are required to average each point are made with a single trigger. To individually trigger the acquisition of each subpoint, send the subPointTrigger (SCPI or COM) command.

- **CW sweeps:** Each trigger initiates acquisition for the entire sweep. This is currently used for wideband pulse profiling.

**P1 thru P4** These four pulse generator outputs are hardwired to rear panel outputs on the Pulse I/O connector (pins 10 - 13). They are also routed to two switches (#4 and #7 on the above diagram) along with the following three lines:

- **Rear Panel** External pulse generator input from Pulse I/O connector pin 8 (RFPulseModIn).

- **OFF** Pulse is constantly in LOW state causing gate and source to be OFF.

- **ON** Pulse is constantly in HIGH state causing gate and source to be ON.

Switch 7 Pulse Modulation - 1 of 7 lines to each of the sources. **Important:** When internally modulating the
source leveling must be set to Open-loop.

Rear-panel Outputs: Pulse I/O connector (pins 10 - 13) hardwired.

Source1 and Source2 pulse modulators: (#8 and #9 on the above diagram)

DSP (bottom block)

- Filters the ADC (digital) output from top block and outputs data to the PNA display.

- See SCPI and COM commands to control DSP settings.

See Also

- Pulse Settings
- Remote RF Path Configuration
- Rear Panel Pulse I/O connector

IF Path Configuration using COM and SCPI

Most of the following elements, highlighted in BLUE in the above diagram, have settings that are made using SCPI or COM commands. These are the same commands that are used to make settings in the RF Path Configurator. In general the command specifies an element name and a setting.

See SCPI command
See COM object and example.

<table>
<thead>
<tr>
<th>Ref#</th>
<th>Element Name</th>
<th>Description</th>
<th>Settings</th>
</tr>
</thead>
</table>
| 1    | "IFSWn"                           | For 2-port PNA-X and N522x, \(n = A, B, R1, R2\)                           | "Internal"
For 4-port PNA-X and N522x, \(n = A, B, C, D, R\) (for \(R1\) to \(R4\)) | "External" Rear Panel IF connectors, 4-port PNA-X and N522x use \(R\) for Ref 1 to 4 |
|      |                                   | **For example:** "IFSWB"                                                    |                                                                           |
|      |                                   | **Requires Opt 020 external IF inputs on the rear panel**                   |                                                                           |
| 2    | "IFSigPathn"                      | For 2-port PNA-X and N522x, \(n = A, B, R1, R2\)                           | "WBF" Wide Band Filter Path (default)                                    |
|      |                                   | For 4-port PNA-X and N522x, \(n = A, B, C, D, R1, R2, R3, R4\)             | "NBF" Narrow Band Filter Path                                            |
|      |                                   | "IFSigPathAll" makes setting for ALL receivers.                            |                                                                           |
| 3 | "NBFATNn"  
For 2-port PNA-X and N522x, n = A, B, R1, R2  
For 4-port PNA-X and N522x, n = A, B, C, D, R1, R2, R3, R4  
For example: "NBFATNB" | 0 to 31 in 1 dB steps  
For example: "28" |
|---|---|
| 4 | "IFGate"  
For 2-port PNA-X and N522x, n = A, B, R1, R2  
For 4-port PNA-X and N522x, n = A, B, C, D, R1, R2, R3, R4  
For example: "IFGateB" | "On" Gate is always ON  
"Off" Gate is always OFF  
"RearPanel" (use Pulse IO pins 1 to 5)  
"Pulse1"  
"Pulse2"  
"Pulse3"  
"Pulse4" |
| 5 | "IFAntiAliasFilter"  
- This filter is labeled ADC Filter on the IF Path Configuration dialog.  
- This setting affects ALL receivers. It can NOT be made for individual receivers.  
- The ADC Filter values depend on the DSP Version. Learn more. | "Auto" PNA selects which filter to use based on other IF settings.  
"Narrow"  
Sets 9 MHz for DSP Version 4  
Sets 11 MHz for DSP Version 5  
"Wide"  
Sets 16 MHz for DSP Version 4  
Sets 38 MHz for DSP Version 5  
"9MHZ" is Superseded - Use "Narrow".  
"16MHZ" is Superseded - Use "Wide". |
| 6 | "PulseTrigInput"  
Requires Opt 025 - Four Internal Pulse Generators  
| "Internal" Internal Pulse In - pulse generators are triggered each period.  
"External" External Pulse Synch In - Pulse I/O pin 7 - An external trigger signal is required to trigger the pulse generators for each pulse. |
| 7 | "PulseModDrive"  
Select from 1 of 7 lines to modulate the OUT1 path of Sources 1 and 2.  
Important: When Pulse 1-4 is selected to modulate the sources, source leveling must be set to Open-loop. | "On" Pulse Mod drive is always ON, leaving "SRC1|2 Out 1" ON and not modulated. Default setting.  
"Off" Pulse Mod drive is always OFF, leaving "SRC1|2 Out 1" OFF.  
"RearPanel" (use Pulse IO pin 8)  
"Pulse1"  
"Pulse2" |
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>8</strong></td>
<td>&quot;Src1Out1PulseModEnable&quot;</td>
<td>Requires Opt 021 - Source1 Pulse Modulator</td>
</tr>
<tr>
<td></td>
<td>&quot;Enable&quot;</td>
<td>&quot;Disable&quot;</td>
</tr>
<tr>
<td><strong>9</strong></td>
<td>&quot;Src2Out1PulseModEnable&quot;</td>
<td>Requires Opt 022 - Source2 Pulse Modulator</td>
</tr>
<tr>
<td></td>
<td>&quot;Enable&quot;</td>
<td>&quot;Disable&quot;</td>
</tr>
<tr>
<td><strong>10</strong></td>
<td>&quot;IFGAIN&quot;</td>
<td>&quot;0&quot;, &quot;2&quot;, &quot;4&quot;, &quot;6&quot;, &quot;8&quot;, &quot;10&quot;, &quot;11&quot;, &quot;13&quot;, &quot;15&quot;, &quot;Auto&quot;</td>
</tr>
<tr>
<td></td>
<td>n</td>
<td></td>
</tr>
<tr>
<td>For 2-port models, n = A, B, R1, R2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>For 4-port models, n = A, B, C, D, R1, R2, R3, R4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>For example: &quot;IFGAINB&quot;</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Last Modified:

- 10-May-2011  Added N522x models
- 25-Aug-2010  Modified for new DSP version and IF Gain (A.09.30)
- 8-Mar-2010   Modified for UI (A.09.20)
- 9-Dec-2009   Added external pulse input polarity and type
- 20-Jul-2009  Added subpointtrig content
- 5-Sep-2007   New Image and minor edits
- 5-Feb-2007   MX New topic
External Millimeter-Wave Module Configuration

This feature, when used with the N526xA test sets and external mmWave Modules, extends the frequency coverage of your PNA.

In this topic:

- **Features and Limitations**
- **How to Configure Millimeter Modules**
- **Mixer Mode**
- **mmWave Module Power Level Control**

**See Also**

- [Millimeter-Wave Network Analyzers Technical Overview](#) (Requires internet connection)
- [mmWave Measurements with No Test Set](#)
- [Download a macro for Configuring VDI Frequency Extenders](#) (Requires internet connection)

**Other IF Access Topics**

**Note:** In the PNA user interface and in this help file, the N526xA Millimeter Head Controller is referred to as a test set. OML test head modules are referred to as mmWave modules.

**CAUTION:** Turn OFF test set power before connecting or disconnecting the DC cable to the mmWave modules.

**Features**

- Controls N5260A, N5261A, and N5262A Test Sets.
- Compatible with [iTMSA (True Mode Stimulus)](#).
- Several methods available to provide [Leveled power to the DUT Input](#)
- Compatible with [Integrated Pulse Application](#)
- Allows mixer measurements using SMC. [Learn more.]

The following configurations are supported:

- PNA-X or N522x with options [Opt 020](#) works with N5260A, N5261A, and N5262A Test Sets.
When using the N5262A test set, a 2-port PNA-X or N522x requires Opt 551.

PNA 'C' models with option H11 works with N5260A ONLY.

Limitations

- **Power Settings** When using mmWave modules with an N5260A test set, the PNA can NOT control the power level into your DUT. Your mmWave modules may have a variable attenuator on them. When used with an N5261A or N5262A, after performing a Source Power Cal, then the PNA power settings may be used to control the power into the DUT. See [Leveled Power Capabilities](#).

- To protect your mmWave modules from damage, the settings on the Millimeter Module Configuration dialog can ONLY be changed manually. They can NOT be reset or changed by performing a Preset, by recalling an Instrument State, or from a remote program.

PNA-X Notes

- **CAUTION:** Connect a 10 dB attenuator to the N5260A LO input from the PNA-X LO Output. Otherwise, damage will occur to the N5260A test set.

- The PNA-X or N522x rear panel IF Inputs use 5 SMA connectors. Previous PNA models use BNC connectors. Adapters may be required.

- Beginning with A.09.00, Frequency Offset and SMC Measurements are supported when using mmWave modules. [Learn more](#).

### How to Configure Millimeter-Wave Modules

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then [Configuration]</td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then [Millimeter Module]</td>
<td>3. then <strong>Configure</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>Millimeter Module Config</strong></td>
</tr>
</tbody>
</table>

There are NO programming commands to configure mmWave modules and test set.
Millimeter Module Configuration dialog box help

**Note:** To protect your mmWave modules from damage, settings on this dialog can ONLY be changed manually. They can NOT be reset or changed by performing a Preset, or by recalling an Instrument State, or by a remote program.

**Available Configurations** Lists the Standard PNA configuration and others that you have created. Select Standard PNA to exit mmWave module operation.

**Selected Configuration** Shows the currently selected configuration. Edit this field to change the configuration name. Type a unique name using only alphanumeric characters and underscore.

**New** Click to create a new Millimeter Module configuration. A name is automatically selected. Edit the Selected Configuration field to change the configuration name.

**Remove** Deletes a Millimeter Module Configuration.

**Test Set Properties**

**Selected Test Set** Select a test set to use in the current configuration. The firmware does NOT check to ensure that the selected test set is connected.

**Route PNA RF to rear panel "SW SRC OUT"** Available ONLY on PNA-X with option 224 or 423 AND when using N5261A and N5262A test sets.

When checked, Port 1 source is switched to J11 and Port 3 source is switched to J8 on the PNA-X rear panel. Use this configuration to quickly switch the RF Output back to the PNA front panel.

**Mixer Mode** Check to allow mixer testing using SMC. Learn more.

**Enable Test Set RF ALC** Available for N5261A and N5262A ONLY. When checked, power is automatically leveled at the mmWave module RF input when using the standard cables and making non-pulsed measurements. Clear this box to use non-standard cables or when making pulse measurements. When cleared, the following fields become available:

**Max Power Limit** The maximum mmWave module RF input is limited to this value when Test Set RF
ALC is OFF. When you exit this dialog box using **OK**, set the power out of the PNA using the [Power and Attenuator dialog](#).

**Power Offset** Sets the loss of the cables. The mmWave module RF input is adjusted by this amount. Positive offset increases the power.

**Power Slope** Helps compensate for cable and test fixture power losses at increased frequency. The mmWave module RF input power increases as the sweep frequency increases in dB/GHz. The slope is defined relative to the mmWave module RF input frequency. The slope starts at 0Hz and a positive slope will increase the power level. Range is +/- 2 dB/GHz.

### Frequency Settings

- **Multiplier RF IN** RF Frequency Range (displayed in grey fields) multiplied by this value = test port frequency range.
- **Multiplier LO IN** LO Frequency Range (displayed in grey field) multiplied by this value plus the IF frequency equals the test port frequency. The IF frequency is:
  - 'C' Models = 8.333 MHz
  - PNA-X models = 7.605 MHz

**Test Port Frequency** Set the Start and Stop frequencies of the selected configuration at the test ports. This becomes the displayed Start and Stop frequency of the PNA.

### Important Notes

- To set Test Port Frequency, first set the appropriate **Multiplier** values that are specified in your mmWave module documentation.
- Ensure that the RF and LO Frequencies (highlighted below) are within the frequency range of the sources. The PNA offers no warning if they are NOT.

![Multiplier Frequency Settings](image)

### Source

Click a button to launch the [External Devices dialog](#) where you can select an internal or external source to be used for the PNA LO source or PNA RF source.

- **Cancel** Closes dialog box without saving changes.
- **OK** Saves the configuration and the PNA is **Preset** before making the appropriate settings.

### Mixer Mode

Mixer measurements can be made at mmWave frequencies using **SMC**. (VMC measurements are NOT supported.)

Beginning with A.09.40, mixer measurements can be made with a 2-port test set connected to a 4-port PNA-X. This
configuration yields a 2-port mmWave system. Learn about 2-port system connections and limitations.

Before A.09.40, the Mixer Mode checkbox could be enabled ONLY when the number of PNA test ports matched the number of ports on the mmWave test set. This means that an N5261A (2 port test set) could ONLY be connected to a 2-port PNA-X and an N5262A (4 -port test set) could ONLY be connected to a 4-port PNA-X. Although N5250A systems can make SMC mixer measurements, these measurements rely on accurate power calibrations which is beyond the capability of the N5250A.

Procedure

1. Connect your DUT to the mmWave system as described below.
2. Configure this dialog (Millimeter Module Configuration). Check Mixer Mode, then press OK. This presets the PNA.
3. Create an SMC measurement.
4. Make mixer settings. As with standard SMC measurements, only two DUT ports can be swept in frequency. The remaining DUT port must be a fixed frequency. See configuration used for harmonic mixers.
5. Increase power for mmWave modules that are connected directly to a PNA port or external source.
6. Calibrate using the SMC Calibration Wizard with mmWave Power Control.

Hardware Connections for Mixer mode

The following image shows the standard connections from a N5261A or N5262A test set port to an OML mmWave module.

mmWave mixers usually require that two of the three mixer ports operate at mmWave frequencies. When Mixer Mode is checked on this dialog (Millimeter Module Configuration), the following restrictions apply:

- On a 2 port mmWave system, only port 1 of the test set can be used as a mmWave-frequency port. Port 2 can NOT be used.
On a 4 port mmWave system, only ports 1 and 3 of the test set can be used as mmWave-frequency ports. Ports 2 and 4 can NOT be used.

The SMC parameter being measured must be within the frequency range of the PNA or within the frequency range of the banded mmWave module. Frequencies in between these ranges are allowed by the SMC mixer setup dialog, but the measurement results on the screen are NOT accurate.

Connections with a 4-port mmWave system

Upconverters

- DUT Input - Connect to PNA port 2 or port 4.
- DUT LO - Connect mmWave module to test set Port 3.
- DUT Output - Connect mmWave module to test set Port 1.

Downconverters

- DUT Input - Connect mmWave module to test set Port 1.
- DUT LO - Connect mmWave module to test set Port 3.
- DUT Output - Connect to PNA port 2 or port 4.

Connections with a 2-port mmWave system

Although supported, testing mmWave mixers with a 2-port system can be challenging for the following reasons:

- Testing mmWave mixers requires that two of the three DUT ports be at mmWave frequencies.
- Only test set port 1 is capable of adequately driving a mmWave module when used as a receiver.
- Therefore, the second DUT port that requires mmWave frequencies must have the mmWave module connected directly to an external source or a PNA second source.
- When using the mmWave module as a source, only the DC Bias and RF cable is necessary. The LO cable to the mmWave module is NOT used. This is because the RF input frequencies are multiplied in the mmWave module to provide the source frequencies. So a mmWave module used as a source can use the the RF cable to connect directly to the PNA second source or an external source. About +5 dBm of RF power is required to adequately drive the mmWave module.

Downconverters - requires two mmWave modules as sources

- DUT Input - Connect the mmWave module to the test set port 1.
- DUT LO - Connect the RF cable of the mmWave module to an external source or the PNA (SRC2) second source.
- DUT Output - Connect to PNA port 2.
**Upconverters** - requires a mmWave module as a **source** at the DUT LO and a mmWave module as a **receiver** at the DUT Output:

- DUT Input - Connect to PNA port 2.
- DUT LO - Connect the RF cable of the mmWave module to an external source or the PNA (SRC2) second source.
- DUT Output - Connect the mmWave module to the test set port 1.

**Measuring Harmonic Mixers**
Harmonic mixers have a multiplier circuit in the LO port of the DUT. Enter the multiplier value in the numerator of the X LO port in the **SMC mixer setup dialog**. This will provide the correct LO frequencies out of the appropriate source.

**mmWave Module Power Level Control**
Beginning with A.09.40, the following TWO features are integrated into **Guided Cal**:

- **For S-parameter Cal** - Use Multiple Sensors
- **For SMC Cal** - Power Table

The following table shows features that can be used to provide leveled power to the input of your DUT for S-parameter and SMC measurements.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
<th>Use when...</th>
<th>Use for...</th>
<th>Access the feature...</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Receiver leveling</strong></td>
<td>Provides a sweep-to-sweep leveled power.</td>
<td>Works anytime.</td>
<td>S-params and SMC</td>
<td>Before or after Cal</td>
</tr>
<tr>
<td><strong>Use Multiple Sensors</strong></td>
<td>Allows several power sensors to be used to calibrate source power.</td>
<td>You require more than one power sensor to complete the source power calibration of the measurement frequency range.</td>
<td>S-params</td>
<td>During <strong>Guided Power Cal</strong></td>
</tr>
<tr>
<td><strong>Power Table</strong></td>
<td>Build or use a file that contains data of mmWave module output power vs frequency.</td>
<td>A power sensor is NOT available for calibration of the mmWave modules being used.</td>
<td>S-params</td>
<td>During <strong>Std Source Power Cal</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SMC</td>
<td>During <strong>Guided Power Cal</strong></td>
</tr>
</tbody>
</table>
S-parameter measurements

If you have one or more power sensors that spans the frequency range of your measurement, then use the following process.

Otherwise, perform a standard Source Power Cal. Learn how.

Using one or more power sensors

Check ALC Enabled (if available) on the Millimeter Module Configuration dialog.

1. With an S-parameter measurement active, press Cal, then Start Cal, then Calibration Wizard, then SmartCal.

2. On the following Select Ports dialog, check Calibrate source and receiver power, then click Next.

3. Important: In the following dialog, check Use Multiple Sensors, even if using only one sensor.

4. Complete the Guided Cal process.

Note: During the 'Connect a power sensor to port n'...step, the following error message may be displayed:
The default power level of 11 dBm is unachievable after calibration. Lower the power before starting calibration.
This means that a high amount of loss was measured in the path, and 11 dBm at the test ports will not be
possible.

**Cancel** the calibration and lower the source power level using the Power and Attenuators dialog.

**Perform a standard Source Power Cal - S-parameter measurements**

When one or more power sensors that spans the frequency range of your S-parameter measurement are NOT available, then use the following process.

**Note:** Perform an S-parameter calibration AFTER performing the following Source Power Cal.

1. Check **ALC Enabled** (if available) on the Millimeter Module Configuration dialog.
2. Press **Cal**, then **Power Cal** then **Source Cal** then **Options** to launch the following dialog:

   ![Source Power Calibration Options](image)

   See the help topic for this dialog

3. If one does not already exist, create a power table to be used to calibrate the PNA receiver. Learn how.
4. Check **Use a power table and the PNA reference receiver**.
5. Click **Power Table**, then navigate to the *.prn file.
6. Click **OK**.
7. Check **Calibrate the source at multiple power levels**.
8. Click **Power Levels**, then enter the Max power, Min power, and Step Size at which source power should be corrected. Be sure that the source power for your measurement is within these power levels. Otherwise, source power will NOT be accurate. Learn more about this feature.
9. Check **Calibrate the PNA reference receiver**, then click **OK**.
10. On the Source Power Cal dialog, click **Take a sweep**. The output of the test set is set to Max power and a sweep is performed to calibrate the reference receiver.

11. Power is dropped for several subsequent sweeps. The calibrated reference receiver is used to fully characterize the source power.

12. The entire correction table can be saved along with the instrument state in a *.csa file. Learn how.

13. Power out of the input module should be flat and accurate.

**SMC Cal**

Use the following calibration process to achieve accurate, leveled power at the mmWave test ports.

1. With a configured SMC measurement active, press **Cal**, the **Start Cal**, then **Calibration Wizard**.

2. At the SMC **Calibration Setup** dialog, when a Thru standard is NOT available, check **Independent power cals for input and output ports (no thru)**.

3. On the **Select Ports** dialog, check **Calibrate source and receiver power**, then click **Next**.

4. At the following Power Cal settings dialog:

   ![Power Cal Settings for Port 1](image)

   **Learn about this dialog**

   a. When you have ONE power sensor that spans the frequency range of your SMC measurement, then click **Power Meter Settings** to configure the power sensor. There are currently NO provisions for using multiple power sensors with SMC Calibration.

   b. Otherwise, use the following Power Table process.

      i. If one does not already exist, create a power table to be used to calibrate the PNA receiver. Learn how.

      ii. Check **Use Power Table**.

      iii. Click **Power Table**, then navigate to the *.prn file. The selected *.prn file is annotated to the dialog.

      iv. Click **OK**.
5. If you checked **Independent power cals for input and output ports (no thru)**, you will ALSO be prompted to select a power table for Port 2.

6. Complete the [Guided SMC Cal](#) process.

### Power Table

**Note:** This is NOT the same table that is used for the [Calibrate the source at multiple power levels](#) feature.

A power table is a text file with data that describes the output power of the module as a function of frequency. This is valid when the mmWave module is driven at high levels (+11 dBm). This file may have been created for you by a third party or shipped with your mmWave Module. If not, you can create this *.prn file from measured values using the following procedure:

**How to create a Power Table from measured data**

1. Setup the mmWave measurements including the [MM Module Configuration dialog](#) settings.

2. If a power sensor is available that spans the frequency range of your measurement, then do a source power calibration using a power meter for the first iteration. Otherwise, skip this step.

3. Set PNA power to the maximum value (+11 dBm).

4. Create an R1 trace and measure. [Learn how.](#) The loss through the MM Module is small enough to not be considered.

5. Click **File** then **Save As** to save the R1 trace to a *.prn file. Make note of the folder.

6. The file is recalled in the above procedure.

This file can now be used instead of connecting the power meter for this module.

This file can also be created manually, using a text file program such as Notepad. Copy the header information, and create the file with two columns, one for frequency and one for output power.
Example .prn file

Note: With Rev. 09.31, the first line of the *.prn file must have the Input power at which these measurements were made. Otherwise, an error message appears with the default value that will be assumed. See above image for format.

Last modified:

26-Apr-2011 Added 2-port test set w/ 4-port PNA (A.09.40)
22-Oct-2010 New multiple power levels and prn format
19-Oct-2010 Added new SPC Options dialog
9-Dec-2009 Added Cal note for waveguide
28-Aug-2009 Added Mixer Mode (A.09.00)
11-Mar-2009 Added Leveled power (A.08.5)
26-Feb-2009 Added Opt 551 requirement
14-May-2008 Major updates for PNA-X 8.2
9-Apr-2007 Updated for PNA-X
9/12/06 Note - NOT compatible with Freq Offset
N5264A Measurement Receiver

The N5264A is a PNA-X with no Source Ports or Tunable receivers. This makes the N5264A a very fast and very sensitive IF receiver that has been designed specifically for antenna and radar cross-section (RCS) measurements. When used with external sources, the N5264A can make frequency-scan measurements of antennas, or make RCS measurements in time domain.

- **N5264A Options and Limitations**

- **Configuring the N5264A**
  - **IF Receivers**
  - **Internal LO Source** (Opt 108)
  - **External Sources**
  - **How to Create Measurements**
  - **Calibrating the N5264A**

- **See Also**
  - **FIFO and other Antenna Features**
  - **Antenna Selection Guide**
  - 85309 Manual updated for PNA -P/N 85310-90002
  - **Antenna and RCS Measurement Configurations**
  - **N5264A Specs**
Typical N5264A Application
Far Field Outdoor Antenna Measurements

N5264A Options and Limitations

The N5264A has the following options:

- **IF Receiver** (base model) - The A,B,C,D, and R receivers are always set at the specified IF frequency.

- **IF Receiver + LO source** (Opt 108) - In addition to the receiver-only option, this option adds a 10 MHz to 26.5 GHz LO source.

- **Fast-Sweep / FIFO Mode** (Opt 118) - These features together allow you to make very fast measurements and save the data to a remote computer. Learn more.

Limitations

The following PNA features are NOT available on the N5264A

- No Source ports

- No S-Parameters - **Arbitrary ratioed parameters ARE allowed.**

- **No Application support**

- **Limited Calibrations**

Configuring the N5264A

IF Receivers

3689
The five IF receiver inputs are on the N5264A rear panel.

All five IF receivers are measured at the same time. However, only those measurements that are displayed are updated.

External mixers are always required to down-convert signals to the IF frequency.

The default IF frequency for the N5264A is 7.605634 MHz.

Change the IF frequency to any value between +14.9999 MHz and -14.9999 MHz using SENS:IF:FREQ (SCPI) or IFFrequency (COM) commands.

Internal LO Source (Option 108) - 10 MHz to 26.5 GHz
The LO source output connector is on the N5264A rear panel.
The power level of the internal LO source is typically about +10 dBm and can NOT be changed.
To change the frequency of the LO Source:

1. Change the IF frequency of the measurement if necessary.
2. On the FOM dialog, change Receivers frequencies to the RF frequency range to be measured.
3. The LO Source frequency range is set automatically. This frequency value can not be viewed.

External Sources
Because the N5264A has no internal sources (except for the optional LO), newly created measurements are displayed with a source port = 0 (zero).

External sources can be configured so that they are controlled by the N5264A. Learn how to configure an external source.
Once configured, all existing and new measurements are changed to use the external source as the source port. When two or more sources are configured, the first configured source is displayed by default.
The external source settings can be changed from the following dialogs:

- Power and Attenuators dialog - Controls the ON | OFF state (set to Auto by default) and Power level of the external source.
- FOM dialog - Controls the frequency range. External sources are listed by name and uncoupled by default.

How to Create Measurements
Create ratioed and unratioed measurements using the standard Receivers tab. S-parameters are not available.

Calibration
The only calibrations available are Response, Source Power and Receiver calibrations. These are performed from the Unguided Cal Wizard.
Last Modified:

27-Sep-2010   Updated Opt 108

6-Apr-2009    Removed N5242A


28-Aug-2008   MX New topic
External Source Configuration

Once configured, an external source appears in PNA dialogs as though it is an internal source.

How to Configure an External Source

1. Create an External Source device by name (one-time).
2. Then click **Device Properties** to **Configure the External Source**.

**PNA Applications** have additional methods of launching this dialog.

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then <strong>[Configure]</strong></td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then <strong>[External Device Configuration]</strong></td>
<td>3. then <strong>Configure</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>External Device Configuration</strong></td>
</tr>
</tbody>
</table>

See **Remotely Specifying a Source Port**

External Device Configuration dialog box help

*Note:* This dialog is also used to configure an External Power Meter as a **PMAR** device. [Learn how.](#)

To use this dialog, your PNA must have **FOM Option 080**. Without this option, you must control an external
This dialog allows you to configure an external source to be controlled by the PNA. Once you create and activate an external source from this dialog box, it becomes available from the following PNA dialog boxes as well as the softkeys and entry toolbar, as if it were an internal PNA source or receiver: Power and Attenuators dialog; FOM dialog, New Trace / Receivers tab dialog

**Important Notes**

- By default, an external device is **de-activated** when the PNA is Preset or when a Instrument State is recalled. This behavior can be changed with a Preference setting so that it remains active through a Preset or Instrument State recall.

- Device configuration is NOT saved in an Instrument State file. Therefore, recalling a state file that refers to a device that has been removed, or recalling a state file on a different PNA will result in a “Device configuration not found” error.

- Multiple configurations for the same physical device can be Active. However, only one configuration for the same external source can have the I/O Enabled.

**External Devices**

The devices that are currently configured appear in this list. The number of devices that can be configured is limited by the specified Interface.

- **New** Click to create a new source configuration. The default name is Device<n>, where <n> is the next number for ‘Device’.

- **Remove** Click to remove the selected device from the list.

**Properties**

- **Name** Enter a device name as it will appear when referring to this device in all PNA dialog boxes. Edit the name at any time. Duplicate names are not allowed.

**"Name" Notes**

- Because External Devices can be used with FOM ranges, do NOT name an external device any of the following FOM range names: “primary”, “receivers”, or “source”, “source1”, source2” and so forth. Learn more about FOM ranges.

- Do NOT use a parameter name, such as “S11”, or “R1”.

**Device Type** Select either Source.

**Driver** Select the appropriate model of the device to be configured. AG is short for Agilent. Choose from:

- AGESG (ESG)
- AGPSG (PSG)
- AG836XX (8360 and 8340)
- AGMXG - (MXG) Requires at least firmware A.01.44 for FOM power sweep to work correctly.
AGGeneric - For sources NOT listed. Click Device Properties, then Edit Commands to send commands to these sources. Learn how.

Active  Check to make the device available for use in the FOM, New Trace, and Receiver Leveling dialogs. An instrument state that is saved with an Active device (checked) will include the device in the state file. Otherwise, if the Active box is cleared, the device will NOT appear in the state file.

Enable I/O  Clear this box to disable communication with the selected device. You would do this to configure a device that is not yet connected to the PNA.

- Communication with devices is attempted when Enable I/O is checked, Active is checked, and OK is pressed.
- If communication with a device is lost the affected channels are put into Hold.
- When communication is attempted, devices with Enable I/O checked are queried for limits for frequency, power, and number of points. If there are limit problems, the PNA sends an error and the affected channels are put into Hold. These limits are enforced by the dialog box in which they are set. Resolve the reported limit problem and then restore the triggering.

Device Properties  Click to launch the Configure External Sources dialog.

IO Configuration

Interface  Select the interface that is used to connect the device to the PNA. These devices will then appear in the 'Available' field. Choose from:

- GPIB - Devices connected to the System Controller GPIB port.
- USB - Devices connected to the PNA USB ports. See Important First-time USB connection note.
- Aliases - Devices that are connected to ANY interface for which you created an alias. See Configure Alias and LAN devices.
- LAN - Devices connected to a network using a LAN connection. The PNA must also be connected to the network.

Note: Devices connected to LAN must first be configured in Agilent IO libraries before they will appear on the Available list. See Configure Alias and LAN devices.

Available  Shows a list of devices that are connected to the specified IO Interface.

Refresh  Click to rescan the specified interface for devices.

Selected  Enter the IO configuration or select from the available list of IO Interfaces found.

Configure Alias and LAN Devices

Use this procedure to configure a device using a LAN interface. Also use for ANY device for which you want to set an alias (easily-recognized) name. The alias name appears in the Available field when Aliases is selected as the Interface.

1. On the PNA, minimize the PNA application.
2. In the system tray (lower-right corner) right-click the IO icon, then click Agilent Connection Expert

To Add a LAN Device:

1. In ACE, click Add Instrument

2. Select Add LAN Instrument (TCPIP0) or USB0, then click OK.

3. Click, then enter the IP address of the external source.

4. Click Test Connection to verify communication.

5. Click OK.

To create an Alias for a connected device:

1. In the list of connected instruments, right click the external source, then Add VISA Alias.

2. Enter the same PNA source name that was, or will be, used in the External Device Configuration dialog.

**External Source Configuration**

**Important Notes about External Sources**

- By default, an external source is de-activated when the PNA is Preset or when a Instrument State is recalled. This behavior can be changed with a Preference setting.

- External Agilent sources are usually limited to 1601 points with List-sweep mode. To 'work around' this limitation, divide the measurement among multiple channels. For example, to attain a sweep of 3200 points, create two channels of 1600 points. You can also use manual source control which supports Step-sweep mode. In this mode an external source can have up to 65,535 points. See Synchronize an External Source for help with manual source control.

- External sources should always share the same 10 MHz Reference signal as the PNA. Connect a BNC cable from the PNA 10 MHz Ref Output to the External Source Input.

- All newly-activated sources are preset, with source power OFF. Source power must be turned ON in the Power dialog. Frequency Offset must be enabled in the FOM dialog.

- When daisy-chaining multiple sources in Hardware List triggering, the source to receive the Trigger signal from the PNA must be the first source listed in the Selected column of the External Device Configuration dialog. Devices are listed in the order in which they are created. You may have to delete, then re-create a source to move it down on the list.

- The same source can NOT be used more than once in the same channel.

- The PNA automatically controls all trigger settings for the external source. The PNA can be triggered internally or externally using the Internal, Manual, or External trigger settings. When set to External, the trigger signal must come through a PNA rear panel connector that is not being used to trigger the
external source. Use Meas Trig to trigger the PNA. For 'C' models, you have to manage the external trigger yourself.

- For more information, see PNA Trigger model

**Programming Commands**

**See Also**

Remotely Specifying a Source Port

**External Source Configuration** dialog box help

This dialog box is used to make external source settings.

See Important Notes about External Sources

**Source Settings**

- **Timeout (sec)** Sets a time limit for the source to make contact with the PNA. If this time limit is exceeded, the PNA stops the measurement procedure and displays the following error message.

  EXECUTION ERROR;OPC QUERY TIMEOUT ERROR: FREQUENCY NOT SETTLED

  If this occurs, check the connections between your PNA and external source.

- **Dwell per point (ms)** Applies a dwell in Hardware List triggering ONLY. Set the time (in milliseconds) the external source will wait before data acquisition.

- **Edit Commands** Provides a method to send SCPI commands to AGGeneric (not listed) sources.

**Trigger Settings**

**Trigger Mode**

- **Software CW (GPIB)** Slowest method.
- The external source receives each CW frequency from the PNA over GPIB, USB, or LAN. No other trigger cables are required.

- The PNA controls ALL external source trigger settings automatically (except for those on this dialog). All settings in the **External Trigger** dialog are ignored.

**Hardware List (BNC)** Fastest method.

- The external source receives a list of CW frequencies from the PNA, then receives BNC trigger signals as required from the PNA.

- If the number of data points used in the measurement exceeds the capability of the external source, the PNA automatically switches to Software CW (GPIB) trigger mode. This will slow the measurement significantly.

- NOT available for AGGeneric (not listed) sources.

- The PNA controls ALL external source trigger settings automatically (except for those on this dialog). All settings in the **External Trigger** dialog are ignored.

**Trigger Port** Used ONLY for Hardware List Trigger Mode. Select the PNA rear panel connector to be used for triggering. The sources must be connected as follows:

- **PNA-X and N522x models**: For single external sources, connect directly using Aux1 or Aux2. [See rear panel Aux connectors](#). Connect multiple sources using the following daisy-chain image.

- **'C' models** - Use rear-panel BNC **Trigger** connectors as follows:

<table>
<thead>
<tr>
<th>1 External Source</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Diagram" /></td>
</tr>
</tbody>
</table>

| Daisy-chain 2 External Sources |
**Note:** Source 1, which receives the trigger out of the PNA, must be the first source listed on the External Devices Configuration dialog box. Devices are listed in the order in which they are created. You may have to delete, then re-create a source to move it down on the list.
Generic Source Commands dialog box help

Enter the SCPI commands that control the following functions on your AGGeneric (not listed) source. A field without a SCPI command entered will be ignored and that function will not be set.

To launch this dialog, click Edit Commands on the External Source Properties dialog.

Operation Complete (*OPC).

Preset   Presets the source
Set CW Frequency  Sets CW Frequency
Set CW Sweep Mode  Sets source sweep mode
Set Power   Sets source power
Set Power State  Turns power ON or OFF

Last Modified:

24-Jun-2011  Several edits - including link to first time note.
29-Apr-2011  Removed PMAR notes
10-Feb-2011  Added Guided Power Cal
13-Oct-2010  Removed PMAR name restriction
30-Jul-2009  Major changes (9.0)
7-Apr-2009   Added Dwell
23-Mar-2009  Added note about MXG and E836x
3-Sep-2008   Removed legacy content
22-Aug-2008  Added list of supported..
22-Aug-2008  Added Generic source control
11-Feb-2008  Added limitation note at top
23-Jan-2008  Added Selected ordering notes
5-Nov-2007   Added links for remote selection
18-Jul-2007  Edited for FCA LO Cal changes
30-Apr-2007  MX Modified for ALL external source config.
Configure a Power Meter As Receiver (PMAR)

When a power meter is configured as a PNA receiver, you can...

- Extend the number of measurement receivers.
- Use the power meter as a scalar detector.
- Monitor the power at any point in a measurement system.
- Use the power meter to level the stimulus power at any point in a measurement system.
- Use the power sensor as a PMAR device to confirm the accuracy of a Source Power Cal. Learn how.

The following Agilent Power Meters are supported as receivers:

- HP 437B / 438A power meters
- E-Series power meters (E4418 and E4419) and all supported sensors.
- P-Series power meters (N1911A and N1912A) and all supported sensors.
- EPM Series power meters (E4416A and E4417A) and all supported sensors.
- N1913A and N1914A power meters and all supported sensors.
- U2000 Series USB power sensors. USB power sensors must be connected directly to one of the PNA USB ports. See Important First-time USB connection note.

N1911A, 12A, 13A, and 14A power meters have a ‘device-side’ USB connector and are controlled by the PNA exactly like a USB sensor. See USB Power Sensors (below). Although these meters may also have a front-panel USB port, USB power sensors must be connected directly to one of the PNA USB ports.

In general, a power meter that is configured as a receiver can be used like any other PNA receiver in the following dialogs:

- **New Trace / Meas dialog** - used in Ratioed and Unratioed measurements.
- **Receiver Leveling**
- **Frequency Offset Mode** - Extend frequencies beyond PNA
How to Create and Configure a PMAR Device

1. Create a PMAR device by name (one-time).
2. Then click **Device Properties** to configure the Power Meter/Sensor.

**PNA Applications** have additional methods of launching this dialog.

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press <strong>SYSTEM</strong></td>
<td>1. Click <strong>Utility</strong></td>
</tr>
<tr>
<td>2. then <strong>[Configure]</strong></td>
<td>2. then <strong>System</strong></td>
</tr>
<tr>
<td>3. then <strong>[External Device Configuration]</strong></td>
<td>3. then <strong>Configure</strong></td>
</tr>
<tr>
<td></td>
<td>4. then <strong>External Device Configuration</strong></td>
</tr>
</tbody>
</table>

**External Device Configuration** dialog box help

This dialog allows you to create and configure a power meter to be used as a receiver by the PNA. Once you create and configure a power meter from this dialog box, it becomes available from PNA dialog boxes as well as the softkeys and entry toolbar, as if it were an internal PNA receiver.

- This dialog is ALSO used to configure an External Source. Learn more.
- To configure a single power meter for a Source Power Cal, use the **Power Meter Settings** dialog.
Important Notes

- By default, an external PMAR device is **de-activated** when the PNA is Preset or when an Instrument State is recalled. This behavior can be changed with a Preference setting so that it remains active through a Preset or Instrument State recall.

- PMAR configuration is NOT saved in an Instrument State file. Therefore, recalling a state file that refers to a device that has been removed, or recalling a state file on a different PNA will result in a “Device configuration not found” error.

External Devices

The devices that are currently configured appear in this list. The number of devices that can be configured is limited by the specified Interface.

**New** Click to create a new PMAR configuration. The default name is Device<n>, where <n> is the next number for ‘Device’.

**Remove** Click to remove the selected device from the list.

Properties

**Name** Enter a device name as it will appear when referring to this device in all PNA dialog boxes. Edit the name at any time. Duplicate names are not allowed.

**Notes**

- Because External Devices can be used with FOM ranges, do NOT name an external device any of the following FOM range names: “primary”, “receivers”, or “source”, “source1”, source2” and so forth. Learn more about FOM ranges.

- Do NOT use a parameter name, such as "S11", or "R1".

**Device Type** Select **Power Meter**.

**Driver** Use **AGPM** for all Agilent Power Meters.

**Active** Check to make the device available for use in the FOM, New Trace, and Receiver Leveling dialogs. An instrument state that is saved with an Active device (checked) will include the device in the state file. Otherwise, if the Active box is cleared, the device will NOT appear in the state file.

**Note:** Multiple PMAR configurations for the same physical device can be Active and Enabled.

**Enable I/O** Clear this box to disable communication with the selected device. You would do this to configure a device that is not yet connected to the PNA.

- Communication with devices is attempted when Enable I/O is checked, Active is checked, and OK is pressed.

- If communication with a device is lost, the affected channels are put into Hold.

- When communication is attempted, devices with Enable I/O checked are queried for limits for frequency, power, and number of points. If there are limit problems, the PNA sends an error and the affected channels are put into Hold. These limits are enforced by the dialog box in which they are set.
Resolve the reported limit problem and then restore the triggering.

- Communication is also attempted when clicking the **Settings** button on the [Configure Power Sensor](#) dialog. You can not change any of the sensor settings unless **Enable I/O** and **Active** are checked and communication is possible with the sensor.

**Device Properties** Click to launch the [Configure Power Sensor](#) dialog.

**IO Configuration**

**Interface** Select the interface that is used to connect the device to the PNA. These devices will then appear in the ‘Available’ field. Choose from:

- **GPIB** - Devices connected to the System Controller GPIB port.
- **USB** - Devices connected to the PNA USB ports. See Important First-time USB connection note.
- **Aliases** - Devices that are connected to ANY interface for which you created an alias. See Configure Alias and LAN devices.
- **LAN** - Devices connected to a network using a LAN connection. The PNA must also be connected to the network. **Note:** Devices connected to LAN must first be configured in Agilent IO libraries before they will appear on the Available list. See Configure Alias and LAN devices.

**Available** Shows a list of devices that are connected to the specified IO Interface.

**Refresh** Click to rescan the specified interface for devices.

**Selected** Enter the IO configuration or select from the available list of IO Interfaces found.

**Configure Alias and LAN Devices**

Use this procedure to configure a device using a LAN interface. Also use for ANY device for which you want to set an alias (easily-recognized) name. The alias name appears in the Available field when Aliases is selected as the Interface.

1. On the PNA, minimize the PNA application.
2. In the system tray (lower-right corner) right-click the IO icon, then click **Agilent Connection Expert**

To Add a LAN Device:

1. In ACE, click ![Add Instrument]
2. Select Add LAN Instrument (TCPIP0) or USB0, then click OK.
3. Click, then enter the IP address of the external source.
4. Click Test Connection to verify communication.
5. Click **OK**.
To create an Alias for a connected device:

1. In the list of connected instruments, right click the instrument, then Add VISA Alias.

2. Enter the same Device Name that was, or will be, used in the External Device Configuration dialog.

---

**Power Sensor Configuration** dialog box help

To launch this dialog, with the PMAR device selected in the External Device Configuration dialog, click Device Properties.

This dialog is used to configure a power meter / sensor for use as a receiver.

To configure a single power meter for a Source Power Cal, use the Power Meter Settings dialog.

---

**About Power Sensor Calibration**

PMAR traces are NOT calibrated using standard PNA calibrations, including response corrections.

PMAR traces are calibrated using methods that are appropriate for the selected sensor. Follow the proper guidelines for zeroing or calibrating the sensors that are in use. Check to ensure that the selected sensor is appropriate for the frequency range and the power level at which PMAR measurements occur.

The PNA does not automatically prompt you to perform a calibration.

To calibrate a power sensor, click Settings on this dialog box, then click Zero/Calibrate Sensor. Learn more.

---

**Note:** By default, a PMAR is de-activated when the PNA is Preset or when a Instrument State is recalled. This behavior can be changed with a Preference setting.

---

**Sensor**
For power sensors that are connected to a power meter, select a sensor to configure.

**Settings**  Click to launch the [Power Sensor Settings](#) dialog.

When pressed, communication with the sensor is tested. Sensor settings can NOT occur unless **Enable I/O** is checked on the [External Device Configuration dialog](#), and the sensor is properly connected and configured.

**Sensor Settling**

Each power meter reading is "settled" when either:

- two consecutive meter readings are within this Tolerance value **or**
- when the Max Number of Readings has been met.

The readings that were taken are averaged together to become the "settled" reading.

**Tolerance**  When consecutive power meter readings are within this value of each other, then the reading is considered settled.

**Max Number of Readings**  Sets the maximum number of readings the power meter will take to achieve settling.

**Sensor Loss Compensation**

**Use Loss Table**  Select this checkbox to apply loss data to Source Power calibration correction (such as for an adapter on the power sensor).

**Edit Table**  Invokes the [Power Loss Compensation](#) dialog box.
This dialog appears when you click the **Settings** button on the Configure Power Sensor dialog.

**Note:** Be sure that the frequency range of your power sensor covers the frequency range of your measurement. This does NOT occur automatically.

**Sensor A (B)** Displays one of the following messages depending on type of sensor.

- **Not connected** The PNA is not detecting a power sensor.

- **Cal factors are contained within this sensor** Internal Reference Cal Factor and Cal Factor data are loaded automatically. The following settings do not apply.

- **Sensor Data** Allows the following entries for power sensor data:

  - **Reference Cal Factor** Specifies the Cal Factor for the 50 MHz reference signal.
  - **Cal Factor Table** Specifies the frequency and corresponding Cal Factor for the sensor.
  - **Delete Cal Factor** Deletes the indicated row in the table.
  - **Delete All** Deletes all data in the table.
  - **To Add a Row** to the table, click on a row in the table and press the down arrow on either the PNA front panel or keyboard. A row is added to the bottom of the table. The table is automatically sorted by frequency when OK is pressed.

**Limit Frequency Range**
- Check to limit the use of the power sensor to those within the Minimum and Maximum frequency values.

- Clear to use the power sensor for all measurements. If the measurement frequency is not within the Minimum and Maximum frequency values, the closest min or max correction data is used for the measurement.

**Minimum Frequency**  Specifies the minimum frequency range for the sensor.

**Maximum Frequency**  Specifies the maximum frequency range for the sensor.

**Perform Sensor Zeroing and Calibration**  Zero and/or calibrate the power sensor before measuring data. Follow prompts that may appear. Press **Zero/Calibrate Sensor** to perform required calibration steps.

If the following settings are 'greyed', Internal or External zeroing is selected automatically based on the power meter/sensor model.

- **Internal Zero** - A switch inside the power sensor removes the zero circuit from the incident power.

- **External Zero** - Requires that you physically remove the sensor from incident power.

**Note for the U2000 Series USB power sensors**

Select External Zero ONLY when the power to be measured is below the specified level. Otherwise, the U2000 series performs internal zeroing automatically when needed. See your power sensor documentation for more details.

- U200xA - below -30 dBm
- U200xH - below -20 dBm
- U200xB - below 0 dBm

If your U2000 power sensor 'hangs' when external zeroing, upgrade the power sensor firmware to Rev. A.01.02.00 or higher to fix this problem.
Power Loss Compensation dialog box help

To Add a Row to the table, click on a row in the table and press the down arrow on either the PNA front panel or keyboard.

To Edit a value, double-click in the cell to be edited.

These values can be loaded from an S2P file using the Characterize Adaptor Macro.

Compensates for losses that occur when using an adapter or coupler to connect the power sensor to the measurement port. These components will be removed when the calibration is complete. To account for components that will remain during the measurement, use the Power Offset setting.

The Frequency / Loss pairs define the amount of loss for the entire frequency range. For example, using the entries in the above dialog image:

- 0.5 dB is used to compensate power sensor measurements up to 1 GHz.
- Each data point between 1 GHz to 2 GHz is linearly interpolated between 0.5 dB and 1 dB.
- 1 dB is used above 2 GHz.
- A single frequency/loss segment is applied to the entire frequency range.

Enter up to 100 segments to achieve greater accuracy.

**Frequency** Enter a frequency in Hz.

**Loss** Enter a loss as a POSITIVE value in dB. To compensate for gain, use NEGATIVE values.

**Delete Table Segment** Deletes row indicated in the field.

**Delete All** Deletes all data in the table.

The Power Loss Compensation table survives PNA Preset and Power OFF. To NOT use Loss compensation, clear the Use Loss table checkbox on the Configure Power Sensor dialog.

---

Use a PMAR Device to confirm a Source Power Cal

Learn how to create and configure PMAR device.

After a Source Power Cal has been performed, use the same sensor as a configured PMAR to analyze the accuracy of the Calibration.
1. Create a PMAR device with the power sensor that will be used for the Source Power Cal.


3. Create an unratioed measurement with the PMAR device. Learn how.

4. With the power sensor still connected to the test port, monitor the corrected source power using Min and Max markers or the Trace Statistics peak-to-peak feature.

Last Modified:

10-Jun-2011 Modified list of supported PMs
29-Apr-2011 Separated source and PMAR
22-Oct-2010 Added support for N991x meter
25-Aug-2009 MX New topic
The E5091A is a popular Agilent Technologies 7-port / 9-port test set. Although the test set was originally designed to work with the ENA Network Analyzer, it also works well with the PNA. This topic describes how to control the test set from the PNA. For more information about the test set, refer to your E5091A documentation.

- Overview
- Connecting the E5091A
- How to make E5091A test set Control Settings
- Calibrating with the E5091A

**Overview**

When connected to the PNA, the E5091A test set provides full 7-port or 9-port test capability. The E5091A can be configured to switch a different test set path for each PNA channel. When all channels have been configured, the entire measurement setup and calibration can be saved to a .cst or .csa file to be recalled later. In addition, the Channel Settings Table that is appended to a printed hardcopy of a measurement includes the E5901A Port Control settings.

**Notes:**

- ONLY the 7-port and 9-port test sets are supported with the PNA.
- Works with all 4-port PNA models.
- The E5091A test set has a maximum useful frequency of 11 GHz.
- The E5091A test set Control can be automated using SCPI and COM commands.
- When enabled, a second status bar row appears which indicates the test set that is being controlled and the current switch state.
- Test set path switching occurs just before a channel is triggered. If a channel trigger state is Hold, switching for that channel does not occur.
- PNA sweep speed will be slightly slower when using the E5091A to switch measurement paths.

**Connect and Configure the E5091A**

The E5091A can be connected to any one of the PNA USB ports. When first installed, Windows will automatically launch the "Add New Hardware" wizard. Click Next to install the E5091A test set.

**Note:** See the power handling limitations of the PNA USB ports.

Connect the PNA test ports to the E5091A test ports. Match PNA test port 1 to E5091A test port 1, and so forth.
Selecting ID for E5091A

The PNA can control up to two E5091A test sets. Set the Instrument ID bit switch to 1 or 2. The test sets will then be identified automatically and referred to by the DIP switch setting on the E5091A rear-panel. Change the ID bit switch setting before connecting to the PNA USB.

Power ON

Immediately after power-on, all of the port connection indicator LEDs of the E5091A go ON. Then, after the PNA detects the E5091A, the four LEDs that indicate the connected test ports remain ON. If the PNA is not powered on or if the E5091A is not connected using a USB cable, all of the LEDs stay ON.

How to make E5091A test set Control Settings

Using front-panel HARDKEY [softkey] buttons

1. Press TRACE/CHAN
2. then [Channel]
3. then [More]
4. then [External test set]
5. then [E5091A]

Using a mouse with PNA Menus

1. Click Trace/Chan
2. then Channel
3. then More
4. then External test set
5. then E5091A

No programming commands are available for this feature.
The title of the dialog shows the test set model and ID number of the active test set.

**Select ID**  ID of the test set to be configured. Up to two E5091A test sets can be controlled. Click to change test set ID. [Learn how to set the test set ID](#).

**Enable Test Set Control**  When cleared, port switching and control line settings are disabled. This selection affects all channels using the selected test set.

**Show Test Set Property**  When checked, a second row on the status bar appears which indicates the test set that is being controlled and the current port control selection. For example, the following image shows the status bar when controlling an E5091A test set.

A. Configured channel

B. Port Control settings for E5091A

C. Port Control settings for Z5623A K64

D. Test set Label. E5091A control does not use this field. It is shared between [Interface Control](#) and [External test set Control](#). The two labels are separated by `/`.

Control of the second status bar is completely separate from the first status bar, which is controlled from the [View, Status Bar](#) menu.

**Port Control**  Controls mapping of Physical ports to Logical ports.

- Physical ports are the port numbers that are labeled on the test set front panel. ([see N44xx test sets](#))
- Logical ports are the port numbers that are referred to by most of the PNA application prompts and dialog boxes.
Port Mapping Notes

- Port Control and Control Line settings affect the channel of the active (selected) measurement. These settings will occur as the channel is being measured.
- Correction is NOT turned OFF when port mappings are changed. However, the calibration is NO LONGER VALID!

Control Lines  Specifies the values of individual control lines. These general purpose control lines on the test set front-panel can be used in your test setup. Each button toggles the control line HIGH and LOW. When first opened, the selections reflect the current control lines. See your test set documentation for more information about the control lines.

OK  When clicked, the changes to the dialog box are implemented and the port selections and control values are immediately sent to the specified test sets. The Port Control and Control line settings are stored with other channel data and used when those channels are swept.

Cancel  (or Escape) Changes to the dialog are not implemented and revert to the settings before the dialog box was opened.

Calibrating with the E5091A

The following are a few changes in the way you calibrate the PNA with the E5091A connected:

1. Create the measurements for the channel and configure the Port Control (switching) on the E5091A Test Set Control dialog box. Enable Show Test Set Property.
2. To calibrate, start the Calibration wizard and select a Calibration method (ECAL, SmartCal, Unguided).
3. Select the DUT connectors that are used at the E5091A measurement reference plane.
4. When prompted to connect a standard to a PNA port, instead connect the standard to the E5091A port as indicated on the test set status bar. For example, when the cal wizard prompts to connect the standard to port 1, if the status bar indicates 1 A, the connect the standard to port A of the E5091A.

Last Modified:

- 7-Apr-2009  Added 7/9 port only, and 4-port PNAs
- 4-Sep-2008  Removed legacy content
- 18-Jun-2007  MX added UI
External Multiport Test Set Control

- **Supported Test Sets**
  - **Option 551**
  - **Option 550**
  - **E5091A** (separate topic)

- Procedure
  1. **Connect Test Set**
  2. **Restart as Multiport**
  3. **Optional External Test Set Control Settings**

- **External Test Set Control and other PNA Functions**

---

**Other System Configuration Topics**

**Supported Test Sets**
The list of test sets that provide integrated solutions with the PNA is constantly growing. For a current list of supported multiport test sets, see [www.agilent.com/find/multiport](http://www.agilent.com/find/multiport)

**Option 551**
- **With** Option 551 enabled on your PNA, **any supported multiport test set** (such as the U3042A E12) can be controlled directly from the PNA to make fully integrated measurements at ALL of the available test ports. To understand what test ports are available to source and receive, see the test set documentation.

- **Without** Option 551, basic operation depends on the number of PNA test ports.
  - For a 2-port PNA, configure two available test ports.
  - For a 4-port PNA, configure four available test ports.

**Option 550**
- **With** Option 550 enabled on your PNA, the N44xx test sets and other supported test sets can be controlled directly from the PNA to make fully integrated **4-port** balanced measurements.

- **Without** Option 550, basic operation allows you to configure any **two** of the test ports to make standard S-parameter or receiver measurements. A different pair of ports can be configured for each channel.
<table>
<thead>
<tr>
<th>N44xx Test Set Model</th>
<th>PNA Model (must have front-panel loops)</th>
<th>Frequency Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4419B</td>
<td>E8362B and N5230A Opt 225</td>
<td>10 MHz to 20 GHz</td>
</tr>
<tr>
<td>N4420B</td>
<td>E8363B and N5230A Opt 425</td>
<td>10 MHz to 40 GHz</td>
</tr>
<tr>
<td>N4421B</td>
<td>E8364B and N5230A Opt 525</td>
<td>10 MHz to 50 GHz</td>
</tr>
<tr>
<td>N4421B Opt H67</td>
<td>E8361A</td>
<td>10 MHz to 67 GHz</td>
</tr>
</tbody>
</table>

**Note:** By default, the system logical test ports are mapped as follows:

- **Port 1** - PNA port 1
- **Port 2** - Test Set port 2
- **Port 3** - PNA port 2
- **Port 4** - Test Set port 4
The ports can be remapped using the Port Control Setting.

**Block Diagram of a 2-port PNA and N44xx Test Set**

Click one of the S-parameters to see switch and path changes. Because the test set does NOT contain receivers, measurement speed and calibration can be affected.

![Block Diagram of a 2-port PNA and N44xx Test Set](image)

**Procedure - How to enable full Multiport Capability**

1. **Enable Option 550 or 551.**
2. Connect the test set to the PNA using the documentation that was shipped with the test set.
3. **Restart as Multiport PNA**
4. **Make optional External Test Set Control Settings**

**Connect and Configure the Test Set**

Connect the test set to the PNA using the test set documentation. Most test set documentation can be found at [www.Agilent.com](http://www.Agilent.com).

**Test Set I/O-controlled test sets**

Test sets that are controlled using the **Test Set I/O connector**, have NO return communication capability. The PNA sends commands out the rear panel connector. It is assumed that the test set is responding appropriately. The “Active” LED, located on the test set front panel, should light when the test set is addressed in Multiport Mode or manual operation. When the test set is not in use, the Active LED will be OFF.

**GPIB-controlled test sets**

Connect the test set to the GPIB using one of the following methods:

- If the **PNA will NOT be controlled** by a remote computer using GPIB, then the test set can be connected directly to the PNA GPIB port. The PNA is automatically switched to **System Controller** mode.
- If the **PNA WILL be controlled** by a remote computer using GPIB, then learn how to connect the test set.

**Restart as Multiport PNA**

**How to Enable Multiport capability**

*Note: If Option 550 or Option 551 has not been enabled, the following Multiport Capability menu selection will NOT be available.*

**Using front-panel HARDKEY [softkey] buttons**

1. Press **System**
2. then **Configure**
3. then **Multiport Capability**

**Using a mouse with PNA Menus**

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **Multiport Capability**
Multiport Restart dialog box help

See Also  External Test Set Control and other PNA Functions

After the test set is connected and PNA Option 550 or 551 is enabled, the following settings are used to enable Multiport operation.

Test Set  Select the test set file to load.  Only the files that are appropriate for use on that PNA model are displayed.

To Add a new Test Set file:

1. On the PNA, click File, then Exit to quit the PNA application.
2. Download the Test Set file from:  http://na.tm.agilent.com/multiport/testsets/
3. Save it to the PNA C:/program files/agilent/network analyzer/testsets
4. Start the PNA application.
5. Click Utility, then System, then Configure, then Multiport Capability.
6. In the Multiport Restart dialog, click Restart as multiport PNA with this testset.
7. The new test set should now be visible from the Testset: menu.

Address  Enter the test set address if the test set is connected to GPIB. Connections over the Test Set I/O connector are determined by their sequence.

- **Restart as a standalone PNA.**  The PNA shuts down and restarts as a standard PNA.  If the test set is left connected to the PNA, switch the test set OFF, then back ON to ensure that the test set routes signals to ports 1 and 2 of the PNA.  In this condition, there is more loss in the test paths than without a test set connected.  If the power switch is OFF, there is Significantly more loss in the test paths.

- **Restart as a multiport PNA with this testset.**  The PNA shuts down and restarts as a multiport PNA with the selected test set.

Click OK  The PNA shuts down and restarts in the selected configuration.
To learn how to change port mapping, see Port Control.

Problems
If the PNA cannot find the test set, the following error is displayed on the PNA:

**GPIB ERROR Address xx cannot open VISA session.**

To correct the problem, verify the following:

- The test set is connected to the PNA using one of the methods described above.
- The correct test set address is set.
- The test set is turned ON.

**Important: After the problem has been fixed:**

1. On the External Test Set Control dialog, click Enable Test Set Control.
2. Restart Triggering - click **Sweep, Trigger, Continuous**.
3. The PNA again tries to find the test set.

External Test Set Control Settings
The following External Test Set Control Settings are used to configure Multiport test sets. For the N44xx test sets, the only setting that is necessary is port control.

**How to access the External Test Set Control Settings**

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press TRACE/CHAN</td>
<td>1. Click Trace/Chan</td>
</tr>
<tr>
<td>2. then [Channel]</td>
<td>2. then Channel</td>
</tr>
<tr>
<td>3. then [Hardware Setup]</td>
<td>3. then Hardware Setup</td>
</tr>
<tr>
<td>4. then [External Testset]</td>
<td>4. then External Testset</td>
</tr>
<tr>
<td>5. then [Other External Test Sets]</td>
<td>5. then Other</td>
</tr>
</tbody>
</table>

**Programming Commands**

3719
External Test Set Control dialog box help

Important Notes:

- With Options 500 and 551, **first** use the Multiport Restart dialog to Restart as Multiport PNA with this test set. The test set file is loaded and the test set is enabled automatically.

- When using GPIB to control an external test set, the PNA is automatically put in System Controller mode. It can NOT also be in talker-listener mode. To have the PNA control a GPIB test set AND be controlled by a remote PC, use a USB to GPIB adapter to control the external test set. This does NOT apply for PNA models with a 1.1 GHz CPU board, which has both a GPIB controller port and a talker/listener port.

- See also External Test Set Control and other PNA Functions

Select ID

- For N44xx test sets: the GPIB address
  
- Multiport test sets: either GPIB address or 0 for Test Set I/O controlled test sets.

Enable Test Set Control When cleared, port switching and control line settings are disabled. This selection affects all channels using the selected test set. When checked, the 'Show Test Set Properties' checkbox is also checked automatically.

Load Test Set File For operating **without** Options 500 or 551.

If your Test Set is not visible, see Add a new Test Set.

The selected test set file is loaded.

1. Navigate to the folder: C:/Program Files/Agilent/Network Analyzer/testsets/
2. Select a test set control file.
The title of the dialog shows the model of the test set file that is currently loaded. See a list of supported test sets.

**Show Test Set Properties** This box becomes checked by default when the Enable Test Set Control is checked. When checked, a second row on the status bar appears which indicates the test set that is being controlled and the current port control selection. For example, the following image shows the status bar when controlling an E5091A test set and a Z5623A K64 test set:

![Status Bar Example](image)

A. Configured channel

B. Port Control settings for E5091A

C. Port Control settings for Z5623A K64

D. Test Set Label. This field is shared between Interface Control and External Test Set Control. The two labels are separated by /.

Control of the second status bar is completely separate from the first status bar, which is controlled from the View, Status Bar menu.

**Test Set Label** NOT available with options 550 and 551. Add text to appear on the second status bar when Show Test Set Properties is checked. See image above.

**Port Control** Controls mapping of Physical ports to Logical ports. (Refer to image of dialog box above.)

- Physical ports are the port numbers that are labeled on the test set front panel.
- Logical ports are the port numbers that are referred to by most of the PNA application prompts and dialog boxes.

**Port Mapping Notes**

- Port Control and Control Line settings effect the channel of the active (selected) measurement. These settings occur as the channel is being measured.
- Correction is turned OFF when port mappings are changed.
- After the physical ports are mapped to logical ports, all PNA references to port numbers refer to LOGICAL port numbers. The only exception to this is during calibrations.

**Control Lines** For use with the U30xxA test sets. Specifies the values of individual control lines. These general purpose control lines on the test set front-panel can be used in your test setup. Each button toggles the control line HIGH and LOW. When first opened, the selections reflect the current control lines. See your test set documentation for more information about the control lines.

**OK** When clicked, the changes to the dialog box are implemented and the port selections and control values are immediately sent to the specified test set. The Port Control and Control line settings are stored with other channel data and used when those channels are swept.

**Cancel** (or Escape) Changes to the dialog are not implemented and revert to the settings before the dialog box was opened.
External Test Sets and other PNA Functions

The following features may work differently with a test set connected to the PNA.

Remote Commands

- See SCPI and COM commands for controlling an External Test Set.
- Use *OPT? (SCPI) or NumberofPorts (COM) to query the number of ports for a PNA/External Test set.
- Use logical receiver notation to refer to double-digit ports.
- Use CALC:PAR:DEF:EXT instead of CALC:PAR:DEF.

Interface Control

When both Interface Control and External Test Set Control are configured, the commands on the Interface Control Before Sweep Start tab are sent out before any External Test Set Control commands are executed on that channel. Similarly, commands on the After Sweep End tab are sent after Test Set Control commands.

Calibration

With an External Test Set connected, calibration is performed exactly like a PNA with the following exceptions:

- Correction is turned OFF when port mappings are modified. This also applies to Source Power Cal.
- Beginning with PNA Rev. A.07.50, for TRL Cal, QSOLT, or Unknown Thru calcs with external test sets:
  - Use of a Delta Match Calibration from a User CalSet is NOT required. However, for a 4-port PNA-L a Global Delta Match Cal must be present. The Global Delta Match Cal can only be performed in stand-alone mode.
  - You can NOT perform any of those 3 cal types on JUST a pair of ports that share a test port receiver, such as Port 1 and Port 2 of a 4-port system. You would need to include an additional port in the calibration.
- With an External Test Set connected, you may be required to perform more than 3 THRU connections.
- A test set such as the Z5623A K44 which is used with 4-port PNA models, does not terminate ports that are not currently in the source path. Because a ports load match on this system is not constant for all possible ports it can be paired with, when calibrating more than two total ports it may be necessary to make Thru measurements on more than the usual minimum number of Thru paths for a PNA calibration. The PNA will ensure that multiport calibrations use a sufficient set of Thru paths so that the calibration can correct for those variations in load match on this type of multiport system.
- As with ALL PNA calibrations, when error correction is ON, both forward and reverse sweeps are required for EACH port pair that is corrected, even if only one reflection measurement is displayed. For example, any displayed measurement with full 4-port calibration ON will require 12 measurement sweeps. Learn more.

Source Power Cal
Source power calibration involves adjusting the source so that the power at an output port is flat across a frequency range. Because of additional loss through some of the test set paths, it may NOT be possible to obtain corrected output power because of limitations on the source signal.

During a Source Power Cal, you are prompted when and where to connect the power sensor. When one of the supported test sets are connected, the prompt refers to the PHYSICAL port number, NOT the LOGICAL port number. To help with translating physical to logical port mappings, enable Show Test Set Properties.

Measurements with Shared Receivers

External test sets do not contain receivers. The PNA receivers are always used to measure signals at the external test set ports. Therefore, when a channel contains two measurements that share a PNA test port receiver, additional sweeps are necessary.

For example, to make S34 and S44 measurements in the same channel with correction OFF:

- On a 4-port PNA, only ONE sweep is required using the C (port 3), D (port 4), and R (reference for All receivers).
- On a N44xx system, TWO sweeps are required since both measurements use the B and R2 receivers. See interactive block diagram above.

Create Ratioed and Unratioed Measurements

When using an external test set, it IS possible to create a Ratioed measurement using two logical receivers that share the same physical PNA receiver. However, this measurement data is NOT valid. Invalid measurement traces show all data at -200 dB (in Log mag format). Learn about Logical Receiver Notation

Logical Port References

When an external test set is enabled, all references to PNA port numbers and test set port numbers (except during calibrations) refer to LOGICAL port numbers. Logical ports can be remapped using the Port Control settings. During a calibration, you are prompted to connect standards to physical port numbers.

Balanced Port Mapping

"Logical Ports" is a term that is used with both External Test Sets and balanced measurements. While the concept is the same, they refer to different scenarios. The two can be easily confused when making Balanced measurements with an External Test Set connected. The important principle to remember is the order in which the logical ports for each are mapped:

1. In the External Test Set - Port Control settings dialog, the physical PNA ports and test set ports are mapped to logical ports as noted above.
2. In the Balanced Topology Dialog, the new (step 1) logical ports are mapped again to become Balanced logical ports.
Preset

Instrument Preset will reset Port Control settings to defaults and remove the test set label. All other settings remain. To maintain port control settings and the test set label, create a User Preset.

Instrument State Save and Recall

Instrument State files include Test Set model, Enable and Status bar settings, and Port mappings and DUT control values for each channel.

If an Instrument State recall requires that a test set configuration file be loaded, recall time may be significant. For example, this would occur if a 2-port PNA with attached test set is configured as a 2-port PNA and then recalls a state file which requires 4-port operation.

Recall Cal Sets

If a Cal Set is saved while an external test set is enabled, when the Cal Set is recalled, then the external test set must be enabled or an error message is displayed.

Copy Channel

Copy Channel copies all relevant test set data from the source channel to the target channel.

Applications

No PNA applications are supported with External Test Set Control. These include FCA (opt 083), SMC (opt 082), GCA (opt 086), NFA (opt 029), Pulsed (opt H08).

Print

Port mapping information appears on the Channel Settings Table when printing.

Save sNp Files

To save sNp data with an external test set enabled, click File, Save As, then select Snp File(*.s*p), then complete the Choose ports dialog.

Last modified:
20-Sep-2010  Added links for programming
21-Apr-2010  Updated per RD
23-Nov-2009  Edited Cals for 7.5
12-Sep-2008  Added K44 image and cal note
 4-Sep-2008  Removed legacy content
15-Jan-2008  No App support
19-Nov-2007  Note to Add TS files
 9-Nov-2007  Fixed S13 image and added delta match notes
17-Sep-2007  Added note for PNA-X test set files
 15-Jan-2007  MX Added UI
 10/16/06    Added clarification to opts.
  9/18/6     MQ Many edits for Opt 551
  9/12/06    Added link to programming commands
Display Colors

You can modify the colors that are used to draw various elements on the PNA screen and on a hardcopy print of the display.

See Also
Print Preview

How to modify DISPLAY Colors

These settings can also be accessed from the Preferences dialog box.

- Using front-panel HARDKEY [softkey] buttons
  1. Press **DISPLAY**
  2. then **[More]**
  3. then **[Display Colors]**

- Using a mouse with PNA Menus
  1. Click **Response**
  2. then **Display**
  3. then **Display Colors**

How to modify PRINT Colors

- 1. Press **PRINT**
  2. then **[Print Colors]**

- 1. Click **Utility**
  2. then **Print**
  3. then **Print Colors**
The Display Colors and Print Colors dialog boxes function in exactly the same manner. See Print Preview procedure below.

**Pen**

"Pen" is a term used to describe the various elements. Each pen can have a unique color.

You can change the color of the following pens:

- **Background** - The background color of the inactive windows.
- **New Active Background** - The background color of the active window.
- **Grid** - The inner lines of all grids in all windows, and the grid frame in inactive windows.
- **Active Labels, Grid Frame** - The labels and grid frame colors in the active window. **Note:** when this pen is selected, the current window becomes inactive. Therefore, changes for this pen color will not be visible until OK is pressed.
- **Inactive Window Labels**
- **Failed Trace** - Limit Line failed traces or failure indicators (dots) and the word Fail.
- **The following pens for up to 8 Traces:**
  - Data and Limits
  - Memory trace
  - Markers
  - Memory markers

**About Trace Pens**

'1st Trace' is NOT always Trace1 (Tr1). For example, the first trace in a window might be Tr2 which is drawn with the "1st Trace" pen.

The first 8 traces are drawn with the defined pen colors. The next eight traces reuse the same colors, and so forth. For example, if all traces are numbered sequentially, the 9th and 17th traces are drawn using the same color as the 1st trace.

**Change Color**  Click the button or the color swatch to launch the Change Color dialog.

**Reset Color**  Restores the default color for the selected pen.

**Color Themes**

A theme is a complete set of pens and their colors. The current theme persists until you change it. Themes can also be saved to a file and then later recalled.

- **Save Theme**  Click to save the current set of pens to a file.
- **Recall Theme**  Click to recall and use a saved theme.
• **Reset Theme**  Click to recall the default PNA color theme.

The colors for the following Display elements can NOT be changed: toolbars, softkeys, menus, dialogs and popup messages.

---

**Change Color dialog box help**

To use a basic color, click the color from the 'Basic colors' palette, then click **OK**.

To define and use a custom color:

1. Click **Define Custom Colors**  to open the right side of the dialog.

2. Optionally, pick a Custom color slot to replace. Otherwise, the replacement will occur at the first slot location and continue with subsequent custom color definitions.

3. Click the color pane, or drag the crosshairs, to the location of the custom color.

4. Drag the arrow to the desired saturation level of the custom color.

5. Click **Add to Custom Colors**

6. Continue to define more colors, or click **OK** to close the Color dialog.

After a custom color has been assigned to a PNA pen, the custom color can be changed. The PNA pen color remains unchanged.

---

**Print Preview Procedure**

Use the following procedure to preview your Print Colors on the PNA screen:
1. From the Print Colors dialog, select **Reset Theme** then **Save Theme**. Name the new theme “MyPrintTheme.colors”. This will give you a starting point equal to the default print colors.

2. Launch the Display Colors dialog, select **Recall Theme**, then select “MyPrintTheme.colors”. The display will now show the default print theme.

3. Customize the display colors. You will be previewing how the hardcopy will appear when printed.

4. Save the customized display colors to “MyPrintTheme.colors”.

5. Go to the Print Colors dialog and Recall “MyPrintTheme.colors.”

Last Modified:

- 22-Feb-2010  Added Active background (A.09.20)
- 7-Aug-2009  MX New topic
Mechanical Devices

- **Overview**
  - How to access Mechanical Devices settings
  - Mechanical Devices dialog

---

**Other System Configuration Topics**

**Overview**

**Note:** To prevent premature wear, the PNA does not allow attenuators or other mechanical switches to switch continuously.

These mechanical devices are set for the entire channel. When more than one channel is used, and a mechanical device setting is NOT the same for all channels, only the ACTIVE channel is allowed to sweep. All other channels are **Blocked** - NOT allowed to sweep. Blocked channels will resume sweeping when they are made ACTIVE, or when the conflict is resolved.

Press **TRIGGER** then [**Restart**] to cause ALL channels to sweep once. Then the active channel will resume sweeping continuously.

Before PNA Rev. A.09.20, conflicting channels were put in trigger Hold.

The Mechanical Devices dialog shows the settings of all of the switches and attenuators in the PNA. The settings for all active channels are shown side-by-side for easy comparison. This dialog allows you to determine the settings which would cause mechanical devices to switch between states on consecutive sweeps, potentially leading to device wear-out. It also allows you to determine if the conflict can be resolved to enable continuous sweeps on all channels.

The following are the mechanical devices that are potentially shown in the dialog. These components may not appear in your PNA model:

- Port 1 through Port 4 Bypass Switches
- Port 1 through Port 4 Source Attenuator settings
- Receiver A through Receiver R Attenuator settings
- Port 1 Noise Tuner Switch and Port 2 Noise Receiver Switch
How to access Mechanical Devices settings

<table>
<thead>
<tr>
<th>Using front-panel HARDKEY [softkey] buttons</th>
<th>Using a mouse with PNA Menus</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Press TRACE/CHAN</td>
<td>1. Click Trace/Chan</td>
</tr>
<tr>
<td>2. then [Channel]</td>
<td>2. then Channel</td>
</tr>
<tr>
<td>3. then [Hardware Setup]</td>
<td>3. then Hardware Setup</td>
</tr>
<tr>
<td>4. then [Mechanical Devices]</td>
<td>4. then Mechanical Devices</td>
</tr>
</tbody>
</table>

No programming commands are available for this feature.
**Mechanical Devices** dialog box help

**See Mechanical Devices Overview (scroll up)**

The devices that appear in the table depend on the PNA model and options.

Blue highlighted cells indicate the following:

- The channel is NOT able to sweep. **Blocked** is shown in the top row.
- The highlighted device settings differ from that of the sweeping channels.

To modify entries in the table, click a cell.

When a selection is changed, the new setting is applied immediately.

If **Port Power is coupled**, a dialog prompts if coupling should be turned OFF.

**Not supported**

- **Measurement Class** can NOT be changed from this dialog.
- The dialog does NOT report device settings for **multiport test sets**.
- This dialog does NOT report device settings for **external sources**.

**Trigger** Launches the Trigger dialog box.

**OK** Closes the dialog box.

**Cancel** Does not apply changes that were made, and closes the dialog box.

---

Last Modified:

- **23-Feb-2010** Added Channel Blocking (A.09.20)
- **31-Jul-2009** MX New topic
Power Limit and Power Offset

- **Overview**
- **How to access Power Limit and Power Offset settings**

### Other System Topics

#### Overview

**Power Limit (Global scope)**

Global power limit sets a maximum source power level for individual PNA ports. This value limits port power for all channels and all applications. Power levels that attempt to exceed the power limit is clipped at the limit.

**Notes**

- The power limit can NOT be set for power levels which are below the power level that is required by the PNA to achieve phase lock - approximately -30 dBm.
- Because [Fast Sweep mode](#) allows power spiking, it is NOT allowed when a power limit is enabled.
- Components that are added to the RF path are accounted for by entering their loss (negative) or gain (positive) in the [Power Offset](#) section of the dialog box.
- [PNA Applications](#) may change RF path components. For example, IMD for Converters may change the combiner path and add an amplifier for LO input. Compensation is NOT made for these changes and port power may exceed the power limit or port power may be clipped unnecessarily.
- Power limiting does NOT clip power spikes that may occur during [frequency band crossings](#).
- [External test set ports](#) are also included for power limiting.

**Power Offset (Channel scope)**

Power Offset provides a method of compensating port power for added attenuation or amplification in the source path. The result is that power at the specified port, all dialogs, and annotation, reflects the added components.
How to access the Offsets and Limits settings
Also accessed through the PNA Preferences dialog.

Using front-panel HARDKEY [softkey] buttons
1. Press POWER
2. then [Power and Attenuators]
3. then [Offsets and Limits]

Using a mouse with PNA Menus
1. Click Stimulus
2. then Power
3. then Power and Attenuators
4. then Offsets and Limits

Offsets and Limits dialog box help
Click a WHITE cell to change values. Shaded cells can NOT be changed.
Remote commands can be sent to lock and unlock the dialog box (UI) settings.

Power Limit
Limits the source power at each PNA port for ALL channels. Use this feature to protect DUTs that are sensitive to overpowering at the input. Power levels that exceed the limit at the specified port are clipped at the limit and an error message is displayed on the screen.
The Power Limit settings survive Instrument Preset. When an Instrument State is recalled, the current Power Limit settings are applied to the recalled state.
To learn more, see Power Limit Overview (scroll up).

State / Limit
- **ON** - Power is limited to the adjacent value at the specified source port.
- **OFF** - Power is NOT limited to this value, but to the maximum power of the PNA source.

For PNA models with a second internal source, the **Port 1 Src2** Power Limit setting is NEVER available. Make the setting at the standard **Port 1**.
**Power Offset**

Power Offset provides a method of compensating port power for added attenuation or amplification in the source path. The result is that power at the specified port, all dialogs, and annotation, reflects the added components.

- For amplification, use positive offset.
- For attenuation, use negative offset.

Then optionally change the Source Power or Port Power values so that the following equation reflects your requirement:

\[
\text{Source Power} + \text{Power Offset} = \text{Port Power}
\]

**Source Cal ON / OFF** - The power offset value is used as the default Offset in Source Power Calibration. After a Source Power Calibration is performed, the minimum (offset+correction) value from the Source Power Cal is reported in this dialog and is NOT available to be changed.

**Notes**

- Power Offset can be used with Power Sweeps. When a power sweep is enabled, the Start and Stop power levels are reported in this dialog.
- When port power offsets are used, port powers are automatically uncoupled. Port powers may not be coupled again until all port offsets are zero.

**OK** Closes the dialog box.
Setting System Impedance

The system impedance can be changed for measuring devices with an impedance other than 50 ohms, such as waveguide devices. The PNA mathematically transforms and displays the measurement data as though the PNA ports were the specified impedance value. Physically, the test ports are always about 50 ohms.

How to change the System Impedance

Using front-panel HARDKEY [softkey] buttons

1. Press **SYSTEM**
2. then **Configure**
3. then **System Z0**

Using a mouse with PNA Menus

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **System Z0**

**System Z0** dialog box help

Allows you to change the system impedance (default setting is 50 ohms).

**Z0**  Displays the current system impedance.

**For 75 ohm devices:**

1. Change the system Z0 to 75 ohms.
2. Connect minimum loss pads (75 ohm impedance) between the analyzer and the DUT to minimize the physical mismatch.
3. Perform a calibration with 75 ohm calibration standards.

**For waveguide devices**

When performing an **Unguided Cal** (NOT required for Guided Calibration):

1. Change the system Z0 to 1 ohm.
2. Perform a calibration with the appropriate waveguide standards.
Synchronize PNA with an External (PSG) Source

Beginning with PNA Rev. 7.22, the PNA External Source Control feature can be used to automatically control external sources. However, this feature requires certain PNA options. Learn more.

Many PNA measurements require the use of at least one external source. For example, when measuring the insertion loss of a mixer, the LO must be swept at the same time as the RF input. This requires the PNA and external source to be synchronized.

The following procedure shows how to manually synchronize the PNA with an Agilent PSG Source. Although the settings will be different, the concept is useful with other sources.

**Hardware configuration**

- Connect the PNA and PSG Time Base ([PNA 10 MHz OUT](#) to PSG 10 MHz IN)

**PNA-L, E866xB Models**

Connect the PSG and [PNA Trigger Connectors](#) as follows:

- PNA Trigger IN to PSG Trigger OUT
- PNA Trigger OUT to PSG Trigger IN

**PNA-X Models**

Connect either pair (1 or 2) of the [AUX Trigger I/O connectors](#) as follows:

- PNA AUX Trig IN to PSG Trigger OUT
- PNA AUX Trig OUT to PSG Trigger IN

Learn more about the AUX Trigger capabilities.

**PNA Settings**

- **Number of points**: Same as PSG
- **Frequency span**: Does NOT have to be the same as PSG

**PNA Trigger Settings**

- Trigger Source:
  - PNA-L and E866xB models: [External](#)
  - PNA-X: [Internal, Manual](#)
• Trigger Scope: **Channel**

• Channel Trigger State: **Same as PSG Sweep Repeat setting** *(Continuous or Single)*

• Point Sweep: **Checked**

### External / Auxiliary Trigger Settings

<table>
<thead>
<tr>
<th>Input</th>
<th>PNA-L and E866xB</th>
<th>PNA-X</th>
</tr>
</thead>
<tbody>
<tr>
<td>Where settings are made:</td>
<td>External Tab</td>
<td>Aux Trigger Tab</td>
</tr>
<tr>
<td>Level / Edge:</td>
<td>Edge</td>
<td>Same as PSG (Hi or Low)</td>
</tr>
<tr>
<td>Accept Trigger Before Armed:</td>
<td>Checked</td>
<td>N/A</td>
</tr>
<tr>
<td>Handshake</td>
<td>N/A</td>
<td>Checked</td>
</tr>
</tbody>
</table>

### Output

<table>
<thead>
<tr>
<th>Where settings are made:</th>
<th>I/O2 Trig Out Tab</th>
<th>Aux Trigger Tab</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable</td>
<td>Checked</td>
<td>Checked</td>
</tr>
<tr>
<td>Polarity:</td>
<td>Same as PSG</td>
<td>Same as PSG</td>
</tr>
<tr>
<td>Position:</td>
<td>After</td>
<td>After</td>
</tr>
<tr>
<td>Per Point</td>
<td>Checked</td>
<td>Checked</td>
</tr>
</tbody>
</table>

### PSG Settings

• Number of points: **Same as PNA**

• Sweep: **Step** or **List**

• Sweep Trig: **Free Run**

• Sweep Repeat: **Same as PNA Channel Trigger State** *(Continuous or Single)*

• Sweep Direction: **UP**

• Point Trig: **Ext**

• Manual Mode: **OFF**

• Trigger In / Out Polarity: **Same as PNA**
What is Happening?

The following is a flow diagram showing the handshake / synchronization process between the PNA and an External Source.

Text Description

1. After the measurement setup is complete, both instruments wait on the first data point of a measurement sweep. Both instruments are configured for Continuous or Single sweep.

2. (see note below) A trigger signal from the source starts the measurement. This is usually accomplished by a key press on the source front panel.

3. PNA data acquisition (measurement) starts, and then stops AFTER the first data point acquisition.

4. The PNA sends a trigger signal out to the source telling it to move to the next frequency data point. This signal can optionally be sent BEFORE data acquisition if required by your application.

5. The external source and PNA move to the next data point. The source usually takes longer than the PNA.

6. The source sends the Ready for Trigger signal to the PNA.

- **PNA-L and E866xB models** - If the source arrives first, the Accept Trigger Before Armed setting is used to accept the trigger signal even if the PNA is not yet ready to start acquisition.

- **PNA-X using AUX Triggering** - If enabled, the PNA waits indefinitely for a trigger signal from the source. Although AUX triggering does NOT have the Accept Trigger Before Armed setting, the Ready for Trigger signal is latched and has the same effect.

**Step 2 Note**  PNA-X (Aux TRIG IN) The PNA looks for a level trigger at the start of each sweep, and an edge thereafter. This assumes that the external source ready line will remain in the ready state (high or low) until triggered (step 4) and will then transition to the NOT ready state while moving to the next frequency, and then transition again to the ready state.
How do you know when the PNA and PSG are in synch?

The measurement results are the ultimate test of whether the source and PNA are synchronized. However, it is possible to see the PSG and PNA sweeping at exactly the same time.

First, lower the PNA IFBandwidth or increase the sweep time so that the sweep is slow enough to watch the sweep indicator moving across the PNA screen. At the same time, watch the PSG "progress bar" as it moves through the entire sweep.

If the PNA is stopped in the middle of a sweep, then re-triggered, it returns to the first data point. The PSG continues from where it stopped. Therefore, to re-synch the two instruments, the PSG needs to return to the first data point. There are a number of ways to do this. One way is to press the PSG Manual button to ON, then OFF. Then trigger a new sweep.

To trigger a sweep

- **Single** Trigger mode: Both the PNA and PSG Single (trigger) buttons must be pressed (in any order) for each trigger.

- **Continuous** Trigger mode: First, reset the PSG to the first data point, then press the PNA Continuous (trigger) button.

Maintaining Synchronization

In general, the above setup should start the two instruments sweeping simultaneously. However, any interaction with the PNA could cause the PNA sweep to abort or delay, in which case the two instruments will be out of sync. To avoid this, you can use the PNA Interface Control feature to send an ABORT to the external device after each sweep.

When the PNA ends a sweep, it sends an ABORT to stop the source. A trigger signal is then sent, either Continuous (automatically) or Single (manual). In either case, both instruments start sweeping in synch.

This takes more time to sweep, but maintains synchronization.

For example, to use this feature with Agilent’s PSG source, you would add the following:

On the “After Sweep End” tab, type:

```
24 :ABORT
```

Where 24 is the GPIB address of the source.

Last Modified:

- 21-Dec-2010  Fixed source model number
- 24-Aug-2010  Fixed PSG trigger labels
- 11-Feb-2008  Updated note
- 1-Jun-2007   Added 7.22 update
- 1-Jan-2007   MX Updated for PNA-X
PNA Application Notes

The following links require an Internet connection.

Note: Check out the multimedia PNA Demo presentations, including 'Network Analyzer Basics'.

Calibrations

Improving Measurement and Calibration Accuracy Using the Frequency Converter Application (5988-9642EN)
On-Wafer Calibration Using a 4-port, 20 GHz PNA-L Network Analyzer (N5230A Option 240/245) (5989-2287EN)
Specifying Calibration Standards and Kits for Agilent Vector Network Analyzers (5989-4840EN)

ECal

Agilent Electronic vs. Mechanical Calibration Kits: Calibration Methods and Accuracy (5988-9477EN)
User Characterization: Electronic Calibration Feature Allows Users to Customize to Specific Needs (5988-9478EN)

Embedding / De-embedding

De-embedding and Embedding S-Parameter Networks Using a Vector Network Analyzer (5980-2784EN)

Amplifier Measurements

High-power measurements using the PNA (5989-1349EN)
New High Power Amplifier Measurements Using NVNA (1408-19)
Amplifier Linear and Gain Measurements (5988-8644EN)
Amplifier Swept-Harmonic Measurements (5988-9473EN)
Amplifier and CW Swept Intermodulation-Distortion Measurements (5988-9474EN)
Making Accurate IMD Measurements with the PNA-X Network Analyzer (5989-7265EN)

Antenna Measurements

Triggering PNA Microwave Network Analyzers for Antenna Measurements (5988-9518EN)
New Network Analyzer Methodologies in Antenna/RCS Measurements (5989-1937EN)
Pulsed Antenna Measurements Using PNA Network Analyzers (5989-0221EN)

Balanced Measurements (Although the following refer to the ENA, they are also relevant to the PNA.)
On-wafer Balanced Component Measurement with the Cascade Microtech Probing System (5988-5886EN)
Network De-embedding/Embedding and Balanced Measurement (5988-4923EN)

Mixer Measurements

Mixer Transmission Measurements Using the Frequency Conversion Application (5988-8642EN)
Mixer Conversion-Loss and Group Delay Measurement Techniques and Comparisons (5988-9619EN)
Comparison of Mixer Characterization using New Vector Characterization Techniques (5988-7827EN)
Novel Method for Vector Mixer Characterization and Mixer Test System Vector Error Correction (5988-7826EN)
Measuring Absolute Group Delay of Multistage Converters Using PNA Microwave Network Analyzers (5989-
Measuring Group Delay of Frequency Converters with Embedded Local Oscillators (5989-7385EN)

Pulsed Measurements
New Active-Device Characterization in Pulsed Operation Using the PNA-X (1408-21)
Pulsed-RF S-Parameter Measurements Using Wideband and Narrowband Detection (1408-12)
Accurate Pulsed Measurements (5989-0563EN)
Pulsed Antenna Measurements Using PNA Network Analyzers (5989-0221EN)

Other Measurements
New High-Accuracy Noise Figure Measurements Using the PNA-X (1408-20)
Time Domain Analysis
Using the PNA Series to Analyze Lightwave Components (5989-3385EN)
Using the PNA for Banded Millimeter-Wave Measurements (5989-4098EN)
MM-Wave Network Analyzers: Analysis of Cable Length on VNA System Performance (5989-1941EN)
Basics of Measuring the Dielectric Properties of Materials (5989-2589EN)

Automation
Connectivity Advances for Component Manufacturers (5980-2782EN)
Introduction to Application Development using the PNA (5980-2666EN)
The ‘Need for Speed’ in Component Manufacturing Test (5980-2783EN)

Last Modified:

20-May-2011 Added new pulse
7-Mar-2011 Added new app notes
22-Oct-2010 Added Noise Figure
14-Apr-2008 Added IMD with PNA-X.
Network Analyzer Basics

This self-paced two hour video discusses the basic concepts of Network Analysis. The files are installed and should work on older PNA models. If the PNA link does not work, then use the internet link, which requires an internet connection.

- **From the PNA:** Proceed with Network Analyzer Basics.
- **From the Internet:** [http://wireless.agilent.com/networkanalyzers/pnademo.htm](http://wireless.agilent.com/networkanalyzers/pnademo.htm) in both streaming and downloadable format.

Last modified:

10/18/06 Added link to pnademo.
Connector Care

Proper connector care is critical for accurate and repeatable measurements. The following information will help you preserve the precision and extend the life of your connectors - saving both time and money.

- Connector Care Quick Reference Guide
- Connector Cleaning Supplies
- Safety Reminders
- About Connectors
- Gaging Fundamentals
- Connector Care Procedures

See also mmWave Connector Care at http://na.tm.agilent.com/pna/connectorcare/Connector_Care.htm

Preventing Test Port Connector Damage

<table>
<thead>
<tr>
<th>Handling and Storing Connectors</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Do</strong></td>
</tr>
<tr>
<td>Keep connectors clean</td>
</tr>
<tr>
<td>Protect connectors with plastic end caps</td>
</tr>
<tr>
<td>Keep connector temperature same as analyzer</td>
</tr>
<tr>
<td><strong>Do Not</strong></td>
</tr>
<tr>
<td>Touch mating-plane surfaces</td>
</tr>
<tr>
<td>Set connectors contact-end down</td>
</tr>
<tr>
<td>Store connectors loose in box or drawer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Visual Inspection</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Do</strong></td>
</tr>
<tr>
<td>Inspect connectors with magnifying glass.</td>
</tr>
<tr>
<td>Look for metal debris, deep scratches or dents</td>
</tr>
<tr>
<td><strong>Do Not</strong></td>
</tr>
<tr>
<td>Use a connector with a bent or broken center conductor</td>
</tr>
<tr>
<td>Use a connector with deformed threads</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cleaning Connectors</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Do</strong></td>
</tr>
<tr>
<td>Clean surfaces first with clean, dry compressed air</td>
</tr>
<tr>
<td>Use lint-free swab or brush</td>
</tr>
<tr>
<td>Use minimum amount of alcohol</td>
</tr>
<tr>
<td>Clean outer conductor mating surface and threads</td>
</tr>
<tr>
<td><strong>Do Not</strong></td>
</tr>
<tr>
<td>Use high pressure air (&gt;60 psi)</td>
</tr>
<tr>
<td>Use any abrasives</td>
</tr>
<tr>
<td>Allow alcohol into connector support beads</td>
</tr>
<tr>
<td>Apply lateral force to center conductor</td>
</tr>
</tbody>
</table>
### Gaging Connectors

<table>
<thead>
<tr>
<th>Do</th>
<th>Do Not</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inspect and clean gage, gage master and device tested</td>
<td>Use an out of specification connector</td>
</tr>
<tr>
<td>Use correct torque wrench</td>
<td>Hold connector gage by the dial</td>
</tr>
<tr>
<td>zero gage before use</td>
<td></td>
</tr>
<tr>
<td>Use multiple measurements and keep record of readings</td>
<td></td>
</tr>
</tbody>
</table>

### Making Connections

<table>
<thead>
<tr>
<th>Do</th>
<th>Do Not</th>
</tr>
</thead>
<tbody>
<tr>
<td>Align connectors first</td>
<td>Cross thread the connection</td>
</tr>
<tr>
<td>Rotate only the connector nut</td>
<td>Twist connector body to make connection</td>
</tr>
<tr>
<td>Use correct torque wrench</td>
<td>Mate different connector types</td>
</tr>
</tbody>
</table>

### Connector Care and Cleaning Supplies

<table>
<thead>
<tr>
<th>Description</th>
<th>Web Site</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lint Free Cloths- Air dusters</td>
<td><a href="http://www.ccrwebstore.com">http://www.ccrwebstore.com</a></td>
</tr>
<tr>
<td>Isopropyl</td>
<td><a href="http://www.techspray.com">http://www.techspray.com</a></td>
</tr>
<tr>
<td>Nitrilite Gloves and Finger Cots</td>
<td><a href="http://www.techni-tool.com">http://www.techni-tool.com</a></td>
</tr>
</tbody>
</table>

### Safety Reminders

**When cleaning connectors:**

- Always use protective eyewear when using compressed air or nitrogen.
- Keep isopropyl alcohol away from heat, sparks and flame. Use with adequate ventilation. Avoid contact with eyes, skin and clothing.
- Avoid electrostatic discharge (ESD). Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.
- Cleaning connectors with alcohol shall only be done with the instruments power cord removed, and in a well-ventilated area. Allow all residual alcohol moisture to evaporate, and the fumes to dissipate prior to energizing the instrument.

### About Connectors
Connector Service Life

Even though calibration standards, cables, and test set connectors are designed and manufactured to the highest standards, all connectors have a limited service life. This means that connectors can become defective due to wear during normal use. For best results, all connectors should be inspected and maintained to maximize their service life.

Visual Inspection should be performed each time a connection is made. Metal particles from connector threads often find their way onto the mating surface when a connection is made or disconnected. See Inspection procedure.

Cleaning the dirt and contamination from the connector mating plane surfaces and threads can extend the service life of the connector and improve the quality of your calibration and measurements. See Cleaning procedure.

Gaging connectors not only provides assurance of proper mechanical tolerances, and thus connector performance, but also indicate situations where the potential for damage to another connector may exist. See Gaging procedure.

Proper connector care and connection techniques yield:

- Longer Service Life
- Higher Performance
- Better Repeatability

Connector Grades and Performance

The three connector grades (levels of quality) for the popular connector families are listed below. Some specialized types may not have all three grades.

- Production grade connectors are the lowest grade and the least expensive. It is the connector grade most commonly used on the typical device under test (DUT). It has the lowest performance of all connectors due to its loose tolerances. This means that production grade connectors should always be carefully inspected before making a connection to the analyzer. Some production grade connectors are not intended to mate with metrology grade connectors.

- Instrument grade is the middle grade of connectors. It is mainly used in and with test instruments, most cables and adapters, and some calibration standards. It provides long life with good performance and tighter tolerances. It may have a dielectric supported interface and therefore may not exhibit the excellent match of a metrology grade connector.

- Metrology grade connectors have the highest performance and the highest cost of all connector grades. This grade is used on calibration standards, verification standards, and precision adapters. Because it is a high precision connector, it can withstand many connections and disconnections and, thus, has the longest life of all connector grades. This connector grade has the closest material and geometric specifications. Pin diameter and pin depth are very closely specified. Metrology grade uses an air dielectric interface and a slotless female contact which provide the highest performance and traceability.
**Note:** In general, Metrology grade connectors should not be mated with Production grade connectors.

**Adapters as Connector Savers**

Make sure to use a high quality (Instrument grade or better) adapter when adapting a different connector type to the analyzer test ports. It is a good idea to use an adapter even when the device under test is the same connector type as the analyzer test ports. In both cases, it will help extend service life, and protect the test ports from damage and costly repair.

The adapter must be fully inspected before connecting it to the analyzer test port and inspected and cleaned frequently thereafter. Because calibration standards are connected to the adapter, the adapter should be the highest quality to provide acceptable RF performance and minimize the effects of mismatch.

**Connector Mating Plane Surfaces**

An important concept in RF and microwave measurements is the reference plane. For a network analyzer, this is the surface that all measurements are referenced to. At calibration, the reference plane is defined as the plane where the mating plane surfaces of the measurement port and the calibration standards meet. Good connections (and calibrations) depend on perfectly flat contact between connectors at all points on the mating plane surfaces (as shown in the following graphic).

![Mating Plane Surfaces](image)

**Gaging Fundamentals**

Connector gages are important tools used to measure center conductor pin depth in connectors. Connector pin depth, measured in terms of recession or protrusion, is generally the distance between the mating plane and the end of the center conductor, or the shoulder of the center conductor for a stepped male pin.

**Typical Connector Gage**
Recession and Protrusion

Pin depth is negative (recession) if the center conductor is recessed below the outer conductor mating plane, usually referred to as the "reference plane". Pin depth is positive (protrusion) if the center conductor projects forward from the connector reference plane.

Pin Depth

1. Recession of female contact
2. Recession of male pin shoulder

Difference with Type-N Connectors
Type-N connectors have the mating plane of the center conductors offset from the connector reference plane. In this case the zero setting “gage masters” generally offset the nominal distance between the center conductor mating plane and the connector reference plane.

**When to Gage Connectors**

- Before using a connector or adapter the first time.
- When visual inspection or electrical performance suggests the connector interface may be out of range.
- After every 100 connections, depending on use.

**Connector Gage Accuracy**

Connector gages (those included with calibration and verification kits), are capable of performing coarse measurements only. This is due to the repeatability uncertainties associated with the measurement. It is important to recognize that test port connectors and calibration standards have mechanical specifications that are extremely precise. Only special gaging processes and electrical testing (performed in a calibration lab) can accurately verify the mechanical characteristics of these devices. The pin depth specifications in the Agilent calibration kit manuals provide a compromise between the pin depth accuracy required, and the accuracy of the gages. The gages shipped with calibration and verification kits allow you to measure connector pin depth and avoid damage from out-of-specification connectors.

**Note:** Before gaging any connector, the mechanical specifications provided with that connector or device should be checked.

**To Gage Connectors**

1. Wear a grounded wrist strap (having a 1 MΩ series resistor).
2. Select proper gage for device under test (DUT).
3. Inspect and clean gage, gage master, and DUT.
4. Zero the connector gage.
   a. While holding gage by the barrel, carefully connect gage master to gage. Finger-tighten connector nut only.
   b. Use proper torque wrench to make final connection. If needed, use additional wrench to prevent gage master (body) from turning. Gently tap the barrel to settle the gage.
   c. The gage pointer should line up exactly with the zero mark on gage. If not, adjust “zero set” knob until gage pointer reads zero. On gages having a dial lock screw and a movable dial, loosen the dial lock screw and move the dial until the gage pointer reads zero. Gages should be zeroed before each set of measurements to make sure zero setting has not changed.
   d. Remove gage master.
5. Gage the device under test.
   a. While holding gage by the barrel, carefully connect DUT to gage. Finger-tighten connector nut only.
   b. Use proper torque wrench to make final connection and, if needed, use additional wrench to prevent DUT (body) from turning. Gently tap the barrel to settle the gage.
c. Read gage indicator dial for recession or protrusion and compare reading with device specifications.

**Caution:** If the gage indicates excessive protrusion or recession, the connector should be marked for disposal or sent out for repair.

6. For maximum accuracy, measure the device a minimum of three times and take an average of the readings. After each measurement, rotate the gage a quarter-turn to reduce measurement variations.

7. If there is doubt about measurement accuracy, be sure the temperatures of the parts have stabilized. Then perform the cleaning, zeroing, and measuring procedure again.

**Connector Care Procedures**

- **Inspecting Connectors**
- **Cleaning Connectors**
- **Making Connections**
- **Using a Torque Wrench**
- **Handling and Storing Connectors**

**To Inspect Connectors**

Wear a grounded wrist strap (having a 1 MΩ series resistor).

Use a magnifying glass (≥10X) and inspect connector for the following:

- Badly worn plating or deep scratches
- Deformed threads
- Metal particles on threads and mating plane surfaces
- Bent, broken, or mis-aligned center conductors
- Poor connector nut rotation

**Caution:** A damaged or out-of-specification device can destroy a good connector attached to it even on the first connection. Any connector with an obvious defect should be marked for disposal or sent out for repair.

**To Clean Connectors**

1. Wear a grounded wrist strap (having a 1 MΩ series resistor).

2. Use clean, low-pressure air to remove loose particles from mating plane surfaces and threads. Inspect connector thoroughly. If additional cleaning is required, continue with the following steps.

4. Clean contamination and debris from mating plane surfaces and threads. When cleaning interior surfaces, avoid exerting pressure on center conductor and keep swab fibers from getting trapped in the female center conductor.

5. Let alcohol evaporate—then use compressed air to blow surfaces clean.

6. Inspect connector. Make sure no particles or residue remains.

7. If defects are still visible after cleaning, the connector itself may be damaged and should not be used. Determine the cause of damage before making further connections.

To Make Connections

1. Wear a grounded wrist strap (having a 1 MΩ series resistor).

2. Inspect, clean, and gage connectors. All connectors must be undamaged, clean, and within mechanical specification.

3. Carefully align center axis of both devices. The center conductor pin—from the male connector—must slip concentrically into the contact finger of the female connector.

4. Carefully push the connectors straight together so they can engage smoothly. Rotate the connector nut (not the device itself) until finger-tight, being careful not to cross the threads.
5. Use a torque wrench to make final connection. Tighten until the "break" point of the torque wrench is reached. Do not push beyond initial break point. Use additional wrench, if needed, to prevent device body from turning.

To Separate a Connection

1. Support the devices to avoid any twisting, rocking or bending force on either connector.
2. Use an open-end wrench to prevent the device body from turning.
3. Use another open-end wrench to loosen the connector nut.
4. Complete the disconnection by hand, turning only the connector nut.
5. Pull the connectors straight apart.

To Use a Torque Wrench

1. Make sure torque wrench is set to the correct torque setting.
2. Position torque wrench and a second wrench (to hold device or cable) within 90° of each other before applying force. Make sure to support the devices to avoid putting stress on the connectors.
3. Hold torque wrench lightly at the end of handle—then apply force perpendicular to the torque wrench handle. Tighten until the "break" point of the torque wrench is reached. Do not push beyond initial break point.

**TORQUING DIRECTION**

**STOP WHEN HANDLE BEGINS TO YIELD**

**To Handle and Store Connectors**

- Install protective end caps when connectors are not in use.
- Never store connectors, airlines, or calibration standards loose in a box. This is a common cause of connector damage.
- Keep connector temperature the same as analyzer. Holding the connector in your hand or cleaning connector with compressed air can significantly change the temperature. Wait for connector temperature to stabilize before using in calibration or measurements.
- Do not touch mating plane surfaces. Natural skin oils and microscopic particles of dirt are difficult to remove from these surfaces.
- Do not set connectors contact-end down on a hard surface. The plating and mating plane surfaces can be damaged if the interface comes in contact with any hard surface.
- Wear a grounded wrist strap and work on a grounded, conductive table mat. This helps protect the analyzer and devices from electrostatic discharge (ESD).
Electrostatic Discharge (ESD) Protection

Protection against electrostatic discharge (ESD) is essential while removing or connecting cables to the network analyzer. Static electricity can build up on your body and can easily damage sensitive internal circuit elements when discharged. Static discharges too small to be felt can cause permanent damage. To prevent damage to the instrument:

- **Always** have a grounded, conductive table mat in front of your test equipment.
- **Always** wear a grounded wrist strap, connected to a grounded conductive table mat, having a 1 MΩ resistor in series with it, when making test setup connections.
- **Always** wear a heel strap when working in an area with a conductive floor. If you are uncertain about the conductivity of your floor, wear a heel strap.
- **Always** ground yourself before you clean, inspect, or make a connection to a static-sensitive device or test port. You can, for example, grasp the grounded outer shell of the test port or cable connector briefly.
- **Always** ground the center conductor of a test cable before making a connection to the analyzer test port or other static-sensitive device. This can be done as follows:
  1. Connect a short (from your calibration kit) to one end of the cable to short the center conductor to the outer conductor.
  2. While wearing a grounded wrist strap, grasp the outer shell of the cable connector.
  3. Connect the other end of the cable to the test port and remove the short from the cable.

See **Analyzer Accessories** for ESD part numbers.
**Absolute Output Power**

An absolute output-power measurement displays absolute power versus frequency.

- **What is Absolute Output Power?**
- **Why Measure Absolute Output Power?**
- **Accuracy Considerations**
- **How to Measure Absolute Output Power**

### What is Absolute Output Power?

An absolute-output power measurement displays the power present at the analyzer's input port. This power is absolute—it is not referenced (ratioed) to the incident or source power. In the log mag format, values associated with the grid's vertical axis are in units of dBm, which is the power measured in reference to 1 mW.

- 0 dBm = 1 mW
- -10 dBm = 100 μW
- +10 dBm = 10 mW

In the linear mag format, values associated with the grid's vertical axis are in units of watts (W).

### Why Measure Absolute Output Power?

Absolute output power is measured when the amplifier's output must be quantified as absolute power rather than a ratioed relative power measurement. For example, during a gain compression measurement, it is typical to also measure absolute output power. This shows the absolute power out of the amplifier where 1-dB compression occurs.

### Accuracy Considerations

The output power of the amplifier should be sufficiently attenuated if necessary. Too much output power could:

- Damage the analyzer receiver
- Exceed the input compression level of the analyzer receiver, resulting in inaccurate measurements.

Attenuation of the amplifier's output power can be accomplished using either attenuators or couplers.

The amplifier may respond very differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.

### How to Measure Absolute Power
Do the following to measure absolute output power:

1. Preset the analyzer.
2. Select an unratioed power measurement (receiver B).
3. Set the analyzer’s source power to 0 dBm.
4. Select an external attenuator (if needed) so the amplifier’s output power will be sufficiently attenuated to avoid causing receiver compression or damage to the analyzer’s port-2.
5. Connect the amplifier as shown in the following graphic, and provide the dc bias.

6. Select the analyzer settings for your amplifier under test.
7. Remove the amplifier and connect the measurement ports together. Store the data to memory. Be sure to include the attenuator and cables in the test setup if they will be used when measuring the amplifier.
8. Save the instrument state to memory.
9. Reconnect the amplifier.
10. Select the data math function Data/Memory.
11. Scale the displayed measurement for optimum viewing and use a marker to measure the absolute output-power at a desired frequency.
12. Print or save the data to a disk.
AM-PM Conversion

The AM-PM conversion of an amplifier is a measure of the amount of undesired phase deviation (PM) that is caused by amplitude variations (AM) inherent in the system.

- What Is AM-PM Conversion?
- Why Measure AM-PM Conversion
- Accuracy Considerations
- How to Measure AM-PM Conversion

What Is AM-PM Conversion?

AM-to-PM conversion measures the amount of undesired phase deviation (PM) that is caused by amplitude variations (AM) of the system. For example, unwanted phase deviation (PM) in a communications system can be caused by:

Unintentional amplitude variations (AM)

- Power supply ripple
- Thermal drift
- Multipath fading

Intentional modulation of signal amplitude

- QAM
- Burst modulation

AM-to-PM conversion is usually defined as the change in output phase for a 1-dB increment in the power-sweep applied to the amplifier's input (i.e. at the 1 dB gain compression point). It is expressed in degrees-per-dB (°/dB). An ideal amplifier would have no interaction between its phase response and the power level of the input signal.
Why Measure AM-PM Conversion

AM-to-PM conversion is a critical parameter in systems where phase (angular) modulation is used, such as:

- FM
- QPSK
- 16QAM

It is a critical parameter because undesired phase deviation (PM) causes analog signal degradation, or increased bit-error rates (BER) in digital communication systems. While it is easy to measure the BER of a digital communication system, this measurement alone does not help you understand the underlying causes of bit errors. AM-to-PM conversion is one of the fundamental contributors to BER, and therefore it is important to quantify this parameter in communication systems.

Refer to the I/Q diagram below for the following discussion on how AM-to-PM conversion can cause bit errors.
The desirable state change is from the small solid vector to the large solid vector.

With AM-to-PM conversion, the large vector may actually end up as shown with the dotted line. This is due to phase shift that results from a change in the input power level.

For a 64QAM signal as shown (only one quadrant is drawn), we see that the noise circles that surround each state would actually overlap, which means that statistically, some bit errors would occur.

**Accuracy Considerations**

With this method of measuring AM-to-PM conversion, the modulation frequency is approximately the inverse of the sweep time. Even with the fastest power sweep available on most network analyzers, the modulation frequency ends up being fairly low (typically less than 10 Hz). This could cause a slight temperature change as the sweep progresses, especially if the amplifier has low thermal mass, typical of an unpackaged device. Results using this method could differ slightly if the nonlinear behavior of an amplifier is extremely sensitive to thermal changes. (The PNA series analyzers can make power sweeps <1 ms.)

- The amplifier may respond very differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.
- The output power of the amplifier should be sufficiently attenuated if necessary. Too much output power could:
  - damage the analyzer receiver
  - exceed the input compression level of the analyzer receiver, resulting in inaccurate measurements
- **Attenuation** of the amplifier’s output power can be accomplished using:
  - Attenuators
  - Couplers
- The frequency-response effects of the attenuators and couplers must be accounted for during calibration since they are part of the test system. Proper error-correction techniques can reduce these effects.
- The frequency response is the dominant error in an AM-to-PM conversion measurement setup. Performing a
thru-response measurement calibration significantly reduces this error. For greater accuracy, perform a 2-port measurement calibration.

**How to Measure AM-PM Conversion**

1. Preset the analyzer.

2. Select an S21 measurement in the power-sweep mode.

3. Enter the start and stop power levels for the analyzer's power sweep. The start power level should be in the linear region of the amplifier's response (typically 10-dB below the 1-dB compression point). The stop power should be in the compression region of the amplifier's response.

4. Select an external attenuator (if needed) so the amplifier's output power will be sufficiently attenuated to avoid causing receiver compression or damage to the analyzer's port 2.

5. Connect the amplifier as shown in the following graphic, and provide the dc bias.

![Amplifier Diagram](image)

6. Select the analyzer settings for your amplifier under test in order to perform a swept-power gain compression measurement at a chosen frequency. See *Gain Compression*.

7. Remove the amplifier and perform a measurement calibration. Be sure to include the attenuator and cables in the calibration setup if they will be used when measuring the amplifier.

8. Save the instrument state to memory.

9. Reconnect the amplifier.

10. Use a reference marker to target the amplifier's input power at the 1-dB gain compression point. Select a second marker and adjust its stimulus value until its response is 1-dB below the reference marker.

11. Change the S21 measurement from a log magnitude format to a **phase** format (no new calibration is required).

12. Find the **phase** change between the markers. The value is the AM-to-PM conversion coefficient at the 1-dB gain compression point.

13. Print the data or save it to a disk.
Amplifier Parameters Reference

- **Gain**
- **Gain Flatness**
- **Reverse Isolation**
- **Gain Drift Versus Time**
- **Deviation from Linear Phase**
- **Group Delay**
- **Return Loss (SWR, ρ)**
- **Complex Impedance**
- **Gain Compression**
- **AM-to-PM Conversion**

**See Also**

- **High-Gain Amplifiers**
- **High Power with PNA-X**

**Gain**

\[
\text{Gain (dB)} = -20\log_{10}\left(\frac{V_{\text{trans}}}{V_{\text{inc}}}\right)
\]

\[
\text{Gain (dB)} = P_{\text{out}}(\text{dBm}) - P_{\text{in}}(\text{dBm})
\]

The ratio of the amplifier’s output power (delivered to a Z₀ load) to the input power (delivered from a Z₀ source). Z₀ is the characteristic impedance, in this case, 50Ω.

For small signal levels, the output power of the amplifier is proportional to the input power. Small signal gain is the gain in this linear region.

As the input power level increases and the amplifier approaches saturation, the output power reaches a limit and the gain drops. Large signal gain is the gain in this nonlinear region. See **Gain Compression**.

**Gain Flatness**

The variation of the gain over the frequency range of the amplifier. See **Small Signal Gain and Flatness**.
**Reverse Isolation**

The measure of transmission from output to input. Similar to the gain measurement except the signal stimulus is applied to the output of the amplifier. See Reverse Isolation.

**Gain Drift versus Time (temperature, bias)**

The maximum variation of gain as a function of time, with all other parameters held constant. Gain drift is also observed with respect to other parameter changes such as temperature, humidity or bias voltage.

**Deviation from Linear Phase**

The amount of variation from a linear phase shift. Ideally, the phase shift through an amplifier is a linear function of frequency. See Deviation from Linear Phase.

**Group Delay**

\[ \tau_g (\Delta \omega) = - \frac{\Delta \theta}{\Delta \omega} \]

\[ = - \frac{1}{360} \cdot \frac{\Delta \theta}{\Delta f} \]

The measure of the transit time through the amplifier as a function of frequency. A perfectly linear phase shift would have a constant rate of change with respect to frequency, yielding a constant group delay. See Group Delay.

**Return Loss (SWR, \( \rho \))**

\[ \Gamma = \frac{V_{ref}}{V_{inc}} = \rho \angle \phi \]

Reflection coefficient = \( \rho \)

Return loss (dB) = \(-20 \log_{10} \rho\)

\[ SWR = \frac{1 + \rho}{1 - \rho} \]

The measure of the reflection mismatch at the input or output of the amplifier relative to the system \( Z_0 \) characteristic impedance.

**Complex Impedance**

\[ Z = \frac{1 + \Gamma}{1 - \Gamma} \cdot Z_0 \]

\[ = - R + jX \]

Complex impedance \((1+G)\). The amount of reflected energy from an amplifier is directly related to its impedance. Complex impedance consists of both a resistive and a reactive component. It is derived from the characteristic impedance of the system and the reflection coefficient. See Complex Impedance.

**Gain Compression**
The AM-to-PM conversion coefficient is expressed in units of degrees/dB at a given power level (usually $P_{1\text{dB}}$, which is the 1 dB gain compression point). See AM-PM Conversion.
Antenna Measurements

This topic describes how to setup the PNA to make S21 measurements on an array of antennas. Measurements can be made on up to 100 antenna arrays (Ports) and up to 15 discrete frequencies.

Measurement Sequence

1. The PNA is set to a start frequency.
2. As the antenna moves, the PNA responds to each external trigger signal by measuring an antenna port.
3. When all ports are measured, the PNA increments to the next frequency.
4. Again the PNA measures all ports, and so forth until all ports are measured at all frequencies in the forward direction.
5. As the antenna begins moving in the opposite direction, the same sequence occurs, except the PNA decrements in frequency until all ports are measured at all frequencies and the PNA is set back to the original start frequency.

Once setup, only external trigger signals are sent to the PNA. After each trigger, measurement data is stored in internal PNA memory.

How to set up the PNA

1. On the System menu click Preset.
2. On the Sweep menu point to Trigger then click Trigger.
3. In Trigger Source click External.
4. In Trigger Scope click Channel.
5. Click OK.

Forward Sweep

2. Click S21 then Channel Number 1.
3. On the Sweep menu point to Trigger then click Trigger.
4. In Channel Trigger State check Point Sweep.
5. Click OK.
6. On the Sweep menu click Sweep Type: then Segment Sweep.
7. Click OK.
8. On the View menu point to Tables then click Segment Table

9. Do this 15 times - Sweep menu point to Segment Table then Insert Segment

10. For each Segment in the Segment table:

   1. Click State: and select ON
   2. Double click both START and STOP Frequency: (each new segment ascends in frequency)
   3. Double click Points: type Number of Ports (elements)

Reverse sweep
Repeat the following steps for each frequency: (up to 15)

- Increment the channel number (X) Starting with Channel 2
- Decrement the frequency (F)

1. On the Trace menu click New Trace...
2. Click S21 then Channel Number X
3. When a window contains four traces, check Create in New Window.
4. Click OK
5. On the Sweep menu point to Trigger then click Trigger
6. In Channel Trigger State check Point Sweep
7. Click OK
8. On the Sweep menu click Sweep Type: then Segment Sweep
9. Click OK
10. On the View menu point to Tables then click Segment Table
11. In the Segment table

   1. Click State: and select ON
   2. Double click both START and STOP Frequency F
   3. Double click Points: type Number of Ports (elements)

Last Modified:

8-Apr-2008    Removed reference to antenna macro
Balanced Measurements

- **What are Balanced Devices?**
- **Differential and Common Modes Model**
- **Measuring Mixed Mode (Balanced) S-Parameters**
- **Measuring Imbalance Parameters**
- **Measuring CMRR**
- **Port Mapping**
- **How the PNA makes Balanced Measurements**

Other Measurement Setup Topics

Check out the Integrated True Mode Stimulus Application ([iTMSA](#)).

**What are Balanced Devices?**

Standard **Single-ended devices** generally have one input port and one output port. Signals on the input and output ports are referenced to ground.

Balanced devices have two pins on either the input, the output, or both. The signal of interest is the difference and average of the two input or output lines, not referenced to ground.

**Differential and Common Modes Model**

On balanced devices, the signal of interest is the **difference** and **average** of the two input or output lines. In balanced device terminology, these signals are known as the Differential and Common modes.

The following model shows how two signals (A and B) combine to create Differential and Common mode signals:
- Signal A is fixed at 1V peak
- Signal B is selectable
- Differential is calculated as \( A - B \)
- Common is calculated as the \( \text{AVERAGE} \) of A and B

**Note:** Click Signal B selections to see various Differential and Common signals.

<table>
<thead>
<tr>
<th>Signal B = SELECTABLE</th>
<th>Calculations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single-ended 0V</td>
<td>1 - 0 = 1</td>
</tr>
<tr>
<td>180° Out of Phase 1V</td>
<td>1 - (-1) = 2</td>
</tr>
<tr>
<td>180° Out of Phase 2V</td>
<td>1 - (-2) = 3</td>
</tr>
<tr>
<td>In Phase 1V</td>
<td>1 - 1 = 0</td>
</tr>
<tr>
<td>In Phase 2V</td>
<td>1 - 2 = -1</td>
</tr>
</tbody>
</table>

**Notes:**

- Even when Signal B is 0V, like a Single-ended signal, there is still a unique Differential and Common mode representation of the two individual signals.
- The above model does not show a DUT. The difference and average of two signals can be calculated for both the balanced INPUT and balanced OUTPUT of a device.

**Measuring Mixed Mode (Balanced) S-Parameters**

Mixed mode S-parameters combine traditional S-parameter notation with balanced measurement terminology. Some balanced devices are designed to amplify the differential component and reject the common component. This allows noise that is common to both inputs to be virtually eliminated from the output. For example, a balanced device may amplify the differential signal by a factor of 5, and attenuate the common signal by a factor of 5. Using
traditional S-parameter notation, an S21 is a ratio measurement of the device **Output** / device **Input**. Mixing this with balanced terminology, we could view the amplifier's Differential Output signal / Differential Input signal. To see this parameter on the PNA, we would select an Sdd21 measurement using the following balanced notation:

**Sabxy** -

Where

- **a** - device output mode
- **b** - device input mode

(choose from the following for both a and b:)

- **d** - differential
- **c** - common
- **s** - single ended

- **x** - device output "logical" port number
- **y** - device input "logical" port number

**See Also**

[Logical port mapping](#)

[Port mapping with External Test Sets](#)

[iTMSA](#)

**Measuring Imbalance Parameters**

Imbalance is a measure of how well two physical ports that make up a balanced port are matched. With a perfectly balanced port, the same amount of energy flows to both ports and the magnitude of the ratio of these ports is 1. The notation is similar to traditional S-parameters. In the following diagrams, the letters a, b, c, and d are used because any PNA port can be assigned to any logical port using the [port mapping process](#).

For example, in the following single-ended - balanced formula, **Sba** indicates the device output port is logical port b and the input port is logical port a.

![Imbalance parameter when measuring a single-ended - balanced device.](image)
Imbalance1 and Imbalance2 parameters when measuring a balanced - balanced device.

\[
\text{Imbal1} = -\frac{S_{ac} - S_{ad}}{S_{bc} - S_{bd}}
\]

\[
\text{Imbal2} = -\frac{S_{ca} - S_{cb}}{S_{da} - S_{db}}
\]

Imbalance1 and Imbalance2 parameters when measuring a single-ended - single-ended - balanced device.

\[
\text{Imbal1} = -\frac{S_{ac} - S_{ad}}{S_{bc} - S_{bd}}
\]

\[
\text{Imbal2} = -\frac{S_{ca} - S_{cb}}{S_{da} - S_{db}}
\]

**Measuring CMRR (Common Mode Rejection Ratio)**

CMRR is a ratio of the transmission characteristic in differential mode over the transmission characteristic in the common mode of the balanced port as the measurement parameter. A high value indicates more rejection of common mode, which is desirable in a device that transmits information in the differential portion of the signal. The table below shows the CMRR parameter you can select when measuring each balanced device.
Device Topology and Port Mapping

As we have seen on balanced inputs and outputs, the signal of interest is the difference or average of two BALANCED input or BALANCED output lines. It is also possible to have single-ended ports AND balanced ports on the same device. The two balanced input or output lines are referred to as a single "logical" port.

When configuring a balanced measurement on the PNA, select a device ‘topology’. Then map each PNA test port to the DUT ports. The PNA assigns "logical ports". See how to set device topology in the PNA.

The following device topologies can be measured by a 4-port PNA.

- **Balanced / Balanced**
  (2 logical ports - <4 actual ports>)
- **Single-ended / Balanced**
  (2 logical ports - <3 actual ports>)

- **Single-ended - Single-ended / Balanced**
  (3 logical ports - <4 actual ports>)

These topologies can be used in the reverse (⇐⇒) direction to measure:

- **Balanced / Single-ended** topology
- **Balanced / Single-ended - Single-ended** topology

For example, to measure a **Balanced / Single-ended** topology, measure the S12 (reverse direction) of a **Single-ended / Balanced** topology.

**How the PNA makes Balanced Measurements**

When using standard Balanced measurements, the PNA does not provide true balanced measurements by stimulating both balanced inputs together and measuring both outputs relative to one another. Instead, the PNA makes only Single-ended measurements. On a Balanced/Balanced device, it stimulates each input and measures each output individually. From the output data, the PNA calculates the Differential and Common outputs from the DUT using the same math formulas as the above model. However, all measurements and calculations on the PNA are performed in frequency domain using complex (magnitude and phase) data. The Balanced S-parameter display data is then calculated from the Differential and Common inputs and outputs.

In iTMSA mode, the PNA **DOES** stimulate both balanced inputs with true balanced sources. [Learn more](#).

---

**Last Modified:**

15-May-2008   Edited for iTMSA
Complex Impedance

When making an S11 or S22 measurement of your device under test, you can view complex-impedance data such as series resistance and reactance as well as phase and magnitude information. Complex impedance data can be viewed using either the Smith Chart format or the Polar format.

- What Is Complex Impedance?
- Accuracy Considerations
- How to Measure Complex Impedance

What Is Complex Impedance?
Complex-impedance data is information that can be determined from an S11 or S22 measurement of your device under test, such as:

- Resistance
- Reactance
- Phase
- Magnitude

The amount of power reflected from a device is directly related to the impedances of both the device and the measuring system. For example, the value of the complex reflection coefficient ($\Gamma$) is equal to 0 only when the device impedance and the system impedance are exactly the same (i.e. maximum power is transferred from the source to the load). Every value for $\Gamma^*$ corresponds uniquely to a complex device impedance (as a function of frequency), according to the equation:

$$Z_L = \frac{(1 + \Gamma^*)}{(1 - \Gamma^*)} \times Z_0$$

where $Z_L$ is your test device impedance and $Z_0$ is the measuring system's characteristic impedance.

Complex Impedance is best viewed using either Polar or Smith Chart format.

Accuracy Considerations

- The Smith chart is most easily understood when used with a full scale value of 1.0.
- For greater accuracy when using markers in the Smith chart or polar formats, activate the discrete marker mode.
- The uncertainty of reflection measurements is affected by:
  - Directivity
  - Reflection tracking
  - Source match
  - Load match (with 2-port devices)
With a 2-port calibration, the effects of these factors are reduced. A 1-port calibration provides the same accuracy if the output of the device is well terminated. Refer to the graphic below for the following discussion.

- If you connect the device between both analyzer ports, it is recommended that you use a 10 dB pad on the output of the device to improve measurement accuracy. This is not necessary if you use a 2-port calibration since it corrects for load match.

- If you connect a two-port device to only one analyzer port, it is recommended that you use a high-quality load (such as a calibration standard) on the output of the device.

**How to Measure Complex Impedance**

1. Connect the device as shown in the previous graphic.
2. Preset the analyzer.
3. Set up, calibrate, and perform an S11 or S22 measurement.
4. View impedance data:
   - a. Select the Smith Chart format.
   - b. Scale the displayed measurement for optimum viewing.
   - c. Position the marker to read the resistive and reactive components of the complex impedance at any point along the trace.
   - d. Print the data or save it to a disk.
5. View the magnitude and phase of the reflection coefficient:
   - a. Select the Smith chart format or the Polar format.
   - b. Select either Lin Marker or Log Marker formats.
   - c. Scale the displayed measurement for optimum viewing.
   - d. Position the marker to read the frequency, magnitude, and phase of the reflection coefficient (Γ) at any point along the trace.
   - e. Print the data or save it to a disk.
Comparing the PNA "Delay" Functions

The PNA has three Delay functions which are similar but are used in different ways.

1. **Group Delay format** is used to display the Group Delay of a network. Group Delay is defined as:

\[ -\frac{d(\phi)}{d(\omega)} \]  

where \( \phi \) is radian angle, and \( \omega \) is radian frequency.

Since it is defined by a derivative, the value must be determined from an analytic function. However, the PNA makes discrete measurements, so we approximate the group delay by taking the finite difference:

\[ \frac{1}{360} \frac{\delta(\phi)}{\delta(f)} \]  

where \( \phi \) is degree angle and \( f \) is frequency in Hz. The \( 1/360 \) does the proper conversion of degrees to radians and Hz frequency to radian frequency.

From this we can see that, if the phase response of a network varies with frequency, then the Group Delay must vary as well. In fact, many filters are specified by the variation of their Group Delay.

If we measure the phase response of a lossless cable, it should be a straight line. But, of course, nothing is perfect. The phase response will have a small amount of noise. This is due to trace noise of the PNA, and the loss with real cables or transmission lines, which causes a small amount of non-linear phase change with frequency. So, if we look at the Group Delay of a cable, we will see a small amount of variation. Also, if the frequency spacing is small enough when you make the measurement, the \( \delta(f) \) in the denominator becomes very small, so the delay can have wide swings with just a little noise.

To overcome this issue, we sometimes add smoothing to a phase trace, which widens the effective \( \delta(f) \), called the aperture, and provides a less noisy Group Delay response. The Group Delay of a device is only valid for a given frequency aperture. Learn more about Group Delay.

2. **Electrical Delay** function. On many filters, the passband response is specified for a maximum value of "Deviation from Linear Phase". When looking at the passband of a multi-pole filter, one sees the phase changing very rapidly. This makes it difficult to determine the linearity of the phase response. The Electrical Delay function subtracts out a "LINEAR PHASE" equivalent to the delay time value computed as above. When you use this function, you dial in the Linear Delay such that a CONSTANT PHASE SLOPE is removed from the phase trace, until the phase trace is mostly flat. The remaining variation is the deviation from linear phase.

To make this task a little less tedious, the PNA has a marker function called Marker =>> Delay. This function computes the Group Delay value at the marker position, using a 20% smoothing aperture, then changes the Electrical Delay value to this value. Obviously, if the phase trace is not perfectly linear, moving the marker and recomputing the delay will result in different values. The phase slope added by the electrical delay function applies only to the current measurement. That is, each measurement (S11, S22, S12, S21) can have its own value of electrical delay. Learn more about Deviation from Linear Phase.

3. **Port Extension** is a function that is similar to calibration. It applies to all the traces in a given channel. It compensates for the phase response change that occurs when the calibration reference plane is not the same as the measurement plane of the device.

Let's look at an example of a DUT that is mounted on a PCB fixture with SMA connectors. We can easily calibrate at the SMA connectors. But if we add the fixture to measure the board-mounted device, the apparent phase of the DUT is changed by the phase of the PCB fixture. We use port extensions to add a LINEAR PHASE (constant delay) to the calibration routines to shift the phase reference plane to that of the DUT. This is ONLY valid if the fixture consists of a transmission line with linear phase response, and this limitation is usually met in practice. The main reason that it is NOT met is that there is mismatch at the SMA-to-PCB interface. This mismatch was not removed with the error correction because it occurs AFTER the SMA connector. Ripple can be seen on the display as signals bounce back and forth between the mismatch and the DUT. If the DUT is well matched, the ripple effect is very small. However, when we use Automatic Port Extension (APE), and we leave the fixture open (the DUT removed), the reflection is large and we see larger ripples. That is why APE uses a curve fitting process to remove the effects of the ripple. For best effect, the wider the IF Bandwidth, the better we can "smooth-out" the ripples with...
curve fitting. Still, we are fitting a LINEAR PHASE SLOPE to the phase response, and thus we use only a single Port Extension Delay value to represent the phase slope.

The method used by older VNAs to get this same functionality was to add a mechanical line stretcher to the reference channel, which removed a fixed delay amount from the port. Port extensions give 1x the delay for transmission at each port, and 2x the delay for reflection, so it differs somewhat from Electrical Delay above, in that the math function depends upon the measurement being made. The signal passes twice through the fixture for reflection (out and back), but only once for each port on transmission. For S21, the phase slope added is the sum of the port 1 and port 2 Port Extension Delay values.

The "User Range" APE function is used in cases where a fixture has limited bandwidth, perhaps due to tuning elements or bias elements. In this case, the model of constant delay for the fixture over the whole bandwidth is not valid, so a narrower "User Range" of frequencies can be selected to compute the delay. Since the aperture is smaller, there is more uncertainty in the delay computation for port extension. Also, for those who had been using the Marker =>> Delay function to estimate the delay, we added the "Active Marker" selection to APE, which works exactly the same as Marker->Delay. Learn more about Automatic Port Extensions.
Deviation from Linear Phase

Deviation from linear phase is a measure of phase distortion. The electrical delay feature of the analyzer is used to remove the linear portion of the phase shift from the measurement. This results in a high-resolution display of the non-linear portion of the phase shift (deviation from linear phase).

- What Is Linear Phase Shift?
- What Is Deviation from Linear Phase?
- Why Measure Deviation from Linear Phase?
- Using Electrical Delay
- Accuracy Considerations

See also Comparing the PNA Delay Functions

What Is Linear Phase Shift?

Phase shift occurs because the wavelengths that occupy the electrical length of the device get shorter as the frequency of the incident signal increases. Linear phase-shift occurs when the phase response of a device is linearly proportional to frequency. Displayed on the analyzer, the phase-versus-frequency measurement trace of this ideal linear phase shift is a straight line. The slope is proportional to the electrical length of the device. Linear phase shift is necessary (along with a flat magnitude response) for distortionless transmission of signals.

What Is Deviation from Linear Phase?

In actual practice, many electrical or electronic devices will delay some frequencies more than others, creating non-linear phase-shift (distortion in signals consisting of multiple-frequency components). Measuring deviation from linear phase is a way to quantify this non-linear phase shift.

Since it is only the deviation from linear phase which causes phase distortion, it is desirable to remove the linear portion of the phase response from the measurement. This can be accomplished by using the electrical delay feature of the analyzer to mathematically cancel the electrical length of the device under test. What remains is the deviation from linear phase, or phase distortion.

Why Measure Deviation from Linear Phase?

The deviation from linear phase measurement accomplishes the following:

- Presents data in units of phase rather than units of seconds (group delay). For devices that pass modulated signals, units of phase may be most practical.
- Provides a less noisy measurement than a group delay measurement.
Using Electrical Delay

The electrical delay feature is the electronic version of the mechanical "line stretcher" of earlier analyzers. This feature does the following:

- Simulates a variable-length lossless transmission line, which is effectively added to or removed from the reference signal path.
- Compensates for the electrical length of the device under test.
- Flattens the measurement trace on the analyzer's display. This allows the trace to be viewed at high resolution in order to see the details of the phase nonlinearity.
- Provides a convenient method to view the deviation from linear phase of the device under test. See the following graphic.

Learn how to set Electrical Delay.

Accuracy Considerations

The frequency response of the test setup is the dominant error in a deviation from linear phase measurement. To reduce this error, perform a 2-port measurement calibration.

How to Measure Deviation from Linear Phase:

1. Preset the analyzer.

2. If your device under test is an amplifier, it may be necessary to adjust the analyzer's source power:
   - Set the analyzer's source power to be in the linear region of the amplifier's output response (typically 10-dB below the 1-dB compression point).
   - Select an external attenuator (if needed) so the amplifier's output power will be sufficiently attenuated to avoid causing receiver compression or damage to the analyzer’s port 2.

3. Connect the device under test as shown in the following graphic.
3. Select an S21 measurement.

4. Select the settings for your device under test, including the following:
   - **Format**: phase
   - **Scale**: autoscale

5. Remove the device and perform a calibration.

6. Reconnect the device.

7. Scale the displayed measurement for optimum viewing.

8. **Create a marker** in the middle of the trace.

9. Press the >**Delay** key to invoke the **Marker to Electrical Delay** function. This flattens the phase trace.

10. If desired, on the **Scale** menu, click **Electrical Delay** to fine-tune the flatness of the phase trace.

11. Use the markers to measure the maximum peak-to-peak deviation from linear phase.

12. Print the data or save it to a disk.
Small Signal Gain and Flatness

Small signal gain is the gain in the amplifier's linear region of operation. This is typically measured at a constant input power over a swept frequency. Gain flatness is the measure of the variation of gain over a specified frequency range.

- What Is Gain?
- What Is Flatness?
- Why Measure Gain and Flatness?
- Accuracy Considerations
- How to Measure Gain and Flatness

What Is Gain?

RF amplifier gain is defined as the difference in power between the amplifier output signal and the input signal. It is assumed that both input and output impedances of the amplifier are the same as the characteristic impedance of the system.

- Gain is called $S_{21}$ using S-parameter terminology
- Gain is expressed in dB-a logarithmic ratio of the output power relative to the input power.
- Gain can be calculated by subtracting the input from the output levels when both are expressed in dBm, which is power relative to 1 milliwatt.
- Amplifier gain is most commonly specified as a minimum value over a specified frequency range. Some amplifiers specify both minimum and maximum gain, to ensure that subsequent stages in a system are not under or over driven.

What Is Flatness?

Flatness specifies how much the amplifier's gain can vary over the specified frequency range. Variations in the flatness of the amplifier's gain can cause distortion of signals passing through the amplifier.

Why Measure Small-Signal Gain and Flatness?

Deviations in gain over the bandwidth of interest will induce distortion in the transmitted signal because frequency components are not amplified equally. Small-signal gain allows you to quantify the amplifier's gain at a particular frequency in a 50-ohm system. Flatness allows you to view the deviations in the amplifier's gain over a specified frequency range in a 50-ohm system.
Accuracy Considerations

- The amplifier may respond very differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.

- The output power of the amplifier should be sufficiently attenuated if necessary. Too much output power could:
  - damage the analyzer receiver
  - exceed the input compression level of the analyzer receiver, resulting in inaccurate measurements.

**Attenuation** of the amplifier's output power can be accomplished using:

- attenuators
- couplers

The frequency-response effects and mismatches of the attenuators and couplers must be accounted for during calibration since they are part of the test system. Proper error-correction techniques can reduce these effects.

- The **frequency response** is the dominant error in a small-signal gain and flatness measurement setup. Performing a thru-response measurement calibration significantly reduces this error. For greater accuracy, perform a 2-port measurement calibration.

- Reducing IF bandwidth or using averaging improves measurement dynamic range and accuracy, at the expense of measurement speed.

How to Measure Gain and Flatness

1. Preset the analyzer.

2. Select an S21 measurement parameter.

3. Set the analyzer's source power to be in the linear region of the amplifier's output response (typically 10-dB below the 1-dB compression point).

4. Select an external attenuator (if needed) so the amplifier's output power will be sufficiently attenuated to avoid causing receiver compression or damage to the analyzer's port-2.
5. Connect the amplifier as shown in the following graphic, and provide the dc bias.

6. Select the analyzer settings for your amplifier under test.

7. Remove the amplifier and perform a measurement calibration. Be sure to include the attenuator and cables in the calibration setup if they will be used when measuring the amplifier.

8. Save the instrument-state to memory.

9. Reconnect the amplifier.

10. Scale the displayed measurement for optimum viewing and use a marker to measure the small signal gain at a desired frequency.

11. Measure the gain flatness over a frequency range by using markers to view the peak-to-peak ripple.

12. Print or save the data to a disk.

13. This type of measurement can be automated.
Gain Compression

Gain compression measures the level of input power applied to an amplifier that will cause a distorted output. The Gain Compression Application (Opt 086) makes fast and accurate compression measurements.

- What Is Gain Compression?
- Why Measure Gain Compression?
- Accuracy Considerations
- How to Measure Gain Compression

See other Amplifier Parameter topics

What Is Gain Compression?

Gain compression occurs when the input power of an amplifier is increased to a level that reduces the gain of the amplifier and causes a nonlinear increase in output power.

The analyzer has the ability to do power sweeps as well as frequency sweeps. Power sweeps help characterize the nonlinear performance of an amplifier. Refer to the graphic below (a plot of an amplifier’s output power versus input power at a single frequency) for the following discussion.

- The amplifier has a linear region of operation where gain is constant and independent of power level. The gain in this region is commonly referred to as "small-signal gain."
- As the input power increases, the amplifier gain appears to decrease, and the amplifier goes into compression.
- The most common measurement of amplifier compression is the 1-dB compression point. This is defined as the input power (or sometimes the output power) which results in a 1-dB decrease in amplifier gain (relative to the amplifier’s small-signal gain).

Why Measure Gain Compression?

When driven with a sinusoid, the output of an amplifier is no longer sinusoidal in the compression region. Some of the amplifier output appears in harmonics, rather than occurring only at the fundamental frequency of the input.
As input power is increased even more, the amplifier becomes saturated, and output power remains constant. At this point, further increases in amplifier input power result in no change in output power.

In some cases (such as with TWT amplifiers), output power actually decreases with further increases in input power after saturation, which means the amplifier has negative gain.

Since gain is desired in amplifier operation, it is important to know the limit of input signal that will result in gain compression.

Accuracy Considerations

The network analyzer must provide sufficient power to drive the amplifier into saturation. If you need a higher input-power level than the source of the analyzer can provide, use a preamplifier to boost the power level prior to the amplifier under test. (See High-Power Component Measurements.) If using a preamplifier, you can increase measurement accuracy in the following ways:

- Use a coupler on the output of the preamplifier so that a portion of the boosted input signal can be used for the analyzer's reference channel. This configuration removes the preamplifier's frequency response and drift errors from the measurement (by ratioing).
- Perform a thru-response calibration including the preamplifier, couplers, and attenuators in the test setup.

The output power of the amplifier should be sufficiently attenuated if necessary. Too much output power could:

- Damage the analyzer receiver
- Exceed the input compression level of the analyzer receiver

Attenuation of the amplifier's output power can be accomplished using:

- Attenuators
- Couplers

The frequency-response effects of the attenuators and couplers must be considered during calibration since they are part of the test system. Proper error-correction techniques can reduce these effects.

- The frequency response is the dominant error in a gain compression measurement setup. Performing a thru-response measurement calibration significantly reduces this error.
- The amplifier may respond very differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.
- Reducing IF bandwidth or using measurement averages improves accuracy, at the expense of measurement speed.

How to Measure Gain Compression

This procedure shows you how to make the following three measurements used to determine amplifier gain compression:
1. A Swept-Frequency Gain Compression measurement locates the lowest frequency at which the 1-dB gain compression first occurs.

2. A Swept-Power Gain Compression measurement shows the input power at which a 1-dB drop in gain occurs as a power ramp is applied to the amplifier at a particular frequency point (found in measurement 1).

3. An Absolute Power measurement shows the absolute power out (in dBm) at compression.

**Swept-Frequency Gain Compression Measurement**

A measurement of swept frequency gain compression locates the frequency point where 1-dB compression first occurs.

1. Preset the analyzer.

2. Select an S21 measurement parameter.

3. Set the analyzer’s source power to be in the linear region of the amplifier’s output response (typically 10-dB below the 1-dB compression point).

4. Select an external attenuator (if needed) so the amplifier’s output power will be sufficiently attenuated to avoid causing receiver compression or damage to the analyzer’s port.

5. Connect the amplifier as shown in the following graphic, and provide the dc bias.

6. Select the analyzer settings for your amplifier under test. To reduce the effects of noise, you may want to specify a narrower IF bandwidth.

7. Remove the amplifier and perform a thru-response calibration. Be sure to include the attenuator and cables in the calibration setup if they will be used when measuring the amplifier.

8. Save the instrument-state to memory.

9. Reconnect the amplifier.

10. Position a marker at approximately mid-span.

11. Adjust the analyzer’s scale to 1 dB per division.
12. Store the trace in memory and display Data/Mem.

13. Gradually increase the source power until a 1-dB decrease in gain is observed at the first frequency over some portion of the trace.

14. Use markers to locate the frequency where the 1-dB decrease in gain first occurs. Note this frequency for use in the following measurement.

15. Print the data or save it to a disk.

**Swept-Power Gain Compression Measurement**

A swept-power gain compression measurement shows the input power resulting in a 1-dB drop in gain as a power ramp at a particular frequency (found in step 13 of the previous measurement) is applied to the amplifier.

1. If not already done, perform the previous measurement of swept-frequency gain compression.

2. Setup an S21 measurement in the power-sweep mode. Include the following settings:
   - Set the CW frequency to the frequency noted in step 14 of the previous measurement of swept-frequency gain compression.
   - Enter the start and stop power levels for the sweep. The start power should be in the linear region of the amplifier’s response (typically 10 dB below the 1-dB compression point). The stop power should be in the compression region of the amplifier’s response.

3. Adjust the scale to 1-dB per division.

4. Use markers (including reference marker) to find the input power where the 1-dB decrease in gain occurs.

5. Print the data or save it to a disk.

**Absolute Output Power Measurement**

An absolute-power measurement shows the absolute power-out (in dBm) of the amplifier at compression.

1. Select an unratioed (absolute) power measurement. Choose the B input if using the test setup in the previous graphic.

2. Retain the CW frequency used in the previous measurement of swept-power gain compression.

3. Set a marker to the input power level where the 1-dB decrease in gain occurs (found in step 4 of the previous measurement).

4. Scale the displayed measurement for optimum viewing.

5. Read the marker value to find the absolute output power of the amplifier (in dBm) where the 1-dB decrease in gain occurs.

6. Print the data or save it to a disk.
Note: The measurement calibration does not apply to absolute power. Therefore, if there is any attenuation external to the analyzer, you will have to correct for it manually.
Group Delay

Group delay is a measure of phase distortion. Group delay is the actual transit time of a signal through a device under test as a function of frequency. When specifying group delay, it is important to specify the aperture used for the measurement.

- What is Group Delay?
- Group Delay versus Deviation from Linear Phase
- What Is Aperture?
- Accuracy Considerations
- How to Measure Group Delay

See also Comparing the PNA Delay Functions.

See other Amplifier Parameter topics

What Is Group Delay?

Group delay is:

- A measure of device phase distortion.
- The transit time of a signal through a device versus frequency.
- The derivative of the device’s phase characteristic with respect to frequency.

Refer to the graphic below for the following discussion:

The phase characteristic of a device typically consists of both linear and higher order (deviations from linear) phase-shift components.
### Linear phase-shift component:  
Represents average signal transit time. 
Attributed to electrical length of test device.

### Higher-order phase-shift component:  
Represents variations in transit time for different frequencies.  
Source of signal distortion.

Refer to the graphic below for the following discussion:

In a group delay measurement:

- The linear phase shift component is converted to a constant value (representing the average delay).
- The higher order phase shift component is transformed into deviations from constant group delay (or group delay ripple).
- The deviations in group delay cause signal distortion, just as deviations from linear phase cause distortion.
- The measurement trace depicts the amount of time it takes for each frequency to travel through the device under test.

Refer to the following equation for this discussion on how the PNA computes group delay:

\[
\text{Group Delay} = t_g = \frac{-d\phi}{d\omega} = \frac{-1}{360^\circ} \cdot \frac{d\Theta}{df} \quad (\omega \text{ in Radians/Sec}, \Theta \text{ in Degrees}, f \text{ in Hz, } \omega = 2\pi f)
\]

- Phase data is used to find the phase change (-d\(\phi\)).
- A specified frequency aperture is used to find the frequency change (d\(\omega\)).
- Using the two values above, an approximation is calculated for the rate of change of phase with frequency.
- This approximation represents group delay in seconds (assuming linear phase change over the specified frequency aperture).

**Group Delay versus Deviation from Linear Phase**
Group delay is often a more accurate indication of phase distortion than Deviation from Linear Phase.

**Deviation from linear phase** results are shown in the upper region of the following graphic: Device 1 and device 2 have the same value, despite different appearances.

**Group Delay** results are shown in the lower region: Device 1 and device 2 have different values of group delay. This is because in determining group delay, the analyzer calculates slope of phase ripple, which is dependent on number of ripples which occur per unit of frequency.

**What Is Aperture?**

During a group delay measurement, the PNA measures the phase at two closely spaced frequencies and then computes the phase slope. The frequency interval (frequency delta) between the two phase measurement points is called the aperture. Changing the aperture can result in different values of group delay. The computed slope ( - delta phase / delta frequency) varies as the aperture is increased. This is why when you are comparing group delay data, you must know the aperture that was used to make the measurements.

Refer to the graphic below for the following discussion:
Narrow aperture:  |  Wide aperture:
---|---
Provides more detail in phase linearity.  |  Provides less detail in phase linearity because some phase response averaged-out or not measured.
Makes measurement susceptible to noise (smaller signal-to-noise ratio) and PNA phase detector resolution.  |  Makes measurement less susceptible to noise (larger signal-to-noise ratio).

Group delay measurements can be made using the following sweep types:

- Linear frequency
- List frequency sweep segment - The group delay aperture varies depending on the frequency spacing and point density. Therefore the aperture is not constant in segment sweep. In segment sweep, extra frequency points can be defined to ensure the desired aperture.

### How to set Group Delay Aperture

#### Using front-panel HARDKEY [softkey] buttons

1. Press **Avg**
2. then **[More]**
3. then **[Group Delay Aperture]**

#### PNA Menu using a mouse

1. Click **Response**
2. then **Avg**
3. then **Group Delay Aperture**

![Programming Commands]
Although the Group Delay Aperture is defined as the difference in frequency between two data points (see What Is Aperture?), the group delay calculation can be averaged over many adjacent data points, similar to the PNA smoothing feature. The number of adjacent data points can be set using any of the following methods:

**Note:** You can change the default Group Delay Aperture to two points using a PNA Preference. Learn how.

**Points**  Number of adjacent data points to average. Default setting is 11 points. Choose a value between 2 and the current number of points in the channel.

**Percent of Span**  The data points within this percentage of the current frequency span are averaged. Choose a value between (2 points / current number of points) and 100 percent. The span must contain at least two data points.

**Frequency**  The data points within this frequency range are averaged. The frequency range must contain at least two data points.

When the frequency span or number of points is reduced so that the current Group Delay Aperture is NOT attainable, the Aperture is adjusted to the new frequency span or number of points.

**OK**  Applies setting changes and closes the dialog box.

**Cancel**  Closes the dialog. Setting changes are NOT applied.

---

**Accuracy Considerations**

It is important to keep the phase difference between two adjacent measurement points less than 180° (see the following graphic). Otherwise, incorrect phase and delay information may result. Undersampling may occur when measuring devices with long electrical length. You can verify that the phase difference measured between two adjacent points is less than 180° by adjusting the following settings until the measurement trace no longer changes:

- Increase the number of points
- Narrow the frequency span

Electrical delay may also be used to compensate for this effect.

The frequency response is the dominant error in a group delay test setup. Performing a thru-response
measurement calibration significantly reduces this error. For greater accuracy, perform a 2-port measurement calibration.

Particularly for an amplifier, the response may vary differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.

**How to Measure Group Delay**

1. Preset the analyzer.

2. If your DUT is an amplifier, it may be necessary to adjust the PNA source power:
   - Set the source power to be in the linear region of the amplifier’s output response, typically 10 dB below the 1 dB compression point.
   - If needed, use an external attenuator so the amplifier output power will be sufficiently attenuated to avoid causing receiver compression or damage to the PNA port 2.

3. Connect the DUT as shown in the following graphic.

   ![Diagram](image)

   - Direct Connection

4. Select an S21 measurement.

5. Select the settings for your DUT:
   - frequency range
   - number of measurement points.
   - format: delay
   - scale: autoscale

6. Remove the DUT and perform a measurement calibration.

7. Reconnect the DUT.

8. Scale the displayed measurement for optimum viewing.

9. Use the Group Delay Aperture setting to increase the aperture, reducing noise on the trace while maintaining
meaningful detail.

10. Use the markers to measure group delay (expressed in seconds) at a particular frequency of interest.

11. Print the data or save it to a disk.

Last Modified:

23-Feb-2010  Added new dialog and text

4-Jan-2010  Fixed default aperture formula
High-Gain Amplifier Measurements

When measuring High-Gain Amplifiers, errors in measuring any of the S-parameters during calibration can result in error in the S21 measurement. This is because all the S-parameters are used in the error correction math.

A particular problem occurs with high gain amplifiers because the source power is set very low. Thus, when making reverse measurements (S22, S12) the signal-to-noise is poor and the raw measurements can be dominated by noise. This noise in the raw measurements will result in a noisy trace appearing for corrected S21 or S11.

If you are using a large attenuator on port 2 (which improves output match), perform an Enhanced Response Calibration as follows. This corrects for the same errors as the full 2-port correction EXCEPT the interaction between the raw load match and the DUT output match.

1. There is NO need to Uncouple the port powers.
2. Set port powers to an acceptable level. Do NOT overpower the PNA test port.
3. Perform Enhanced Response Cal. Learn how. (Does not measure or correct for the S12 or S22 / PNA port match).

If you want to do a full correction (for example, when your amplifier output match is poor so the Enhanced Response Cal above is not adequate), then...

1. Uncouple the port powers. Learn how.
2. Set input (port 1) power to approximately the output power of the amplifier up to 0 dBm
3. Set reverse (port 2) power to the same power (for measuring isolation and S22)
4. Perform a Full 2-port Cal.
5. Re-set the input power (port 1) to a lower power level appropriate for driving the amplifier.

Additional Error due to Mismatch of DUT Output Match and Raw Load Match
This topic is now covered in detail in Application Note 1408-10, High-power measurements using the PNA (5989-1349EN) at Agilent.com.

See Also

High-Power Amplifier Measurements using a PNA-X
High-Power Amplifier Measurements with the PNA-X

The following is a block diagram of the PNA-X Opt 423. The configuration displayed here is used to make high power amplifier measurements using a preamplifier at the rear panel. The preamplifier can then be switched (SW1) as needed using the RF Configurator.

Legend

<table>
<thead>
<tr>
<th>Color</th>
<th>Component</th>
<th>Damage Level</th>
</tr>
</thead>
<tbody>
<tr>
<td>Green</td>
<td>Bridges</td>
<td>+33 dBm</td>
</tr>
<tr>
<td>Blue</td>
<td>Couplers</td>
<td>+43 dBm</td>
</tr>
<tr>
<td>Orange</td>
<td>Bias-tees</td>
<td>+30 dBm</td>
</tr>
<tr>
<td>Purple</td>
<td>User-supplied pre-amp and high-power attenuator</td>
<td>N/A</td>
</tr>
</tbody>
</table>

Notes

At J11 (rear-panel), max power is 4 dB to 11 dB higher than Source 1 Out at front panel jumper due to loss of the coupler thru arms, bias-tees, and cables.

At J10 max power +33 dBm, which is the damage level of the bridge. With +30 dBm into J10, there will be about +15 dBm at R1, assuming 15 dB coupling factor for the R1 bridge. +15 dBm is the damage level of that receiver. Therefore, it may be necessary to add attenuation in place of the R1 loop, not only to protect the receiver, but to bring it out of compression. The 0.1 dB compression level spec for the R1 receiver is between -3 and -18 dBm, depending on the frequency and option configuration.
At Test Port 2 (DUT output): With the bias-tees (orange), only +30 dBm is allowed into the test port. With Opt H85 (bias-tees removed), +43 dBm is allowed. Add appropriate attenuation to not damage other components.

About the PNA-X Option H85
A modified version of the standard PNA-X, the option H85 is designed to permit insertion of high power amplifiers and other signal conditioning equipment to allow high power network measurements at RF levels up to 20 Watts (+43 dBm) from 10 MHz to 26.5 GHz. The Option H85 modification supplies extended power range attenuators without bias tees. This is similar to the PNA-X -219 (add extended power range and bias-tees to 2-Port analyzer) or PNA-X -419 (add extended power range and bias-tees to 4-Port analyzer) but deletes the bias tees from the test set.

See Also

- High-Gain Amplifier Measurements
- N5242-H85 Manual
- RF Path Configurator
- IF Path Configurator

Last Modified:

- 6-Apr-2009    Replaced N5242A with PNA-X
- 14-Aug-2008    Added link to H85 manual
- 10-May-2007    MX New topic
Phase Measurements

Knowledge of both magnitude and phase characteristics is needed for successful higher-level component integration.

- **What are Phase Measurements?**
- **Why Measure Phase?**
- **Using the Analyzer's Phase Format**
- **Types of Phase Measurements**

**See other Tutorials**

**What are Phase Measurements?**

Phase measurements are made using S-parameters, just like amplitude measurements. A phase measurement is a relative (ratio) measurement and not an absolute measurement. Phase measurements compare the phase of the signal going into a device (the incident signal) to the phase of the device's response signal. The response signal can be either reflected or transmitted. Assuming an accurate calibration has been performed, the difference in phase between the two signals (known as phase shift) is a result of the electrical characteristics of the device under test.

The following graphic shows the phase shift (in time or degrees) between an incident signal and a transmitted signal (as might be seen on an oscilloscope display).

**Why Measure Phase?**

Measuring phase is a critical element of network analysis. The following graphic lists five reasons for measuring both magnitude and phase.
When used in communications systems to pass signals, components or circuits must not cause excessive signal distortion. This distortion can be:

- Linear, where flat magnitude and linear phase shift versus frequency is not maintained over the bandwidth of interest.
- Nonlinear, such as AM-to-PM conversion.

It is important to measure how reflective a component or circuit is, to ensure that it transmits or absorbs energy efficiently. Measuring the complex impedance of an antenna is a good example.

**Using the Analyzer’s Phase Format**

The analyzer’s phase format displays a phase-versus-frequency or phase-versus-power measurement. The analyzer does not display more than ±180 degrees phase difference between the reference and test signals. As the phase value varies between +180 degrees and -180 degrees, the analyzer display creates the sawtooth pattern as shown in the following graphic.

The sawtooth pattern does not always reach +180 degrees and -180 degrees. This is because the measurement is made at discrete frequencies, and the data point at +180 degrees and -180 degrees may not be measured for the selected sweep.

**Types of Phase Measurements**

Complex impedance data is information such as resistance, reactance, phase, and magnitude that can be
determined from an S11 or S22 measurement. Complex impedance data can be viewed using either the Smith Chart format or the Polar format.

**AM-to-PM conversion** is a measure of the amount of undesired phase deviation (PM) that is caused by amplitude variations (AM) of the system. AM-to-PM conversion is usually defined as the change in output phase for a 1-dB increment in the input power to an amplifier (i.e. at the 1 dB gain compression point). This is expressed in degrees-per-dB (°/dB).

**Deviation from linear phase** is a measure of phase distortion caused by a device. Ideally, the phase shift through a device is a linear function of frequency. The amount of variation from this theoretical phase shift is known as its deviation from linear phase (also called phase linearity).

**Group delay** is another way to look at phase distortion caused by a device. Group delay is a measure of transit time through a device at a particular frequency. The analyzer computes group delay from the derivative of the measured phase response.

### Deviation from Linear Phase Versus Group Delay

Although deviation from linear phase and group delay are similar measurements, they each have their purpose. The following are the advantages of deviation from linear phase measurements:

- Less noisy than group delay.
- Able to characterize devices that pass phase modulated signals, and show units of phase rather than units of seconds.

The following are the advantages of group delay measurements:

- More easily interpreted indication of phase distortion than deviation from linear phase.
- Able to most accurately characterize a device under test. This is because in determining group delay, the analyzer calculates the slope of the phase ripple, which is dependent on the number of ripples which occur per unit of frequency. Comparing two phase responses with equal peak-to-peak phase ripple, the response with the larger phase slope results in:
  - More group delay variation.
  - More signal distortion.

See also [Comparing the PNA Delay Functions](#).
Reverse Isolation

Reverse isolation is a measure of amplifier reverse transmission response- from output to input.

- What is Reverse Isolation
- Why Measure Reverse Isolation?
- Accuracy Considerations
- How to Measure Reverse Isolation

What is Reverse Isolation?
Reverse isolation is a measure of how well a signal applied to the device output is "isolated" from its input. The measurement of reverse isolation is similar to that of forward gain, except:

- The stimulus signal is applied to the amplifier's output port.
- The response is measured at the amplifier's input port.

The equivalent S-parameter is S12.

Why Measure Reverse Isolation?
An ideal amplifier would have infinite reverse isolation-no signal would be transmitted from the output back to the input. However, reflected signals can pass through the amplifier in the reverse direction. This unwanted reverse transmission can cause the reflected signals to interfere with the desired fundamental signal flowing in the forward direction. Therefore, reverse isolation is important to quantify.

Accuracy Considerations
Since amplifiers often exhibit high loss in the reverse direction, generally there is no need for any attenuation that may have been used to protect the port 2 receiver during forward transmission measurements. Removing the attenuation will:

- Increase the dynamic range, resulting in improved measurement accuracy.
- Require a new calibration for maximum accuracy.

The RF source power can be increased to provide more dynamic range and accuracy.

Note: With the attenuation removed and the RF source power increased, a forward sweep could damage the analyzer's port 2 receiver. Do not perform a forward sweep or use 2-port calibration unless the forward power is set low enough to avoid causing port 2 receiver compression or damage.
If the isolation of the amplifier under test is very large, the transmitted signal level may be near the noise floor or crosstalk level of the receiver. To lower the noise floor:

- Use or increase measurement averages.
- Reduce the IF bandwidth of the analyzer.

**Note:** Reducing IF bandwidth or using averaging improves measurement dynamic range and accuracy, at the expense of reduced measurement speed.

- When crosstalk levels affect the measurement accuracy, reduce the crosstalk error term by performing a response and isolation calibration. When performing the isolation part of the calibration it is important to use the same average factor and IF bandwidth during the calibration and measurement.
- The frequency response of the test setup is the dominant error in a reverse isolation measurement. Performing a thru-response measurement calibration significantly reduces this error. This calibration can be done as part of the response and isolation calibration.
- The amplifier may respond very differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.

**How to Measure Reverse Isolation**

1. Connect the amplifier as shown in the following graphic.

   ![Diagram of amplifier connection](image)

2. Preset the analyzer.
3. Select an S12 measurement.
4. Select the settings for your amplifier under test.
5. Remove the amplifier and perform a thru-response calibration or a response and isolation calibration.
6. Scale the displayed measurement for optimum viewing and use a marker to measure the reverse isolation at a desired frequency.
7. Print or save the data to a disk.
Reflection Measurements

Reflection measurements are an important part of network analysis.

- **What are Reflection Measurements?**
- **Why Make Reflection Measurements?**
- **Expressing Reflected Waves**
  - **Return Loss**
  - **VSWR**
  - **Reflection Coefficient**
  - **Impedance**
- **Summary of Expressions**

---

### See other Tutorials

---

**What are Reflection Measurements?**

To understand reflection measurements, it is helpful to think of traveling waves along a transmission line in terms of a lightwave analogy. We can imagine incident light striking some optical component like a clear lens. Some of the light is reflected off the surface of the lens, but most of the light continues on through the lens. If the lens had mirrored surfaces, then most of the light would be reflected and little or none would be transmitted.

1. Incident  
2. Reflected  
3. Transmitted

1. Incident  2. Reflected  3. Transmitted

With RF energy, reflections occur when the impedance of two mated devices are not the same. A reflection measurement is the ratio of the reflected signal to the incident signal. Network analyzers measure the incident wave with the R (for reference) channel and the reflected wave with the A channel. Therefore, reflection is often shown as the ratio of A over R (A/R). We can completely quantify the reflection characteristics of our device under test (DUT) with the amplitude and phase information available at both the A and R channel. In S-parameter terminology, S11 is a reflection measurement of port 1 of the device (the input port); S22 is a reflection measurement of the port 2 (the output port).
Why Make Reflection Measurements?

One reason we make reflection measurements to assure efficient transfer of RF power. We do this because:

1. RF energy is not cheap. When energy is reflected, that means less energy is transmitted to where it is intended to go.
2. If the reflected energy is large, it can damage components, like amplifiers.

For example, in the following graphic, the radio station on the left is not operating at peak efficiency. The amplifier impedance is not the same as the transmission line, and the transmission line impedance is not the same as the antenna. Both of these conditions cause high reflected power. This condition results in less transmitted power, and the high reflected power could damage the amplifier.

The radio station on the right installed properly "matched" transmission line and antenna. Very little of the transmitted signal is reflected, resulting in increased broadcast power, more listeners, more advertising revenue, and more profit. The amplifier, transmission, and antenna all need to be measured to ensure that reflected power is minimized.

Expressing Reflected Waves

After making a reflection measurement, the reflection data can be expressed in a number of ways, depending on what you are trying to learn. The various expressions are all calculated by the analyzer from the same reflection measurement data. Each method of expressing reflection data can be graphically displayed in one or more formats. For more information, see display formats.

Return Loss

The easiest way to convey reflection data is return loss. Return loss is expressed in dB, and is a scalar (amplitude only) quantity. Return loss can be thought of as the absolute value or dB that the reflected signal is below the incident signal. Return loss varies between infinity for a perfect impedance match and 0 dB for an open or short circuit, or a lossless reactance. For example, using the log magnitude format on the analyzer, the measured reflection value on the screen may be -18dB. The minus sign is ignored when expressing return loss, so the component is said to have 18dB of return loss.

VSWR

Two waves traveling in opposite directions on the same transmission line cause a "standing wave". This condition can be measured in terms of the voltage standing wave ratio (VSWR or SWR for short). VSWR is defined as the maximum reflected voltage over the minimum reflected voltage at a given frequency. VSWR is a scalar (amplitude only) quantity. VSWR varies between one for a perfect match, and infinity for an open or short circuit or lossless reactance.

Reflection Coefficient

Another way of expressing reflection measurements is reflection coefficient gamma (\(\Gamma\)). Gamma includes both
magnitude and phase.
The magnitude portion of gamma is called rho ($\rho$). Reflection coefficient is the ratio of the reflected signal voltage to the incident signal voltage. The range of possible values for $\rho$ is between zero and one. A transmission line terminated in its characteristic impedance will have all energy transferred to the load; zero energy will be reflected and $\rho = 0$. When a transmission line terminated in a short or open circuit, all energy is reflected and $\rho = 1$. The value of rho is unitless.

Now for the phase information. At high frequencies, where the wavelength of the signal is smaller than the length of conductors, reflections are best thought of as waves moving in the opposite direction of the incident waves. The incident and reflected waves combine to produce a single "standing" wave with voltage that varies with position along the transmission line.

When a transmission line is terminated in its characteristic impedance ($Z_0$) there is no reflected signal. All of the incident signal is transferred to the load, as shown in the following graphic. There is energy flowing in one direction along the transmission line.

![Standing Wave Graphic](image)

When a transmission line is terminated in a short circuit termination, all of the energy is reflected back to the source. The reflected wave is equal in magnitude to the incident wave ($\rho = 1$). The voltage across any short circuit is zero volts. Therefore, the voltage of the reflected wave will be 180 degrees out of phase with the incident wave, canceling the voltage at the load.

![Standing Wave Graphic](image)

When a transmission line is terminated in an open circuit termination, all of the energy is reflected back to the source. The reflected wave is equal in magnitude to the incident wave ($\rho = 1$). However, no current can flow in an open circuit. Therefore, the voltage of the reflected wave will be in phase with the voltage of the incident wave.
When a transmission line is terminated in a 25 ohm resistor, some but not all of the incident energy will be absorbed, and some will be reflected back towards the source. The reflected wave will have an amplitude 1/3 that of the incident wave and the voltage of the two waves will be out of phase by 180 degrees at the load. The phase relationship will change as a function of distance along the transmission line from the load. The valleys of the standing wave pattern will no longer go to zero, and the peaks will be less than that of the open / short circuit.

For more information, see [Phase Measurements](#).

**Impedance**

Impedance is another way of expressing reflection data. For more information on Impedance, see [Smith Charts](#).

**Summary of the Expressions of Reflection Measurements:**

<table>
<thead>
<tr>
<th>No reflection (Z_i = Z_0)</th>
<th>Full reflection (Z_i = open, short)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Return loss</strong> = -20 log(\rho), \rho =</td>
<td></td>
</tr>
<tr>
<td>\infty dB</td>
<td>RL</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>VSWR</strong> = \frac{E_{max}}{E_{min}} = \frac{1 + \rho}{1 - \rho}</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Reflection Coefficient</strong> \Gamma = \frac{V_{reflected}}{V_{incident}} = \frac{\rho}{\rho} = \frac{Z_i - Z_0}{Z_i + Z_0}</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>
Select the PNA Model

Please select the PNA model of the specifications you would like to see.

**Note:** Beginning with PNA Rev. A.09.33, PNA specs are no longer embedded in PNA Help.

An internet connection is required to view ALL specifications documents.

<table>
<thead>
<tr>
<th>Doc Number</th>
<th>PNA Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>E8361-90007</td>
<td>E8361A/C (67 GHz)</td>
</tr>
<tr>
<td>E8364-90031</td>
<td>E8362A/B/C (20 GHz)</td>
</tr>
<tr>
<td></td>
<td>E8363A/B/C (40 GHz)</td>
</tr>
<tr>
<td></td>
<td>E8364A/B/C (50 GHz)</td>
</tr>
<tr>
<td>N5230-90016</td>
<td>N5230A/C 2-Port (6, 13, 20, 40, 50 GHz)</td>
</tr>
<tr>
<td>N5230-90020</td>
<td>N5230A/C 4-Port (13.5 and 20 GHz)</td>
</tr>
</tbody>
</table>

**N522x Models**

<table>
<thead>
<tr>
<th>Doc Number</th>
<th>PNA Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5221-90001</td>
<td>N5221A (13.5 GHz)</td>
</tr>
<tr>
<td></td>
<td>N522A (26.5 GHz)</td>
</tr>
<tr>
<td>N5224-90001</td>
<td>N524A (43.5 GHz)</td>
</tr>
<tr>
<td></td>
<td>N525A (50 GHz)</td>
</tr>
<tr>
<td>N5227-90002</td>
<td>N527A (67 GHz)</td>
</tr>
</tbody>
</table>

**PNA-X Models**

<table>
<thead>
<tr>
<th>Doc Number</th>
<th>PNA Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5242-90007</td>
<td>N5241A (13.5 GHz)</td>
</tr>
<tr>
<td></td>
<td>N5242A (26.5 GHz)</td>
</tr>
<tr>
<td>N5245-90008</td>
<td>N5244A (43.5 GHz)</td>
</tr>
<tr>
<td></td>
<td>N5245A (50 GHz)</td>
</tr>
<tr>
<td>N5247-90002</td>
<td>N5247A (67 GHz)</td>
</tr>
</tbody>
</table>

**Others**

<table>
<thead>
<tr>
<th>Doc Number</th>
<th>PNA Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5264-90003</td>
<td>N5264A (Measurement Receiver)</td>
</tr>
<tr>
<td>N5245-90016</td>
<td>N5244A or N5245A with Option H29 (Noise Figure to 26.5 GHz)</td>
</tr>
</tbody>
</table>

See the [equations that are used to generate uncertainty curves](#).

**Block Diagrams** for the following models are available in PNAHelp:
- PNA-X Models with Opt 224 and 423
- N522x Models with Opt 219 and 419

The following are specifications for discontinued models:

- E8356A, E8357A, E8358A
- E8801A, E8802A, E8803A
- N3381A, N3382A, N3383A
- E8362A, E8363A, E8364A
Glossary

12-Term Error Correction See Error Correction, 12-Term.

1-Port Device A device with a single connector or path to the device's circuitry. Examples include an oscillator and a load.

2-Port Calibration, Full See Error Correction, 12-Term.

2-Port Device A device with two connectors or other paths to the device's circuitry. Examples include filters, SAW devices, attenuators, matching pads, and amplifiers.

3-Term Error Correction See Error Correction, 3-Term.

A

Active Channel The highlighted channel affected by front panel functions.

Active Function Readout The area of a display screen where the active function and its state are displayed. The active function is the one that was completed by the last key selection or remote programming command.

Active Marker The marker on a trace that can be repositioned either by front panel controls or by programming commands.

Active Trace A trace that is being swept (updated) with incoming signal information.

ADC Analog to Digital Converter

Address The identification (represented by a name, label, or number) for a register, location in storage, or any other data source or destination. Examples are the location of a station in a communications network, or a device on the GP-IB.

ADM Add-Drop Multiplexer

Admittance (Y) The inverse of an impedance (i.e. the ratio of current to voltage). Complex admittances take the form \( Y = G + jB(t) \).

ALC Automatic Level Control. See Automatic Gain Control.

AM Amplitude Modulation

AM Group Delay A technique for the measurement of group delay through a device which utilizes an amplitude modulated (AM) source. Note: The actual delay of the modulation envelope is measured directly with an external scalar detector. Devices that distort the amplitude of a signal cannot be measured. These include amplifiers with automatic gain control (AGC) and devices subject to saturation or power limiting.

Amplitude Modulation The process, or result of the process, of varying the amplitude of a carrier signal. The resulting modulated carrier contains information that can be recovered by demodulation. See also Modulation.

Analog The general class of devices or circuits in which the output varies as a continuous function of the input.

Annotation The labeling of specific information on the display (such as frequency or power).

ANSI American National Standards Institute: A national membership organization (open to manufacturers, organizations, users, and communications carriers) that approves standards, accredits standards development groups and certificate programs, and represents and coordinates US interests in non-treaty and non-government
standards bodies.

**Aperture** The frequency span of the network analyzer used for calculating group delay. The narrower the aperture, the finer the resolution of the group delay variations, but noise is reduced by increasing the aperture.

**Array** A set of numbers or characters that represents any given function.

**ASCII** American Standard Code for Information Interchange

**Attenuation** Denotes a reduction in signal amplitude. The difference between transmitted and received power due to loss through equipment, lines, or other transmission devices; usually expressed in decibels.

**Attenuator** An RF or microwave device used to reduce the power level of a signal by precise, incremental amounts over its entire frequency range.

**Automatic Calibration System** AutoCal: Feature offered on Rohde & Schwarz network analyzers.

**Automatic Gain Control (AGC)** A circuit used in amplifiers and other active devices to keep its RF power level constant as other parameters change, such as frequency. Synonym: Automatic Leveling Control (ALC)

**Autoscale** An analyzer feature that evaluates waveforms and adjusts controls to stable and enhance the display.

**AUX** Auxiliary; refers to rear-panel input connector.

**Averaging** A noise reduction technique that computes each data point based on consecutive sweeps and weighted by a user-specified averaging factor. Each new sweep is averaged into the trace until the total number of sweeps is equal to the averaging factor.

**B**

**B/R** The ratio of data sampled at B to the data sampled at R.

**Band Pass** A range of frequencies that are passed through a device, such as a filter. Frequencies not within the band pass are limited or attenuated. See also **Cutoff Frequency**.

**Bandwidth (BW)** The difference between the frequencies of a continuous frequency band within which performance of a device falls within specifications.

**Bandwidth Limit** The condition prevailing when the system bandwidth is exceeded and signal distortion occurs beyond specifications.

**Bandwidth Selectivity** A measure of a filter's ability to resolve signals unequal in amplitude. It is the ratio of the 60 dB bandwidth to the 3 dB bandwidth for a given resolution filter (IF). Bandwidth selectivity tells us how steep the filter skirts are. Bandwidth selectivity is sometimes called shape factor.

**Binary** A method of representing numbers in a scale of two (on or off, high-level or low-level, one or zero). A compact, fast format used to transfer information to and from the analyzer.

**BMP** Bit-Mapped

**Brightness** See **Color Brightness**.

**Broadband Device** A device that operates over a very wide frequency range and exhibits only small variations in response over that range.

**Buffer** A storage device used when transmitting information to compensate for a difference in the rate of flow of information between two devices.

**Burst Carrier** A carrier that is periodically turned off and on. A burst carrier may or may not be modulated.

**BUS** Basic Utility System

**Bus** One or more conductors used as a path to deliver transmitted information from any of several sources to any of several destinations.
BW Bandwidth

Byte Eight bits of data representing one character processed as a unit.

C

CAD Computer Aided Design

CAE Computer Aided Engineering

Calibration In HP instrumentation, the process of periodically (usually annually) verifying an instrument is performing to specifications. A calibration certificate is awarded after verification.

In network analyzers, the process of removing systematic errors from measurements. See Error Correction.

Calibration Kit Hardware and software required to perform error correction on a network analyzer for a specific measurement and/or test set.

Calibration, 2-Port See Error Correction, 12-Term.

Calibration, Blackburn Calibrations of transmission path with corrected source match involving 15 calibration terms. Synonym: 15-term error correction

Calibration, Frequency Response The simplest error correction procedure to perform, but only corrects for a few of the twelve possible systematic error terms. Frequency response corrections can be made for reflection measurements, transmission measurements, and isolation measurements.

Calibration, Interpolation A user selectable network analyzer feature that calculates (interpolates) new error correction terms from existing terms when there is a change in network analyzer parameters, such as IF bandwidth, power, or sweep time. The resulting error correction is not as accurate as completing a full 2-port calibration.

Calibration, Port Extension See Port Extension.

Calibration, Reference Plane See Reference Plane.

Calibration, Set Z Sets the system impedance, usually 50 or 75 ohms.

Calibration, SOLT A calibration using four known standards: Short-Open-Load-Through. Also known as a full two-port calibration and 12-term error correction. See also Error Correction.

Calibration, TRL and LRM A calibration used in environments where the DUT cannot be connected directly to the network analyzer ports, (MMIC, microstrip, beam-lead diodes etc.). Thru-Reflect-Line (TRL) and M (Match) standards are fabricated and used because known high-quality standards are not readily available. The requirements for characterizing these standards are less stringent, but the calibration is not as accurate as the traditional full two-port calibration using S-O-L-T standards. The terms are used interchangeably (TRL, LRL, LRM etc.) but they all refer to the same basic calibration method.

Characteristic Impedance The impedance looking into the end of an infinitely long lossless transmission line.

Color Brightness A measure of the intensity (brightness) of a color.

Command A set of instructions that are translated into instrument actions. The actions are usually made up of individual steps that together can execute an operation.

Continuous Sweep Mode The analyzer condition where traces are automatically updated each time trigger conditions are met.

Controller A device capable of specifying the talker and listeners for an information transfer. An external computer connected to an instrument to control its operation.

Corrected Measurements made after performing error correction.

Coupler See Directional Coupler.
CPU Central Processing Unit
Crosstalk The occurrence of a signal at one port of a device being affected by a signal in any other path. Isolation is the measurement of crosstalk.
Cursor An electronically generated pointer that moves across the display to manipulate controls.
Cutoff Frequency In filters, the frequency at which attenuation is 3dB below the band pass signal level, known as the 3dB points.
CW Continuous wave: A single frequency (rather than a swept frequency).

D
DAC Digital to Analog Converter
dB Decibel: a relative unit of measure. The ratio in dB is given by: \(10 \log_{10} \left( \frac{P_1}{P_2} \right)\) where \(P_1\) and \(P_2\) are the measured powers. The dB is preferred instead of arithmetic ratios or percentages because when components are connected in series, their effect on power, expressed in dB, may be arithmetically added and subtracted. For example, if a 3dB attenuator is connected to a 10dB amplifier, the net gain of the two components is \((-3dB + 10dB = +7dB)\).

dBm Absolute unit of measure in decibels: 0dBm = 1 mW. The conventions of the dB (adding and subtracting) continue to apply.
DBMS Database Management System
DC Direct Current
Default A known set of conditions used in the absence of user-defined conditions.
Delay See Group Delay.
Demodulation The process of recovering from a modulated carrier, information in the form of a signal having essentially the same characteristics as the original modulating signal. Recovery of the modulating signal accomplished by signal detection.
Detection The process of demodulating signal carriers. There are two basic ways of providing signal detection in network analyzers: Diode detectors (used in broadband applications) and heterodyning, (used in narrowband applications).
Detector, Diode A device used to convert a RF signal to a proportional DC level. If the signal is amplitude modulated, the diode strips the RF carrier signal from the modulation. Many sources used with scalar analyzers are amplitude modulated with a 27.778 kHz signal and then detected in the network analyzer. Phase information on the signal carrier is lost in diode detection.

Deviation from Linear Phase Linear phase refers to the nature of the phase shift of a signal through a device. The phase is linear if a plot of phase shift versus frequency is a straight line using linear scales. Deviation from linear phase causes signal distortion.
Digital Pertaining to the class of devices or circuits in which the output varies in discrete steps.
Digital Demodulation Describes a technique of extracting the information used to modulate a signal. Digital signal processing algorithms are used on the signal after it has been converted from an analog to a digital form (digitized).
Dimension To specify the size of an array. The number of array rows or columns.
Directivity In a 3-port directional coupler, the ratio of the power present at the auxiliary port when the signal is traveling in the forward direction to the power present at the auxiliary port when the same signal is traveling in the reverse direction.
Directional Coupler A 3-port device typically used for separately sampling the backward (reflected) wave in a
transmission line.

**Disk** A circular, magnetic storage medium.

**Display**

Noun: See **Screen**.

Verb: To show annotation and measurement data on the display.

**Display Detector Mode** The manner in which analog, video information is processed prior to being digitized and stored in memory.

**Display Dynamic Accuracy** The amplitude uncertainty, usually in dB, over the display dynamic range.

**Display Dynamic Range** The amplitude range, in dB, over which the display dynamic accuracy applies.

**Display Formats** Graphical formats for displaying measurement data. These include single channel, overlay (multiple traces on one graticule), split (each trace on separate graticules).

**Display Modes** The ways in which measurement data can be presented graphically. On a network analyzer, the choices are Cartesian/rectilinear (XY plot with log or linear magnitude, phase, group delay, SWR, real and imaginary, and dBV, dBmV and dBuV), polar (magnitude and angle), magnitude and phase, and Smith chart. Not all display modes are available on all network analyzers. In addition, displays can present this information in various combinations of traces. Common modes are dual, (the ability to display more than one trace, usually over the same frequency range), and alternate, (the ability to display more than one trace, each with different frequency range and type).

**Display Phase Dynamic Accuracy** The phase measurement uncertainty, usually in degrees, for measurements whose units are in degrees.

**Display Points** The total number of measurement points made in a single measurement. The points can be in units of frequency, power, or time. The number of points often dictates measurement speed, resolution, and aperture.

**Display Trace Noise, Magnitude** The amplitude uncertainty of the trace, in dB, due to random noise in the test system.

**Display Trace Noise, Phase** The phase uncertainty of the trace, in degrees, due to random noise in the test system.

**Display Type** The type of display screen built into the analyzer. Data can be displayed as a raster drawing (a computer-like dot map) or as a vector drawing (lines drawn on the display). Color and display standard can also be specified as monochrome (single color), or color (two or more colors). The format standard may also be specified, such as VGA or SVGA, for IBM-compatible personal computers.

**Distortion** Deterioration of a signal's quality due to the nonlinear characteristics of a device or system transfer function. Distortion is measured as a combination of the changes in amplitude, frequency and phase of signal at the output of a device or system as compared to the signal at the input.

**Drift** The slow change in signal frequency.

**DSP** Digital Signal Processing

**DUT** Device Under Test

**DVM** Digital Volt Meter

**Dynamic Range** In a receiver, the range of signal levels, from minimum to maximum, that can be reliably measured simultaneously. Dynamic range allows small signals to be measured in the presence of large signals. Source power and receiver compression usually limits the maximum boundary to dynamic range. Receiver residual responses and noise floor usually limit the minimum power boundary.
See Electronic Calibration.

Electrical Delay A simulated variable length of lossless transmission line, added to or subtracted from a receiver input, to compensate for interconnecting cables. The firmware equivalent of mechanical or analog "line stretchers" in other network analyzers.

Electronic Calibration (ECal) A calibration system for electronic calibration of RF and microwave vector network analyzers. The electronic calibration system creates a twelve-term, two-port error model and then provides a confidence check of the calibration. The Ecal system consists of a repeatable, variable-impedance, solid-state calibration standard and a mainframe control unit which interfaces with the 8510, 8720 series, and the 8753 network analyzers or a USB module which interfaces with the PNA series network analyzers.

EMC Electro-Magnetic Compatibility

EMI Electro-Magnetic Interference: Unintentional interfering signals generated within or external to electronic equipment. Typical sources could be power-line transients, noise from switching-type power supplies and/or spurious radiation from oscillators. EMI is suppressed with power-line filtering, shielding, etc.

Engage To activate a function.

Enter The process of inputting information.

EPROM Electronically Programmable, Read-Only Memory

Error Correction In network analyzers, a process that removes or reduces systematic (repeatable) measurement errors by measuring known standards from a calibration kit. Synonym: measurement calibration

Error Correction, 3-Term Used to remove systematic measurement errors on a device with one port, such as a load.

Error Correction, 12-Term Correction for a two port device using six parameters: Directivity Source match Load match Reflection frequency response Transmission frequency response Isolation

To completely characterize a two-port device, these six parameters must be characterized in the forward and reverse directions, making a total of 12 terms. The user usually has the option of omitting isolation from the correction process. Synonym: Full two-port error correction

Error Correction, 1-Port Corrects a test set for port 1 or port 2 directivity, frequency response, and source match errors. The process requires three known standard terminations, for example, open, short, and load.

Error Message A message on a display that indicates an error condition. Missing or failed hardware, improper user operation, or other conditions that require additional attention can cause an error condition. Generally, the requested action or operation cannot be completed until the condition is resolved.

ESD Electro Static Discharge

Ethernet A network that adheres to the IEEE 802.3 Local Area Network standard.

Ethernet address A hexadecimal number which is used to identify a machine on a network. Each analyzer is assigned a unique Ethernet address at the factory and it is stored in the analyzer's ROM.

External trigger signal A TTL signal that is input to an analyzer and initiates a measurement sweep or similar event, making the measurements synchronous with the external triggering source.
Filter  A passive device that allows some frequencies to pass and attenuates others, depending on the type and specifications. A high-pass filter passes frequencies above the cutoff frequency, a low-pass filter passes frequencies below the cutoff frequency, and a band-pass filter passes frequencies between two specific frequencies.

Firmware  An assembly made up of hardware and instruction code. The hardware and instruction code is integrated and forms a functional set that cannot be altered during normal operation. The instruction code, permanently installed in the circuitry of the instrument, is classified as ROM (read only memory). The firmware determines the operating characteristics of the instrument or equipment.

Flatness  The amplitude and phase response of a device under test (DUT), a signal source, a receiver, or a combination of these. See also Frequency Response.

FM  Frequency Modulation

Frequency  The number of periodic oscillations, vibrations, or waves per unit of time, usually expressed in cycles per second, or Hertz (Hz).

Frequency Accuracy  The uncertainty with which the frequency of a signal or spectral component is indicated, either in an absolute sense or relative to another signal or spectral component. Absolute and relative frequency accuracies are specified independently.

Frequency Range  The range of frequencies over which a device or instrument performance is specified.

Frequency Resolution  The ability of a network analyzer to measure device characteristics at closely spaced frequencies and display them separately. Resolution of equal amplitude responses is determined by IF bandwidth. Resolution of unequal amplitude responses is determined by IF bandwidth and bandwidth selectivity.

Frequency Response  The peak-to-peak variation in the displayed amplitude response over a specified center frequency range. Frequency response is typically specified in terms of dB, relative to the value midway between the extremes.

Frequency Span  The magnitude of the displayed frequency component. Span is represented by the horizontal axis of the display. Generally, frequency span is given as the total span across the full display. Some analyzers represent frequency span (scan width) as a per-division value.

Frequency Stability  The ability of a frequency component to remain unchanged in frequency or amplitude over short and long-term periods of time. Stability refers to an oscillator's ability to remain fixed at a particular frequency over time.

Front Panel Key  Keys that are located on the front panel of an instrument. The key labels identify the function the key activities. Numeric keys and step keys are two examples of front panel keys.

Full 2-Port Calibration  See Error Correction, 12-Term.

Function  The action or purpose that a specific item is intended to perform or serve. The network analyzer contains functions that can be executed via front panel key selections, or through programming commands. The characteristics of these functions are determined by the firmware in the instrument. In some cases, a DLP (downloadable program) execution of a function allows you to execute the function from front panel key selections.

Fundamental Frequency  In any waveform, the lowest frequency component; all other components are harmonics. A pure sinusoid has only one component, the fundamental.

Gb  Gigabit

GB  Gigabyte
GHz Gigahertz
GIF Graphics Interchange Format - Standard graphic format to store bitmapped graphics files.
Giga Prefix for one billion.
GP I/O General Purpose Input / Output; a connector usually on the back of an instrument that allows communication with other test equipment, external test sets, switches, and computers that enable the instrument to be triggered or to trigger external equipment. An example is a foot switch that continues or cycles a measurement, allowing the operator to use both hands on the test hardware.
GPIB General Purpose Interface Bus - IEEE 488 bus is interconnect bus and protocol, allows linking of instruments and computer.
Graticule (or Grid) Enclosed area where waveform is displayed on instrument. Tick marks, on frame or axis, are a scaling aid for making visual measurements.
Group Delay A measure of the transit time of a signal through a DUT versus frequency. Group delay can be calculated by differentiating the DUT’s insertion-phase response with respect to frequency. See also AM Group Delay and Deviation from Linear Phase.
GUI Graphical User Interface

H
Hardcopy Paper copy of data.
Hardkey A front-panel key, which engages a single analyzer function or presents a single menu of softkeys.
Horizontal Reference See Reference Level.
Horizontal Resolution The analyzer’s ability to take closely spaced horizontal data points over the full sweep.
Host Computer A computer or device on a network that provides end users with services such as computation and database access and that usually performs network control functions.
Host Name A unique name that is used to identify each host machine on a network. The host name is directly linked to, and can usually be used in place of, the IP address. The user or the system administrator usually creates the host name.
HP Hewlett-Packard Company
HPGL Hewlett-Packard Graphics Language
HP-IB Hewlett-Packard Interface Bus. A parallel interface that allows "daisy chaining" of more than one device to a port on a computer or instrument. Interface protocol is defined in IEEE 488.2; equivalent to the industry standard GPIB.
HTTP HyperText Transfer Protocol: Used to carry World Wide Web (WWW) traffic.
Hue The dimension of color referred to a scale of perceptions ranging from red through yellow, green, and blue, and back to red. A particular gradation of color, tint, shade.

I
I/O Input/Output
I/O Path Input/Output Path
IEEE Institute of Electrical and Electronic Engineers
IF Intermediate Frequency: the frequency at which a signal is processed after mixing.
Impedance  The ratio of voltage to current at a port of a circuit, expressed in ohms.

Initialize  The process that assigns information locations to a disk to prepare the magnetic media to accept files.

Input  A path intended for putting a signal into an instrument.

Most network analyzers have either 3 (labeled A, B, and R) or 4 inputs (labeled A, B, R1, and R2). Inputs are not the same as channels.

Input Attenuator  An attenuator between the input connector and the first mixer of a spectrum analyzer (also called an RF attenuator). The input attenuator is used to adjust the signal level incident to the first mixer, and to prevent gain compression due to high-level or broadband signals. It is also used to set the dynamic range by controlling the degree of internally-generated distortion. For some analyzers, changing the input attenuator settings changes the vertical position of the signal on the display, which then changes the reference level accordingly. In Agilent microprocessor-controlled analyzers, the IF gain is changed to compensate for changes in input attenuator settings. Because of this, the signals remain stationary on the display, and the reference level is not changed.

Insertion Loss  The difference between the power measured before and after the insertion of a device. The attenuation between the input and output of a device.

Intensity  Brightness; emitting or reflecting light; luminosity.

Interface  A connection that allows a common communication link between two or more instruments.

Intermodulation Distortion  Undesired frequency components resulting from the interaction of two or more spectral components passing through a device having nonlinear behavior, such as a mixer or an amplifier. The undesired components are related to the fundamental components by sums and differences of the fundamentals and various harmonics. The algorithm is: f1 ± f2, 2xf1 ± f2, 2xf2± f1, 3xf1 ± 2x f2, and so on.

Internet  The connection of two or more distinct networks. Often a gateway or router is used to make the connection.

Interpolate  To determine a value of a signal between to adjacent points by a procedure or algorithm.

IP  Internet Protocol

IP Address  Internet protocol address: a unique number that is assigned to each device which is to be connected to a TCP/IP network. Before using an analyzer on a network, your network administrator will need to assign an IP address. An IP address consists of a 32-bit value presented in decimal dot notation: 4 octets (bytes) separated by a dot.

ISDN  Integrated Services Digital Network: A standard digital service capability that features one or more circuit-switched communication channels capable of carrying digital voice, data, or image signals, a packet-switched channel for out-of-band signaling and control. In addition, ISDN provides a collection of standard and optional features that support information productivity for the user, providing higher-speed Internet access than analog systems.

ISO  International Standards Organization

Isolation  A specification or measure of the immunity that one signal has to being affected by another adjacent signal. The occurrence is known as crosstalk.

Isolator  An RF device used for providing isolation between paths and components. Made from a 3-port circulator, the third port being terminated in a 50ohm load.
Kilo Prefix for one thousand.
KB Kilobyte
Kb/s Kilobytes per second

L
LAN Local Area Network
LANS Local Area Network System
LCD Liquid Crystal Display
LED Light Emitting Diode
LIF Logical Interchange Format (used for older HP disk drives/computers)
Limit Lines Lines input by the user that overlay the analyzer's measurement data to allow automatic detection of data that is out of the acceptable range. Pass/Fail annotation, audio alarms, or electronic output can be triggered to notify the operator or on-line computer program of the over-limit condition.
Limit-Line File The user-memory file that contains the limit-line table entries.
Limit-Line Table The line segments of a limit line are stored in the limit-line table. The table can be recalled to edit the line segments, then restored in the limit-line file.
Linear Device A device in which the output is continuously proportional to the input.
LO Local Oscillator. In a superheterodyne system, the LO is mixed with the received signal to produce a sum or difference equal to the intermediate frequency (IF) of the receiver.
LO Feedthrough The response that in a superheterodyne system when the first local oscillator frequency is equal to the first IF.
Load A one port microwave device used to terminate a path in its characteristic impedance.
Load Match A measure of how close the device's terminating load impedance is to the ideal transmission line impedance. Match is usually measured as return loss or standing wave ratio (SWR) of the load.
Local Lock Out A condition or command that prevents analyzer front-panel entries (and disables the Local key).
Local Operation To operate manually from the front panel.
Log Logarithm
Log Display The display mode in which vertical deflection is a logarithmic function of the input signal amplitude.
Log display is also called logarithmic display. The display calibration is set by selecting the value of the reference level position and scale factor in dB per division.
LRM Line-Reflect-Match. See Calibration, TRL, and LRM.

M
Magnitude The amplitude of a signal measured in its characteristic impedance without regard to phase. See also Scalar.
Marker A graphical symbol along a display trace that is annotated with measurement characteristics of that specific data point.
Marker Functions Mathematical or statistical computation on the data of one or more markers to provide the operator more information. For example, the marker delta function calculates and displays the difference between two markers.
**Maximum Input Level** The maximum signal power that may be safely applied to the input of an analyzer. The maximum input level is typically 1 W (+30 dBm) for Agilent spectrum analyzers.

**MB** Megabyte

**Measurement Uncertainty** The quantified amount of error in a measurement situation. Calibrations are intended to reduce the amount of uncertainty. The following are sources of measurement errors that lead to uncertainty:

- Systematic errors (imperfections in calibration standards, connectors, cables, and instrumentation)
- Random errors (noise, connector repeatability)
- Drift (source and instrumentation)

**Mega** Prefix for one million.

**Memory** A storage medium, device, or recording medium into which data can be stored and held until some later time, and from which the entire original data may be retrieved.

**Memory Card** A small memory device shaped like a credit card that can store data or programs.

**Menu** The analyzer functions that appear on the display and are selected by pressing front panel keys. These selections may invoke a series of other related functions that establish groups called menus.

**MHz** Megahertz

**milli** Prefix for one-thousandth.

**Modem** Modulator/Demodulator

**Modulation** The process, or the result of the process, of varying a characteristic of a carrier signal with an information-bearing signal, causing the carrier to contain the information. See AM and FM.

**Monitor** Any external display.

**Monochrome** Having only one color (chromaticity).

**ms** Millisecond

**mW** Milliwatt: one thousandth of a watt

**Multisync** A type of monitor that can synchronize its horizontal sweep to various frequencies within a specified range.

**N**

**Narrowband** In network analysis, the frequency resolution of the analyzer's receiver that is sufficiently narrow to resolve the magnitude and phase characteristics of narrowband devices. The reduced receiver bandwidth usually decreases the noise floor of the receiver, providing more measurement amplitude range.

**Narrowband Device** A device whose transfer characteristics are intended to operate over a very narrow frequency range and are designed to provide well-defined amplitude responses in that range, such as a band pass filter.

**Network Analysis** The characterization of a device, circuit, or system derived by comparing a signal input going into the device to a signal or signals coming out from the device.

**NIST** National Institute of Standards and Technology

**Nit** The unit of luminance (photometric brightness) equal to one candela per square meter.

**Noise** Random variations of unwanted or disturbing energy in a communications system from man-made and natural sources that affects or distorts the information carried by the signal. See also Signal-to-Noise Ratio.
**Noise Figure** (F): For a two-port device, a measure of how the noise generated inside the device degrades the signal-to-noise ratio of a signal passing through the device at 290 degrees, usually expressed in dB.

**Noise Floor** The analyzer’s internal displayed noise. The noise level often limits how small a signal magnitude can be measured. In network analysis, noise floor is measured with the test ports terminated in loads, full two-port error correction, 10 Hz IF bandwidth, maximum test port power, and no averaging during the test.

**Non-Insertable Devices** In measurement calibration, a device that cannot be substituted for a Zero-Length Through Path. It has the same type and sex connectors on each port, or a different type of connector on each port.

**Nonvolatile Memory** Memory data that is retained in the absence of an ac power source. This memory is typically retained with a battery. Refer also to battery-backed RAM.

**Normalize** To subtract one trace from another to eliminate calibration data errors or to obtain relative information.

---

**Offset** To move or set off a determined amount. Used in instruments for offsetting frequencies, limits, delay, loss, impedance, etc.

**Output Attenuation** The ability to attenuate the signal, the source, in order to control its power level.

---

**PC** Personal Computer

**PDF** Portable Document Format (used on the Web)

**Parser, Command** Reads program messages from the input queue of a device in the order they were received from the controller. The parser determines what actions the analyzer should take. One of the most important functions of the command parser is to determine the position of a program message in the analyzer SCPI command tree. When the command parser is reset, the next element it receives is expected to arise from the base of the analyzer command tree.

**Peak Search** A function on an analyzer that searches for the largest response and places a marker on it.

**Phase** The fractional part of a cycle through which an oscillation has advanced, measured from an arbitrary starting point; usually measured in radians or degrees. In network analysis, the phase response of the device under test is the change in phase as a function of frequency between the input stimulus and the measured response.

**Port** The physical input or output connection of an instrument or device.

**Port Extension** Redefining the reference plane to other than that established at calibration. A new reference plane is defined in seconds of delay from the test set port.

**Positive Peak** The maximum, instantaneous value of an incoming signal.

**Postscript (.ps files)** Stores bitmapped graphics files in an encapsulated format for direct use by postscript printers.

**Power, Max Input** The upper limit to input power for which the specifications apply. Some specifications may have different levels of maximum inputs. For example, compression power maximum is usually higher than the harmonic distortion maximum.

**Power, Safe Input** The input power, usually in dBm, allowed without damaging the instrument.

**Preset** A pre-defined instrument state (that also runs an analyzer self-test). The action of pushing the Preset key.

**Protocol** A set of conventions that specify how information will be formatted and transmitted on a network, and how machines on a network will communicate.
**Q**

**Q or Q Factor** The ratio of energy stored to energy lost in a resonant circuit. High Q indicates a sharp resonance response over frequency.

**Query** Any analyzer programming command having the distinct function of returning a response. These commands may end with a question mark (?). Queried commands return information to the computer.

---

**R**

**r + jx** Expression for complex impedance, where r represents the resistive portion and x represents the reactive portion.

**R Channel** Reference Channel

**RAM** Random Access Memory, or read-write memory: A storage area allowing access to any of its storage locations. Data can be written to or retrieved from RAM, but data storage is only temporary. When the power is removed, the information disappears. User-generated information appearing on a display is RAM data.

**ROM** Read Only Memory: A storage area that can be read only; it cannot be written to or altered by the user. In instruments, the storage area that contains the "brains" or operational programming; the firmware.

**Receiver** A circuit or system designed for the reception and/or measurement of signals in a specified frequency spectrum.

**Receiver Dynamic Range** See Dynamic Range.

**Reference Level** An instrument function that allows the user to set the amplitude value at the reference position. On network analyzers, the reference position is also selectable. On some spectrum analyzers, the reference position is fixed at the top of the display.

**Reference Plane** The electrical location at which a network analyzer assumes the system connectors and fixturing ends and the DUT begins. The reference plane is set by using calibration standards with known electrical length. The closer the reference plane is to the device under test (DUT), the better the characterization of the device because of the elimination of test system uncertainties.

**Reference Receiver** In a network analyzer, the receiver that measures signals as they come out of the source, before they are incident on the test port and DUT. Typically, these signals are used to compare with the signal at the Test Port Receiver, to determine the affect that the DUT has on the signal. In a 2-port network analyzer, these are typically named ‘R1’ (port 1) and ‘R2’ (port 2). See a block diagram of the receivers in your PNA.

**Reflection** The phenomenon in which a traveling wave strikes a discontinuity and returns to the original medium.

**Reflection Coefficient** The ratio of the reflected voltage to the incident voltage into a transmission line or circuit. If a transmission line is terminated in its characteristic impedance, the reflection coefficient is zero. If the line is shorted or open the coefficient is 1. See also Return Loss and SWR.

**Reflection Measurements** Measurements that characterize the input and /or output behavior of the device under test (DUT). Measured as the ratio of the reflected signal to the incident signal as a function of frequency. Parameters are called return loss, reflection coefficient, impedance, and standing wave ratio (SWR), all as a function of frequency. See also S-Parameters.

**Remote** A mode of operation where another device (or computer) controls an instrument via the HP-IB. In this mode, the instrument front panel keys are disabled. Front panel operation is called local operation.

**Remote Programming** The automatic operation of an instrument by a computer, usually through a HP-IB, LAN, or RS-232 link.

**Resolution** The ability of a receiver to resolve two signals.
Resolution Bandwidth The ability of a spectrum analyzer to display adjacent responses discretely (Hertz, Hertz decibel down). This term is used to identify the width of the resolution bandwidth filter of a spectrum analyzer at some level below the minimum insertion loss point (maximum deflection' point on the display). Typically, it is the 3 dB resolution bandwidth that is specified, but in some cases the 6 dB resolution bandwidth is specified.

Return Loss The amount of dB that the reflected signal is below the incident signal. If zero signal is reflected, the impedance of the device is equal to the characteristic impedance of the transmission system, and return loss is infinite. If the entire incident signal is reflected, the return loss is zero. See also S-Parameters, Reflection Coefficient, and SWR.

Reverse Measurement The measurement of a device from output to input.

RF Radio Frequency (from approximately 50 kHz to approximately 3 GHz). Usually referred to whenever a signal is radiated through the air.

ROM Read Only Memory

S

S/N Signal-to-Noise Ratio

Sampler An electronic component that captures the signal level and phase across a known impedance at a uniform rate. In Network Analyzers, this sampling rate must be sufficiently high and precisely timed to make accurate measurements. Network analyzers typically have three or four samplers or mixers.

Sampler Bounce The leakage or crosstalk between a network analyzer's samplers. Delay in this crosstalk caused by leakage transmission propagation, give the interference its "bounce" appearance. Sampler bounce causes an increase in the noise level of the affected channel, reducing the sensitivity of the analyzer.

Saturation The degree of color purity, on a scale from white to pure color.

Scalar A quantity that has magnitude but no phase. A network analyzer capable of measuring only magnitude.

Scale Factor The display vertical axis calibration in terms of units per division.

SCPI Standard Commands for Programmable Instruments

Screen The physical surface of the CRT or flat panel upon which the measurement results, setup information, softkey definitions, and other instrument communication is presented.

Self-Test A group of tests performed at power-up (or at preset) that verify proper instrument operation.

Sensitivity The minimum input signal required to produce a specified output signal having a specified signal-to-noise ratio, or other specified criteria.

On a spectrum analyzer, the level of the smallest sinusoid that can be observed, usually under optimized conditions of minimum resolution bandwidth, 0 dB input attenuation, and minimum video bandwidth.

The normalized change in YIG component's center frequency resulting from a change in tuning coil current, specified in MHz/mA.

Serial Prefix The five-character prefix that begins an instrument serial number; used to represent versions of firmware or hardware changes that have occurred.

Server A device that is configured to provide a service to other devices on a network, such as shared access to a file system or printer.

Signal-to-Noise Ratio SNR: The ratio of the amplitude of the desired signal to the amplitude of noise signals, usually expressed in dB and in terms of peak values for impulse noise and root-mean-square values for random noise.

Single Sweep Mode The spectrum analyzer sweeps once when trigger conditions are met. Each sweep is initiated
by pressing an appropriate front panel key, or by sending a programming command.

**Small Signal Gain Compression** A situation when the input signal’s measured amplitude is less than its actual level due to overloading of the network analyzer’s input mixer; the analyzer is operating nonlinearly. For broadband analyzer detectors, a signal other than the one under test can put the analyzer into this gain compressed mode, thereby making even lower level signals appear at a lower level than actual. The broadband mode measures all the power incident to the analyzer, not just the signals at the frequency of interest.

**Smith Chart** A graphical mapping of the complex reflection coefficient into normalized complex impedance. Circles on the chart represent constant resistance and radiating lines orthogonal to the circles represent constant reactance. The center of the chart represents the characteristic impedance of the transmission system. Any point on the chart defines a single complex impedance. A line on the chart represents changing impedance over frequency.

**SOLT** Short-Open-Load-Through calibration. See also Calibration, SOLT.

**Source** A device that supplies signal power; a sweep oscillator or synthesized sweeper.

**Source Amplitude Accuracy** The amplitude uncertainty, in dB, of the source power readout.

**Source Amplitude Flatness** The amplitude flatness, in dB, of the source power over the frequency range specified.

**Source Frequency Resolution** The smallest unit of frequency which can be set and/or measured, in Hz.

**Source Frequency Time Base Accuracy** A measure of the analyzer’s frequency stability measured in parts per million (ppm. or 1 part in 10^6). For example, a stability of ±5.0 ppm means that an analyzer will measure 1 MHz to an accuracy of ±5 × 10^-6 × 10^6 Hz = ±5 Hz.

**Source Frequency Time Base Stability** A measure of the analyzer’s time base accuracy over time and temperature. Typically the time base accuracy will be specified for 1 year. A typical temperature frequency stability is ±10 ppm for 250°C ± 50°C.

**Source Harmonics** The level of harmonics generated by the analyzer’s signal source, in dBc from the fundamental.

**Source Match** A measure of how close the signal source impedance is to the ideal transmission line impedance of the test system. Match is usually measured as return loss or standing wave ratio (SWR) of the source.

**Span** The stop frequency minus the start frequency. The span setting determines the horizontal-axis scale of the analyzer display.

**Span Accuracy** The uncertainty of the indicated frequency separation of any two signals on the display.

**S-Parameters (Scattering Parameters)** A convention used to characterize the way a device modifies signal flow using a network analyzer. A two port device has four S-parameters: forward transmission (S21), reverse transmission (S12), forward reflection (S11), and reverse reflection (S22).

**Stop/Start Frequency** Terms used in association with the stop and start points of the frequency measurement range. Together they determine the span of the measurement range.

**Storage States** The number of settings, programs, traces, and other parameters available to be saved, cataloged, and recalled at any one time.

**Storage, Disk** An internal or external digital storage disk for saving test data, instrument settings, IBASIC programs, and other measurement parameters. Storage formats include MS-DOS (R) and HPs standard LIF with binary, PCX, HP-GL, or ASCII data formats.

**Structural Return Loss** Poor return loss in cable due to a periodic fault such as a periodic dent caused by dropping the cable spool or by the cable pulling process during manufacture.

**Supplemental Characteristics** Typical but non-warranted performance parameters, denoted as "typical", "nominal" or "approximate".
**Sweep** The ability of the source to provide a specified signal level over a specified frequency range in a specified time period. Also see [Sweep Mode](#) and [Sweep Type](#).

In data processing mode, a series of consecutive data point measurements, taken over a sequence of stimulus values.

**Sweep Mode** The way in which a sweep is initiated or selected, e.g., single, continuous, alternate, or chopped.

**Sweep Type** The method of sweeping the source, e.g., linear, log, or frequency step.

**Sweeper** A signal source that outputs a signal that varies continuously in frequency.

**SWR** Standing Wave Ratio, calculated as \( \frac{1 + \pi}{1 - \pi} \) where \( \pi \) is the reflection coefficient.

**Sync** Synchronization, or Synchronized

**Syntax** The grammar rules that specify how commands must be structured for an operating system, programming language, or applications.

**System Dynamic Range** The difference between the maximum receiver input level and the receiver's noise floor. System dynamic range applies to transmission measurements only, since reflection measurements are limited by directivity.

T

**T/R** See [Transmission/Reflection](#).

**Termination** A load connected to a transmission line or other device.

**Test Limit** The acceptable result levels for any given measurement.

**Test Port** See [Port](#).

**Test Port Receiver** In a network analyzer, the receiver directly behind the test ports, used to measure the signal as it is reflected off, or transmitted through, the DUT. This signal is typically compared with the signal at the [Reference Receiver](#) to determine how the DUT affects a signal. In a 2-port network analyzer, these are typically named 'A' (port 1) and 'B' (port 2). See a block diagram of the receivers in your PNA.

**Test Set** The arrangement of hardware (switches, couplers, connectors and cables) that connect a test device input and output to the network analyzer’s source and receiver to make s-parameter measurements.

**Third Order Intercept** TOI: The power input to a non-linear device that would cause third order distortion at the same power level. TOI is a measurement to determine the distortion characteristics of a mixer or receiver. The higher the value, the more immune the receiver to internal distortion.

**Thru** Through line: A calibration standard. See [Calibration, SOLT](#).

**Tint** A shade of color; hue.

**Toggle** To switch states, usually to change a function from on to off, or off to on.

**TOM** Thru-Open-Match: A Rohde&Schwarz term to describe a calibration method.

**Trace** A series of data points containing frequency and response information. The series of data points is often called an array. The number of traces is specific to the instrument.

**Tracking** The ability of the analyzer’s receiver to tune to the source frequency over the measurement frequency range. Poor tracking results in amplitude and phase errors due to the receiver IF circuits attenuating and delaying the device under test output.

**Transfer Function** The ratio of the output signal to the stimulus signal, both as a function of frequency.

**Transmission** See [Transmission Measurements](#).
Transmission Intermodulation Spurious A measure of the capability of the transmitter to inhibit the generation of intermodulation distortion products. Intermodulation spurious is sometimes called intermodulation attenuation.

Transmission Measurements The characterization of the transfer function of a device, that is, the ratio of the output signal to the incident signal. Most common measurements include gain, insertion loss, transmission coefficient, insertion phase, and group delay, all measured over frequency. See also S-Parameters.

Transmission/Reflection (T/R) Refers to the suite of measurements made by a scalar or vector network analyzer to characterize a device's behavior over frequency. See also S-Parameters.

Transparent Something that is not visible to the user. Usually a procedure that occurs without the user's initiation or knowledge.

Trigger A signal that causes the instrument to make a measurement. The user can select several options for triggering, such as manual, continuous, or external (for synchronizing measurements to an external source).

TRL Through-Reflect-Line. See Calibration, TRL and LRM.

TTL Transistor-Transistor Logic

Two-Port Error Correction See Error Correction, 12-Term.

U

Uncorrected Measurements made without performing error correction.

Uncoupled Channels Stimulus or receiver settings allowed to be set independently for each channel.

UNI User-Network Interface: The point at which users connect to the network.

Units Dimensions on the measured quantities. Units usually refer to amplitude quantities because they can be changed. In analyzers with microprocessors, available units are dBm (dB relative to 1 mW dissipated in the nominal input impedance), dBmV (dB relative to 1 mV), dBW (dB relative to 1 1W), V (volts), W (watts).

V

Variable A symbol, the value of which changes either from one iteration of a program to the next, or within each iteration of a program.

Vector A quantity that has both magnitude and phase.

A network analyzer capable of measuring both magnitude and phase.

VEE Visual Engineering Environment (Agilent software product)

Velocity Factor A numerical value related the speed of energy through transmission lines with different dielectrics (.66 for polyethylene). Used in making time domain measurements.

Vertical Resolution The degree to which an instrument can differentiate amplitude between two signals.

Video An electrical signal containing timing, intensity, and often color information that, when displayed, gives a visual image.

Video Bandwidth In spectrum analyzers, the cutoff frequency (3 dB point) of an adjustable low-pass filter in the video circuit. When the video bandwidth is equal to or less than the resolution bandwidth, the video circuit cannot fully respond to the more rapid fluctuations of the output of the envelope detector. The result is a smoothing of the trace, or a reduction in the peak-to-peak excursion, of broadband signals such as noise and pulsed RF when viewed in broadband mode. The degree of averaging or smoothing is a function of the ratio of the video bandwidth to the resolution bandwidth.

Video Filter In spectrum analyzers, a post-detection, low-pass filter that determines the bandwidth of the video
amplifier. It is used to average or smooth a trace. Refer also to Video Bandwidth.

**VNA** Vector Network Analyzer

---

**W**

**Waveform** A representation of a signal plotting amplitude versus time.

**Wireless** A term that refers to a broad range of technologies that provide mobile communications for home or office, and "in-building wireless" for extended mobility around the work area, campus, or business complex. It is also used to mean "cellular" for in-or out-of-building mobility services.

**WWW** World Wide Web

---

**X**

---

**Y**

---

**Z**

**Zero-Length Through Path** In a measurement calibration, when the two test cables mate together directly without using adapters or a thru-line. See also Non-Insertable Devices.