

What's New

Notes

- The latest version includes changes from all previous versions.
- See [New Programming Commands](#)
- To see the code version, from the SFP click **Help**, then **About Network Analyzer**

What's New in PXIe Code Version A.03.00

- [Ground Loop embedding/de-embedding](#)
- [Trigger Delay](#)
- The following features are now standard:
 - [Fixturing](#)
 - [Port Z Conversion](#)
 - [4-Port De/Embed](#)
 - [Differential Z Conversion](#)
 - [Common Mode Z Conversion](#)
 - [Differential Port Matching](#)
 - [Power Compensation](#)
 - [Port Extensions](#)
 - [Equation Editor](#)
- [CalPod](#)
- [Cal Plane Manager](#)

What's New in PXIe Code Version A.02.00

- [Measurement Speed Enhancements](#)

Programming the M937xA

The following tips can help you successfully program the M937xA PXIe VNA.

SCPI (Standard Commands for Programmable Instruments)

The PXI VNA is programmable using the SCPI language. The [SCPI documentation](#) is included in this help file.

In the PXI VNA, SCPI commands **MUST** be sent through either **HiSLIP** or through the IVI-COM/Labview driver **pass-through**.

HiSLIP

The PXI VNA software supports the HiSLIP interface for SCPI commands. This **MUST** be enabled from the Soft Front Panel using the IO configuration dialog.

[Learn more](#).

Note: If you can NOT connect with an external PC using LAN (HiSLIP), add a single exception to the firewall – either for the 835x.exe or for port 4880.

IVI-COM / IVI-C / LabView Driver Interfaces

These drivers are the preferred programming interface to the PXI VNA software. The IVI COM driver is extensively documented in its own help file located at "c:\program files\ivi foundation\ivi\drivers\agna\agna.chm". The LabView driver is also documented. Both the driver and help file are included on the CD that was shipped with the product.

- There are numerous examples for using the IVI-COM driver to perform common operations. They are located at: "c:\program files\ivi foundation\ivi\drivers\agna\examples".
- Each of these drivers has a SCPI **pass-through** interface that can be used to send SCPI commands directly to the instrument.
- It is NOT necessary to launch the soft front panel to use these drivers.

Live Soft Front Panel (SFP)

By default, the SFP is NOT visible when using the driver interface to connect to the PXI VNA. However, both the remote program and the SFP can run simultaneously. This can be useful during development to help diagnose any

programming errors. Start the SFP by clicking the "Network Analyzer" icon on the controller desktop.

PXI VNA connection string

The PXI addresses are available in three locations:

1. through the **Launcher application** (Network Analyzer icon on the desktop)
2. through Agilent IO Libraries
3. through the VISA library by using `viFindRsrc()`.

From the **Launcher application** copy the address on the dialog. For a 1-module example, the PXI address may be `PXI15::0::0::INSTR`

If you know the chassis and slot number of your module, you can also format the connection string this way:

`PXI0::CHASSIS<Chassis#>::SLOT<Slot#>::INSTR`. For example, if you only have 1 chassis (this is most common), and the PXI VNA is in slot 9, then the connection string would be:

`PXI0::CHASSIS1::SLOT9::INSTR`

Speed up Measurements and Data Transfers

[Learn how.](#)

Multiple Modules

When using multiple PXI VNA modules, you can choose to use them in a Multisite or Multiport configuration.

For help configuring your hardware, please refer to the Installation Guide on the M937xA documentation website: <http://na.support.keysight.com/pxi/help/>

Multiport

When connecting to multiple modules, at least one of the modules must have option 551. The connection string for Multiport is a semicolon ";" delineated list of PXI addresses.

For example: The following creates a 4-port VNA using a VNA module in slot 9 combined and another in slot 10. Port 1 and Port 2 are measured on slot 9. Port 3 and Port 4 are measured on slot 10.

```
PXI0::CHASSIS1::SLOT9::INSTR;PXI0::CHASSIS1::SLOT10::INSTR
```

[Learn more about Multiport configurations.](#)

Multisite

When connecting to Multisite PXI VNAs, simply connect to each of them separately and configure them individually.

When running your program on the PXI controller using the IVI-COM, Labview, or IVI-C driver, you can use the **PXI VNA connection string** (shown above). The VNA firmware does not need to be started with the launcher GUI.

You can also connect over LAN using the HiSLIP address. First, you must enable the HiSLIP interface using the **GPIB dialog**. Then, you must launch the VNA modules using the Launcher GUI. Use a unique address to identify each of the VNA firmware instances as follows for 2 VNA modules.

- **TCPIP0::<hostname>::hislip0::INSTR**
- **TCPIP0::<hostname>::hislip1::INSTR**

[Learn more about Multisite configurations.](#)

Mix and Match

You can mix-and-match your Multisite and Multiport instances. For example,

with 4 PXI VNA modules, you can run a 6-port Multiport instance using 3 of your PXI VNA modules at the same time that you run a 2-port instance using the last module.

M937xA SFP Launcher

The M937xA SFP is unique in that this software IS the firmware that runs the modules. It does not send programming commands to the instrument, but instead communicates directly with the module hardware. It is a complete, FULL-FEATURED software package. This is a subtle difference, but has implications on how you will configure and connect to the VNA modules.

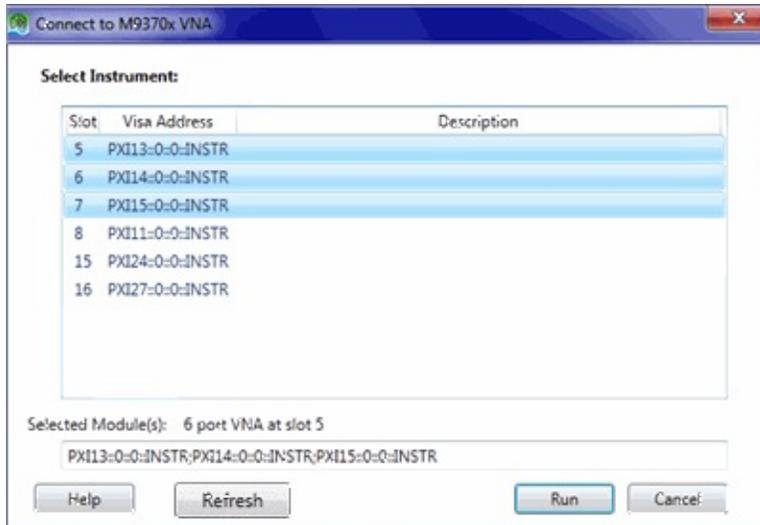
See Also: [Programming the M937xA](#)

How to start the SFP Launcher

Use any of the following methods:

- Click  on the host controller desktop to start the M937xA Soft Front Panel (SFP).
- **Keysight IO Libraries** (Keysight Connection Expert). This method can start only a single module.
- **Command Prompt:** This method does not allow the use of an alias, but ONLY the PXI VISA address. To do this, run 835x.exe with the PXI instrument name as the only argument. The following are TWO examples:
 - C:\program files (x86)\agilent\network analyzer\835x.exe
PXI14::0::0::INSTR;PXI10::0::0::INSTR **or**
 - C:\program files (x86)\agilent\network analyzer\835x.exe
PXI0::CHASSIS1::SLOT6::FUNC0::INSTR;PXI0::CHASSIS1::S

The following Launcher dialog appears. This dialog allows you to configure one or more M937xA VNA modules as a system.



Select the modules to comprise the PXI VNA. Learn how to configure **multiple modules** below.

Refresh - Click to update the list of PXI devices on the system. Do this when you have recently added or removed a module, or if the Launcher is not showing the devices as expected.

Run - Click to run the SFP software.

Multiple Modules

When more than one module is installed in the chassis, you can choose to configure the modules as a **Multiport VNA** or as **Multisite VNAs**.

Multiport VNA (Option 551)

With option 551 purchased for any single module, you can create a Multiport system as long as the single module with option 551 is included somewhere in the chassis.

For example:

The above dialog shows 6 modules installed in the chassis in slots 5, 6, 7, 8, 15, 16. The module in slot 5 (PXI13) has option 551. In this case, slots 5, 6, and 7 are selected. Click **Run**. Those three modules behave as a Multiport (6-port) VNA. That instance of the SFP, and your remote programs, will allow you to calibrate and make measurements at ports 1 through 6. Each test port has a source, measurement receiver, and reference receiver.

- The modules **MUST** be **physically** configured as a Multiport VNA. Do this by connecting the front-panel Ref In/Out, Trigger In/Out, and LO In/Out lines. Learn how in the M937xA Installation Guide at the [M937xA documentation website](#).
- The selected modules in a Multiport VNA **MUST** be sequential and must include at least ONE module with opt 551. For example, to configure a 6-port VNA using the above installed modules, you could choose Slots 5, 6, 7. You could **NOT** choose 5, 7, 8; they are not sequential. Nor could you choose 6, 7, 8; they do **NOT** include PXI13 which has option 551.
- Each VNA module has ports labeled 'Test Port 1' and 'Test Port 2'. In a Multiport VNA, test ports become virtually numbered beginning with the module in the lowest numbered slot to the highest numbered slot. In our example 6-port VNA, the module in slot 6 would be Test Port 3 on top, and Test Port 4 on bottom. The module in slot 7 would be Test Port 5 on top, and Test Port 6 on bottom.
- The first module, from left to right, is the Master module. To control the Reference, Triggering, and LO signals for the Multiport VNA, you make settings on the first module. The other modules follow.

- **Module Number** - A few remote commands require the module number. This is the order in which the module appears in the Multiport VNA. In the above example, Module 1 is in slot 5, module 2 is in slot 6, and so forth.
- The software supports up to 16 modules connected together to form a 32-port VNA.

See Also: [Programming Multiport and Multisite configurations](#)

Multisite VNA

While the 6-port VNA is running, you can also create a different, fully-functioning VNA which would run at the same time as the 6-port VNA. The only practical limit to the number of VNA instances is the CPU speed of the host controller. The benefit of creating Multisite VNAs is measurement throughput. You could test multiple devices at the same time.

Here is how to create a Multisite VNA:

1. With the first VNA already running, again click the desktop icon, which restarts the Launcher dialog.
2. The Launcher dialog shows the same selected slots as the previous time. The selection is 'sticky' to allow you to quickly run the same configuration as before.
3. Click each to unselect the previously-selected slots.
4. Then select one or more of the previously-unused VNA modules. In our example, this would be slots 8, 15, and 16.
5. Click **Run** and another SFP display appears.

Traces, Channels, and Windows on the Analyzer

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

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

Other Quick Start topics

Traces

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** times 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 perform **Trace Hold** (Max or Min)
- How to **Create** a new trace
- How to **Change** the trace parameter
- How to display a custom **trace title** (separate topic)
- How to display a **wide** active trace (separate topic)

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**.

Using Menu

1. Click the **Trace Status** label or trace.

Programming Commands

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**.

Using Menu

1. Right-click the **Trace Status** label, then click Delete.

Programming Commands

How to Move a trace to a different window

You can **DRAG** a trace from one window to another, or...

Using front-panel HARDKEY [softkey] buttons

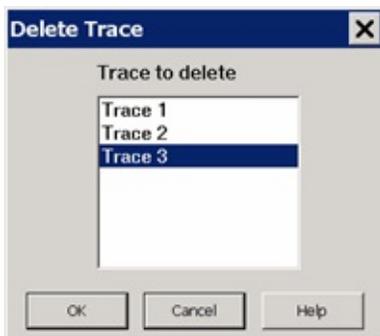
1. Select the trace to move.
2. Press **TRACES**
3. then **[Move Trace]**
4. Select a window number in the following dialog.

Using Menus

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

Select, Delete, Move Traces dialog box help

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



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 Hold

With Trace Hold (Max or Min) enabled, each data point is updated ONLY when the current data point measurement exceeds the highest (for Max) or lowest (for Min) currently-stored data point.

How to hold the active trace at the maximum or minimum points.	
Using front-panel HARDKEY [softkey] buttons <ol style="list-style-type: none"> 1. Press TRACE 2. Then Trace Hold 3. Restart resets the trace. 	Using Menus Not available
	

Maximum/Minimum trace hold can be applied with several conditions:

- Feature is applicable to any data trace, but NOT to memory traces.
 - When the stimulus or any data post processing setting is changed, the trace hold data will be reset. These settings include: Smoothing, Gating, Time Domain Transform, Conversion, Data Math, Equation Editor, Parameter change, Formatting change.
 - Minimum/maximum comparison is done with formatted data. For Smith and Polar formats, absolute data is used and not phase.
 - Trace hold data can be recalled.
 - Data save files formats
 - SnP does NOT save trace hold data
 - Citifile, CSV, MDF, PRN DOES save trace hold data
-

Channels

Channels contain traces. The analyzer can have up to **200 independent channels**.

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

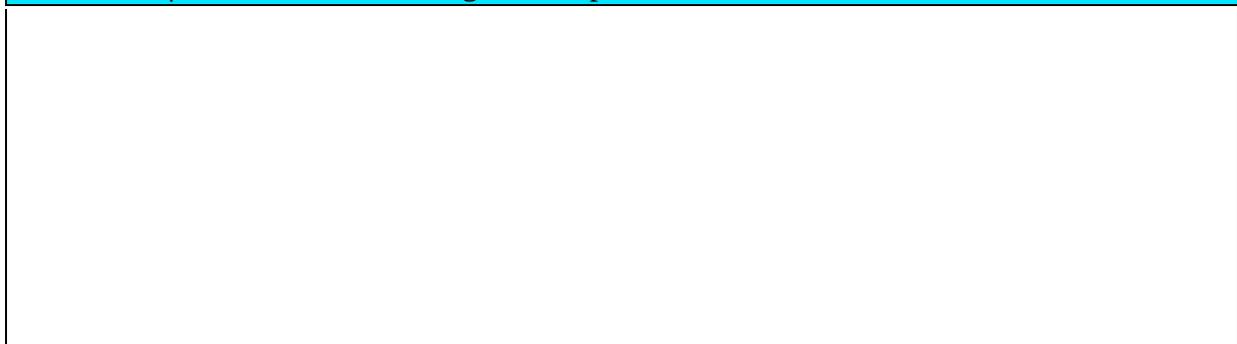
A channel must be selected (active) before its settings can be changed.

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





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.

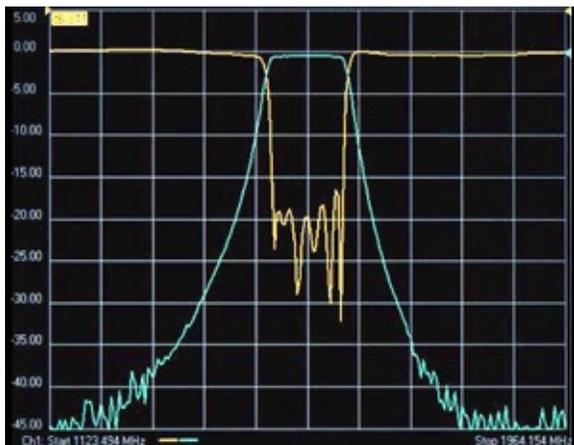
Note: To create more than one trace in a new channel, click Trace, then New Trace

Windows

Windows are used for viewing traces.

- The analyzer can show an **UNLIMITED** number of windows on the screen with the following limitations:
 - The **SCPI status register** can track the status of up to 576 traces.
- Each window can contain up to **24 traces**.
- Windows are completely independent of channels.
- Learn to **create and manage windows**.
- See **Customize the analyzer screen** to learn how to make other window settings.

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.



The window number shows 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

Using front-panel

Using Menus

HARDKEY [softkey] buttons

1. Press **RESPONSE**
2. then **[Display]**
3. then **[Windows]**

1. Click **Response**
2. then **Display**
3. then **Windows**

◀ 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, right-click in the window (away from a trace) then select **Close Window**.

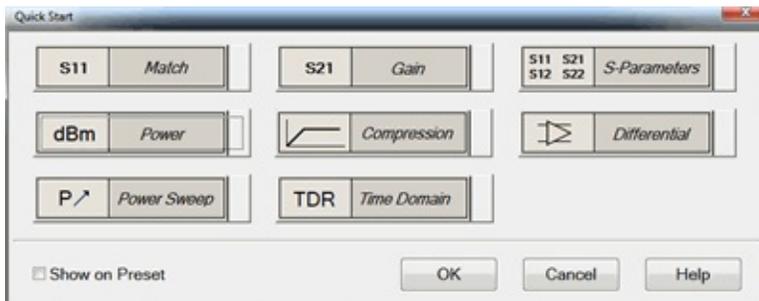
See [Customize the analyzer screen](#) to learn how to make other window settings.

Quick Start Dialog

To show the following dialog, click **Utility**, then **Quick Start**.

You can optionally see the following dialog when the analyzer is Preset.

- Click a button to quickly setup the measurement type.
- The buttons appear **ONLY** if that option is installed on the analyzer.



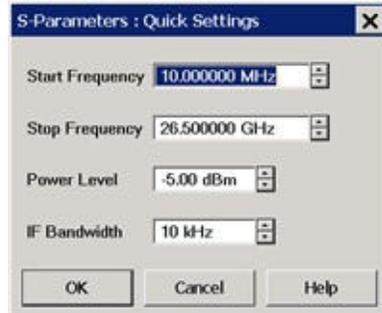
Click the image to learn more about each measurement.

- Clear **Show on Preset** to no longer see this dialog when Preset.
- To see the dialog again, press **Preset**, then **[Quick Start]**.

S-Parameters

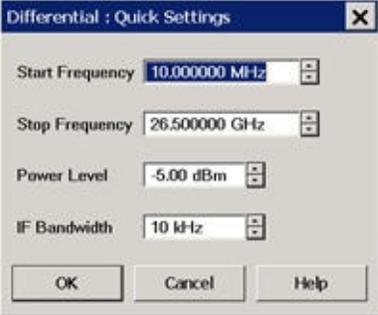
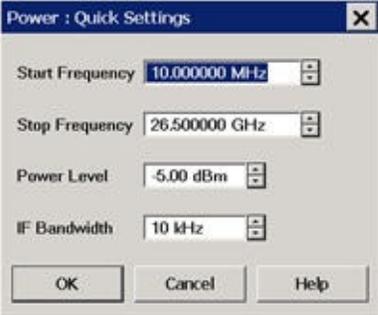
Required:
None

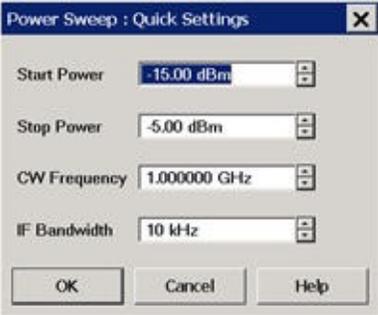
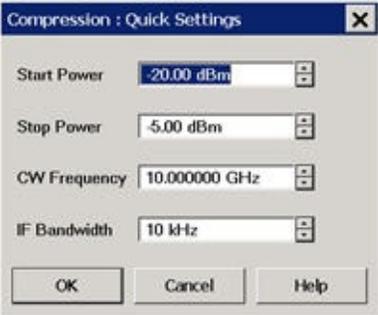
Creates S11 and S21 measurements in a single channel and window.



Enter:

- **Start/Stop frequency**
- **Power Level**
- **IF Bandwidth.**

	<p>Learn more about S-parameter measurements.</p>
<p>Differential Option Required: None</p>	<p>Creates Sdd11 and Sdd21 measurements in a single channel and window.</p>  <p>Enter Start/Stop frequency, Power Level, and IF Bandwidth.</p> <p>Learn more about Differential (Balanced) measurements.</p>
<p>Time Domain Option Required: 010</p>	<p>Creates an S11 measurements and enables Time Domain.</p>  <p>Enter Start/Stop Time</p> <p>Learn more about Time Domain measurements.</p>
<p>Power Option Required: None</p>	<p>Creates R1 and B receiver measurements in a single channel and window. This allows you to view the DUT input power (R1) and output (B) power.</p>  <p>Enter Start/Stop frequency, Power Level, and</p>

	<p>IF Bandwidth.</p> <p>Learn more about Unratioed Receiver measurements.</p>
<p>Power Sweep</p> <p>Option Required: None</p>	<p>Creates a power sweep while viewing R1, B, and S21 measurements in a single channel and window. This allows you to view the DUT input power (R1), output power (B), and DUT gain (S21).</p>  <p>Enter Start/Stop power, CW Frequency, and IF Bandwidth.</p> <p>Learn more about Power Sweep measurements.</p>
<p>Compression</p> <p>Option Required: None</p>	<p>Creates a power sweep while viewing S21 (gain) with Compression Markers.</p>  <p>Enter Start/Stop power, CW Frequency, and IF Bandwidth.</p> <p>Learn more about Gain Compression Markers.</p>

Basic Measurement Sequence

The following process can be used to setup all analyzer 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.

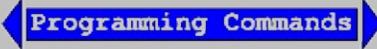
Step 5. Print, Save or Recall Data

Save or print the measurement data.

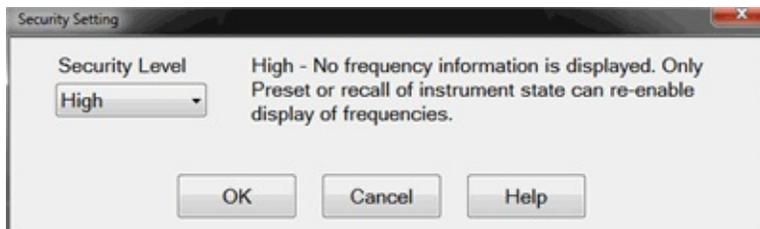
Frequency Blanking

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

How to set Frequency Blanking	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press SYSTEM2. then [Security]	<ol style="list-style-type: none">1. Click Utility2. then System3. then Security



Security Setting dialog box help



Notes

- To learn how to erase memory before moving your analyzer out of a secure area, see <http://na.support.keysight.com/pna/security.html>.
- An ECal Data Wipe Utility destroys all user data per US DoD 5220.22-M. Learn more at <http://na.support.keysight.com/pna/apps/applications.htm>
- PNA 'Undo' is disabled with **High** and **Extra** security levels. [Learn more](#).

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 Keysight 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**.

Preferences

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

How to set Preferences

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

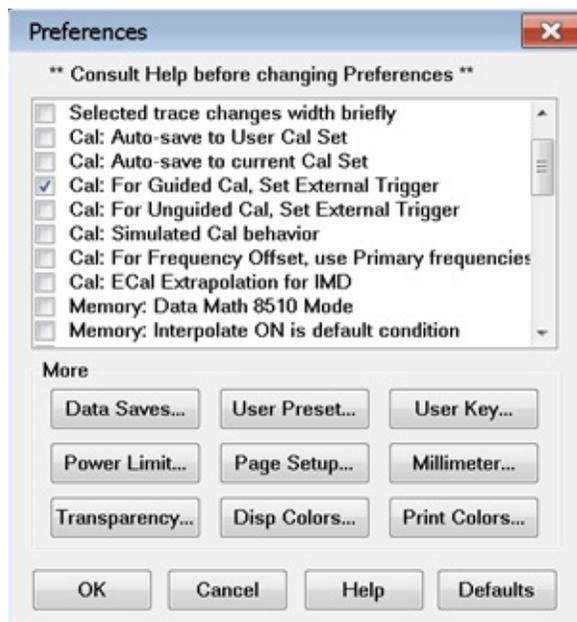
1. Press **SYSTEM**
2. then **[Configure]**
3. then **[More]**
4. then **[Preferences]**

Menu using a mouse

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

[Programming Commands](#)

Preferences dialog box help



Preferences survive a Preset and a Shutdown.

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

Note: The default setting is listed first.

NOT implemented for M937x models.

- Touchscreen ON.** Selections can be made by touching the screen.
- Touchscreen ON.** Selections can NOT be made by touching the screen.

- Selected Trace is wider.** The selected trace is the narrow, default size.
- Selected Trace is wider.** The active (selected) trace is always wider.

- Selected trace changes width briefly.** The selected trace does NOT change width briefly.
- Selected trace changes width briefly.**

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.

- Cal: Auto-save to User Cal Set** Completed calibrations are automatically saved to Cal Registers; NOT to User Cal Sets.
- Cal: Auto-save to User Cal Set** Completed calibrations are automatically saved to an auto-named User Cal Set. **Caution:** this can cause a lot of saved User Cal Sets. [Learn more.](#)

The following message appears when both the Cal Set choices above and below 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.

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

- Cal: Auto-save to current Cal Set**
- Cal: Auto-save to current Cal Set** Always automatically save a completed 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 (and unguided), 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.

Cal: For Guided Cal (and unguided), set external trigger. Do NOT allow 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.

Cal: For Unguided Cal, set external trigger. Same as above, except for Unguided Cals.

With A.09.50, this preference is no longer necessary. Use **For Guided Cal, set external trigger.**

For SCPI behavior only. [Learn more.](#)

Cal: Simulated Cal Behavior

NOT implemented for M937x models.

Cal: For Frequency Offset, use Primary Frequencies

Cal: For Frequency Offset, use Primary Frequencies Use when making mmWave measurements without a test set. [Learn more.](#)

NOT implemented for M937x models.

Cal: ECal Extrapolation for IMD

Cal: ECal Extrapolation for IMD Allows Swept IMD and IMDx channels to be calibrated beyond the stop frequency of the ECal module by extrapolating the error terms. [Learn more.](#)

Memory: Data Math 8510 Mode Standard data processing chain.

Memory: Data Math 8510 Mode Simulate the Keysight 8510 data processing chain as it pertains to Trace Math and Memory. [Learn more.](#)

NOT implemented for M937x models.

Memory: Interpolate ON is default condition Set memory interpolation to OFF as the default.

Memory: Interpolate ON is default condition Set memory interpolation to ON as the default. [Learn more.](#)

NOT implemented for M937x models.

Meas: Mathematical offset for receiver attenuation The reported test port receiver power is mathematically offset by the amount of receiver attenuation. Default for all models.

Meas: Mathematical offset for receiver attenuation The reported test port receiver power is NOT mathematically offset by the amount of receiver attenuation.

[Learn more.](#)

NOT implemented for M937x models.

Meas: Mathematical offset for source attenuation The reported reference receiver power is mathematically offset by the amount of source attenuation.

Meas: Mathematical offset for source attenuation The reported reference receiver power is NOT mathematically offset by the amount of source attenuation.. [Learn more.](#)

NOT implemented for M937x models.

Meas: RF power On during frequency sweep retrace Leave RF power ON during a retrace of single-band frequency or segment sweeps.

Meas: RF power On during frequency sweep retrace Turn RF power OFF during a retrace of single-band frequency or segment sweeps. [Learn more.](#)

NOT implemented for M937x models.

Meas: Power Sweep Retrace At the end of a power sweep, while waiting to trigger the next sweep, the PNA maintains source power at the start power level.

Meas: Power Sweep Retrace Maintain source power at the STOP power level. [Learn more.](#)

NOT implemented for M937x models.

Sets the scope of External Trigger Output signal properties. The PNA is **Preset** after changing this setting.

Meas: External Trigger OUT is Global Channels can have different External Trigger OUT settings. Default for PNA-X and N522xA models. On the Trigger Setup dialog, **Trigger Mode = Point** is ignored for external triggering.

Meas: External Trigger OUT is Global All channels have same External Trigger OUT settings. Default for PNA “C” and PNA-L models. Aux Trig OUT properties apply to all channels except the Per Point setting. To set Per Point for specific channels: On the [Trigger Setup](#) dialog, set **Trigger Scope = Channel**, under **Channel Trigger State**, select the channel, and set **Trigger Mode = Point**.

[See External Triggering dialog.](#)

NOT implemented for M937x models.

Meas: Port 1 Noise Tuner Switch set to External The Noise Figure port 1 DPDT switch is set to EXTERNAL. This setting always provides incident power through the front panel loop.

Meas: Port 1 Noise Tuner Switch set to External The Noise Figure switch is set to INTERNAL, which is one method to make accurate S-parameter measurements when an ECal module is connected as a Noise Tuner.

Note: This preference is NOT available on PNA models with a Built-in Noise Tuner.

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. [Learn more about the Noise Tuner switch.](#)

Meas: Draw failed trace segments in red Failed segments are drawn in red. [Learn more.](#)

Meas: Draw failed trace segments in red Failed data points (dots) are drawn in red.

Meas: Draw Limit Lines in Red Limit lines are drawn in the same color as the trace.

Meas: Draw Limit Lines in Red All Limit lines are drawn in Red.

Marker: Programming treats Mkr 10 as Reference A marker programming command that includes 10 as its marker number argument will operate on the Reference Marker (pre-A.10.40 behavior). [See Marker commands.](#)

Marker: Programming treats Mkr 10 as Reference A marker programming command that includes 10 as its marker number argument will NOT operate on the Reference, but a general-purpose Marker 10

Markers: On Preset, Coupled Markers is ON - Coupled Markers is OFF after Preset

Markers: On Preset, Coupled Markers is ON - Coupled Markers is ON after Preset

Markers: On Preset, Coupling Method is Channel - Marker Coupling Method is set to ALL after Preset.

Markers: On Preset, Coupling Method is Channel - Marker Coupling Method is set to Channel after Preset.

Markers: Coupling controls on/off state of markers - Turning a marker on or off will have no effect on the markers on other traces.

Markers: Coupling controls on/off state of markers - With Coupled Markers ON, when a marker is turned on, the same-numbered marker on all coupled traces will also be turned on. Likewise, turning off a marker will turn it off on all coupled traces.

NOT implemented for M937x models.

For SCPI behavior only. [Learn more.](#)

Report source unlevelled events as errors Source unlevelled events are reported as errors.

Report source unlevelled events as errors Source unlevelled events are NOT reported as errors.

NOT implemented for M937x models.

Ext Device: De-activate on PRESET and recall. External devices are de-activated when the PNA is Preset or when a Instrument State is recalled.

Ext Device: De-activate on PRESET and recall. External devices remain active when the PNA is Preset or when a Instrument State is recalled.

Learn more about External Devices.

On PRESET set two-point group delay aperture Group delay aperture is set to 11 points.

On PRESET set two-point group delay aperture Group delay aperture set to 2 points. [Learn more.](#)

On Preset turn power ON Instrument Preset always turns source power ON.

On Preset turn power ON When the current source power setting is OFF, source power remains OFF after Preset. When the current power setting is ON, source power is turned ON after Preset. [Learn more.](#)

Turn Source Power Off when receiver is overloaded. Power remains ON when a receiver is overloaded.

Turn Source Power Off when receiver is overloaded. 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 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](#).

Report when receiver is overloaded Do 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 analyzer will use when Preset. [Learn more.](#)

User Key

Sets softkey preferences. [Learn more.](#)

<p>Page Setup Standard printer settings (Paper, Orientation, and Size) do NOT survive a shutdown. All other settings DO survive a shutdown. Learn more.</p>
<p>NOT implemented for M937x models.</p> <p>Millimeter settings Sets MM Wave configurations. Learn more.</p>
<p>Power Limit Sets Power Limits and Offsets. Learn more.</p>
<p>Disp Colors Sets display items to custom colors. Learn more.</p>
<p>Print Colors Sets print items to custom colors. Learn more.</p>
<p>Defaults - Restore preferences to their default values.</p>
<p>Dialog Transparency Some dialogs can be viewed in various levels of transparency. Learn more.</p>

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

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

Dialog Transparency

Beginning with PNA release A.010.0, most PNA dialogs can be made to appear with various amounts of transparency. This allows you to view the PNA traces through the dialog as you make dialog settings.

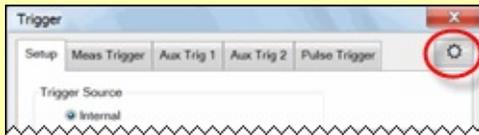
How to set Transparency Level

There are FOUR ways to make the transparency level setting:

1. Right-click in any non-control area of a dialog that allows transparency to see the following selections:



2. In tabbed dialogs, cycle through the above transparency settings by pressing  multiple times.



3. Launch the Transparency dialog (below) from the **Preferences** dialog.

4. Launch the Transparency dialog by doing the following:

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

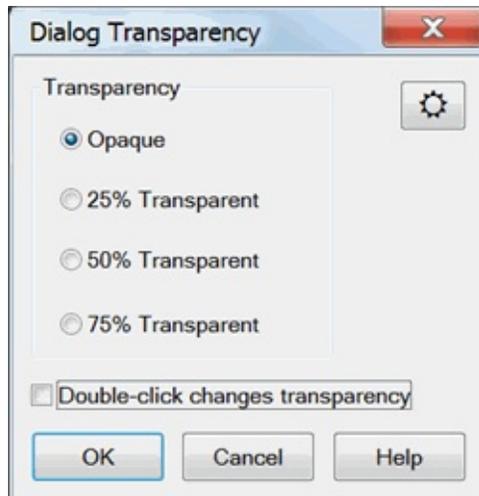
1. Press **DISPLAY**
2. then **[More]**
3. then **[Dialog Transparency]**

Using Menus

1. Click **Response**
2. then **Display**
3. then **Dialog Transparency**

This setting is not programmable

Dialog Transparency dialog box help



Note: This single Transparency setting applies to ALL supported PNA dialogs.

- Opaque (NOT Transparent) - Default setting
- 25% Transparency
- 50% Transparency
- 75% Transparency

Double-click changes transparency - When checked, cycle through the above transparency settings by double-clicking in any non-control area of a dialog that allows transparency.

Notes

- The transparency setting is stored as a **PNA Preference**.
- The setting survives a PNA Shutdown and Preset
- It is NOT saved and recalled with instrument state.

Last Modified:

25-Feb-2014 New topic (10.0)

Using Help

This topic discusses the following:

- [Documentation](#)
- [Printing Help](#)
- [Copying Help to your PC](#)
- [Launching Help](#)
- [Searching Help](#)
- [Help Languages](#)
- [Documentation Warranty](#)

See Also

[Help, About Network Analyzer](#)

Other Quick Start Topics

Help Documentation

This Help file is the **Users Guide and Programming Manual for the PXie VNA Soft Front Panel**.

Hardcopies of this help file is NOT available for purchase.

All 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.support.keysight.com/pixvna/help>

Printing Help

A printable .PDF version of this Help file is available at

<http://na.support.keysight.com/pixvna/help>

Copying Help to your PC

With the Help system on your PC, you can read about the analyzer 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 analyzer hard-drive at **C:/ Winnt/ Help/ PNAHelp.chm**. If both the analyzer 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.support.keysight.com/pna/help/index.html>.

Launching Help

The Help system can be launched in the following ways:

1. From the **Help** drop-down menu.
2. From Dialog Box Help buttons.

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.

Search for	Example	Results will show topics containing:
------------	---------	--------------------------------------

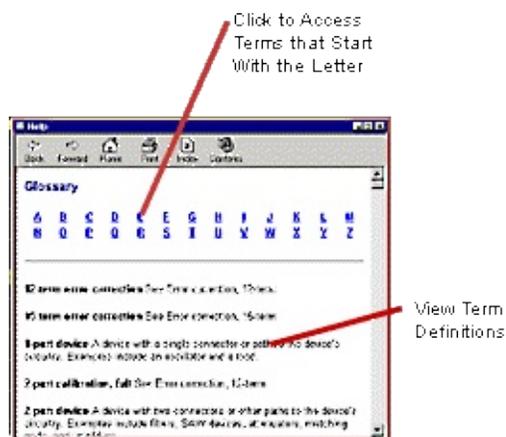
Two words in the same topic	response AND calibration	Both the words "response" and "calibration".
Either of two words in a topic	response OR calibration	Either the word "response" or the word "calibration" or both.
The first word without the second word in a topic	response NOT calibration	The word "response" but not the word "calibration".
Both words in the same topic, close together.	response NEAR calibration	The word "response" within eight words of the word "calibration".

Help Languages

This help file is offered in English ONLY.

Glossary

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



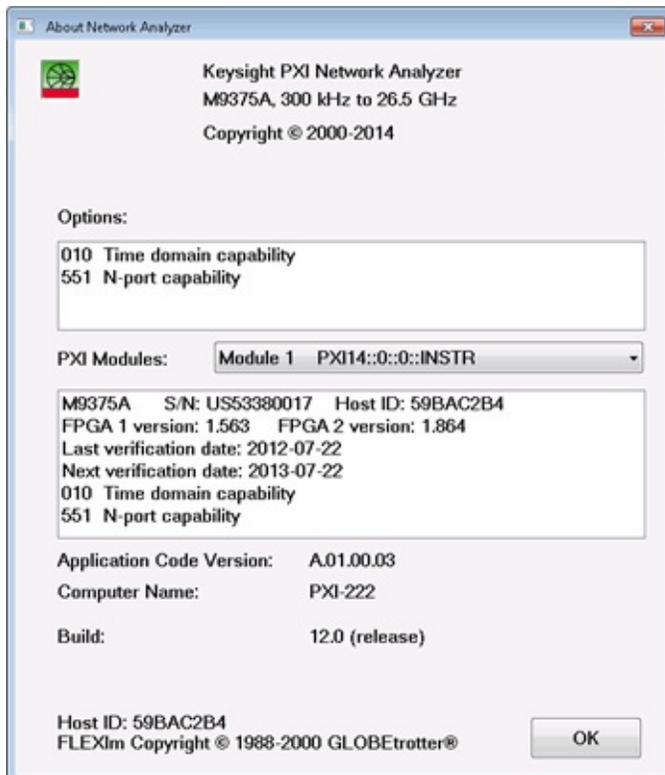
Documentation Warranty

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, KEYSIGHT 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. KEYSIGHT 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 KEYSIGHT 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.

Help - About Network Analyzer

Click **Help**, then **About Network Analyzer** to learn the capabilities of your analyzer.



- Model number ([see list of PXI models](#))
- Frequency range
- Serial number
- Options ([Learn how to install software options](#))
- Application Code (firmware) Version
- Computer Name - Full computer name of the Embedded Controller when used.

Preset the Analyzer

When you Preset the analyzer, it is set to known, or preset conditions.

- **Preset (Default) Conditions**

See other 'Setup Measurements' topics

Preset Default Conditions

How to Preset the Analyzer	
Using front-panel <small>HARDKEY</small> [softkey] buttons	Using Menus
1. Press Preset	1. Click Utility 2. then Preset
	

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

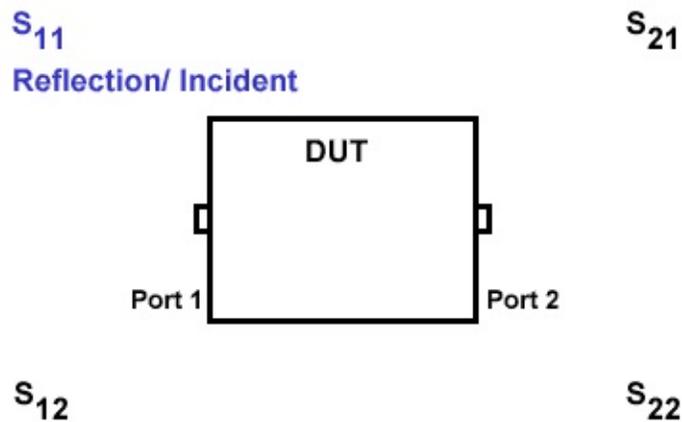
S-parameters (scattering parameters) are used to describe the way a device modifies a signal. For a 2-port device, there are **four S-Parameters**. The syntax for each parameter is described by the following:

S **out** - **in**

out = analyzer port number where the device signal output is measured
(receiver)

in = analyzer port number where the signal is applied (incident) to the device
(source)

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



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 (S_{11} and S_{22})

- Return loss
- Standing wave ratio (SWR)
- Reflection coefficient
- Impedance
- S_{11} , S_{22}

Transmission Measurements (S_{21} and S_{12})

- Insertion loss
- Transmission coefficient
- Gain/Loss
- Group delay
- Deviation from linear phase
- Electrical delay
- S_{21} , S_{12}

Receiver Measurements

All analyzer models have test port receivers and reference receivers.

For 4-port models...

- R1, R2, R3, and R4 are reference receivers. They measure the signal as it leaves the analyzer source.
 - R1 measures the signal out of Port 1
 - ...
 - R4 measures the signal out of Port 4
- A, B, C, and D 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
 - C measures the signal into PNA Port 3
 - D measures the signal into PNA Port 4

Models with more than 4 port must specify receivers using Logical Receiver Notation. [Learn more](#).

Ratioed Measurements

Ratioed measurements allow you to choose your own ratio of any two receivers that are available in your analyzer. 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.

Unratioed (Absolute Power) Measurements

The unratioed power parameter measures the absolute power going into any of the receivers that are available on your analyzer.

The reference receivers are internally configured to measure the source power for a specific analyzer port.

- **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. [Learn more.](#)
- To calibrate ratioed or unratioed receiver (power) parameters, the recommended method is the Guided Power Calibration. The **Unguided Response Calibration** can also be used to calibrate a single unratioed or ratioed parameter at a time.

How to create a NEW trace

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.](#)

Using front-panel

HARDKEY [softkey] buttons

1. Press **TRACE 1, 2, 3, OR 4**

Using Menus

1. Click **Trace/Chan**
2. then **New Trace**

Programming Commands

How to CHANGE the active trace

The only measurements that can be selected are those in the same measurement class as is currently assigned to the channel. To select a measurement other than these, first select the appropriate measurement class to a new or existing channel. [Learn how.](#)

1. Press **MEAS**
2. then **select a new parameter**

1. Click **Response**
2. then **Measure**

3. then select a new parameter

Programming Commands

New / Change Measurement dialog box help

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.](#)

S-Parameter	Balanced	Receivers	
<input type="checkbox"/> S11	<input type="checkbox"/> S12	<input type="checkbox"/> S13	<input type="checkbox"/> S14
<input type="checkbox"/> S21	<input type="checkbox"/> S22	<input type="checkbox"/> S23	<input type="checkbox"/> S24
<input type="checkbox"/> S31	<input type="checkbox"/> S32	<input type="checkbox"/> S33	<input type="checkbox"/> S34
<input type="checkbox"/> S41	<input type="checkbox"/> S42	<input type="checkbox"/> S43	<input type="checkbox"/> S44

Balanced Select a balanced measurement type.

Change Click to invoke the [Balanced DUT Topology / Logical Port mappings](#) dialog box. [Learn more about Balanced Measurements.](#)

S-Parameter	Balanced	Receivers	
<input type="checkbox"/> Sss11	<input type="checkbox"/> Ssd12	<input type="checkbox"/> Ssc12	
<input type="checkbox"/> Sds21	<input type="checkbox"/> Sdd22	<input type="checkbox"/> Sdc22	
<input type="checkbox"/> Scs21	<input type="checkbox"/> Scd22	<input type="checkbox"/> Scc22	
<input type="checkbox"/> Imbal	<input type="checkbox"/> <small>Sds21 Scs21</small>	<input type="checkbox"/> <small>Ssd12 Ssc12</small>	

Topology / Mapping / Stimulus
SE: 1 BAL: 2-3 Single End

Receivers Select receivers to make Ratioed and Unratioed (absolute power)

measurements. [Learn more about receiver measurements.](#)

S-Parameter	Balanced	Receivers	
Activate: <input checked="" type="checkbox"/>	Numerator	Denominator	Source Port
<input checked="" type="checkbox"/>	A	/ 1.0	Port 1
<input type="checkbox"/>	A	/ 1.0	Port 1
<input type="checkbox"/>	B	/ 1.0	Port 1
<input type="checkbox"/>	C	/ 1.0	Port 1
<input type="checkbox"/>	D	/ 1.0	Port 1
<input type="checkbox"/>	R	/ 1.0	Port 1
<input type="checkbox"/>	R1	/ 1.0	Port 1

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 more about Ratioed Measurements.](#)

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

- [Learn More about Unratioed Measurements.](#)

Receiver Notation

Receivers can be also selected using logical receiver notation. This "8510-style" notation makes it easy to refer to multi-port receivers.

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

For example:

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

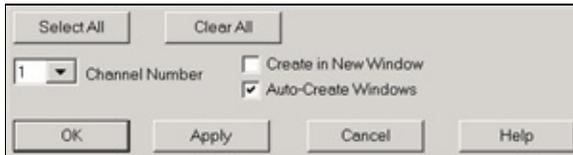
The VNA-style notation (A, B, R1 and so forth) can still be used to refer to **physical** receivers in less than 4 ports. [Learn more.](#)

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.

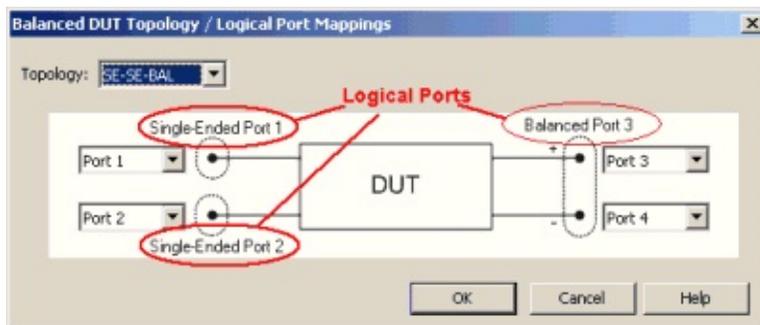
Create in New Window

- Check to create new traces in a new window.
- Clear to create new traces in the active window. When the [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 [number of windows limitation](#).

[About Measurement Parameters](#) (top of page)

Balanced DUT Topology / Logical Port mappings dialog box help



Create or edit DUT Topology and Logical Port Mapping.

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

- Any **Two** physical test ports are mapped to **One Balanced** Logical port
- Any **One** physical test 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 analyzer.

- **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.

See Also

- Learn more about [Balanced Measurements](#)
- Balanced parameters can be saved to SNP files. [Learn more.](#)

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

You can also make these settings and more from the [Sweep Type](#) dialog. See the [frequency ranges of all analyzer models](#).

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

1. Press **FREQUENCY**

Using Menus

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

Frequency Start/Stop - Center/Span - Step dialog box help

[Programming Commands](#)

Field	Value
Start	10.000000 MHz
Center	4.505000000 GHz
Stop	9.000000000 GHz
Span	8.990000000 GHz
Points	201
Step	44.950000 MHz

Either of the following pairs of settings determine the frequency range. The last value that you enter determines the X-Axis labels. For example, if you enter the Start and Span values, the X-Axis will show Center and Span labels.

Start /Stop - Specifies the beginning and end frequency of the swept measurement

range.

Center /Span - Specifies the value at the center and frequency range.

Either of the following settings determine the number of evenly-spaced data points across the frequency range.

Points - Specifies the number of evenly-spaced data points across the frequency range. [Learn more about Data Points](#).

Step - Available ONLY in **Linear sweep type**. Specifies the frequency step size between evenly-spaced data points. Changes to this setting will cause the Points setting to adjust to the closest integer. Any 'remainder' will adjust either the Stop value or Span value depending on which is displayed on the X-Axis label.

Zoom

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**

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 the **Sweep Type** dialog box.

Using front-panel HARDKEY [softkey] buttons

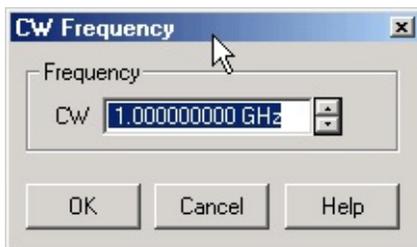
2. Press **FREQ**
3. then **[CW]**

Using Menus

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

◀ Programming Commands ▶

CW Frequency dialog box help



CW Type a value and the first letter of the suffix (k,m,or g) or use the up and down arrows to select any value within the range of the PNA.

Frequency Resolution

The resolution for setting frequency is 1 Hz.

Frequency Band Crossings

Source power is NOT turned OFF during M937xA frequency band crossings. Therefore, there is no need to know the band crossing for the M937x analyzers. See also **Power ON and OFF during Sweep and Retrace**.

Power Level

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

- [How to make Power Settings](#)
- [Power Dialog](#)
- [Power and Attenuator Dialog](#)
- [Power ON and OFF during Save / Recall 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 specifications](#) for your analyzer.

How to make Power settings

Use one of the following methods to set port power.

Using front-panel HARDKEY [softkey] buttons

1. Press **POWER**
2. then [**Power**] or [**Power and Attenuators**]

Using Menus

1. Click **Stimulus**
2. then **Power** or **Power and Attenuators**

[Programming Commands](#)

Power dialog box help



This dialog provides basic control of source power for a specific port.

See [Power and Attenuators dialog box](#).

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

Port Power Sets the power level for the specified port.

Power Sweep

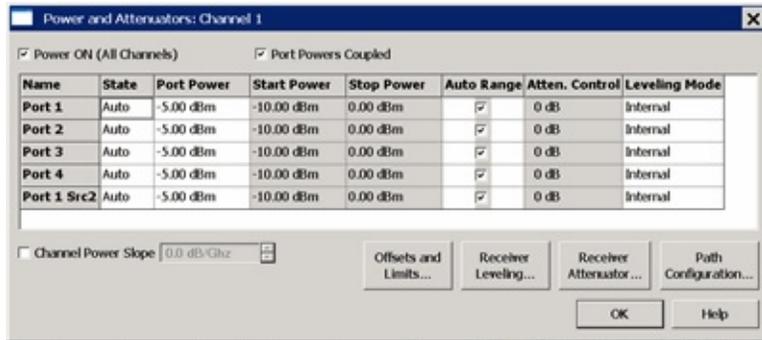
Start / Stop Power Set the start and stop power values of a power sweep.

- These settings are only available when **Sweep Type** is set to Power Sweep.
- You can **Zoom** to easily change the start and stop power levels in a power sweep. [Learn how](#).

Power Slope

Not available on M937x models.

Power and Attenuators dialog box help



Defines and controls the source power and attenuation for the active channel.

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.

Name Lists the analyzer 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.

- 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 how to set Power Sweep.](#)

- 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.](#)
- A power sweep can be performed with **uncoupled power**. Different power ranges can be swept in the forward and reverse directions.

Leveling Mode

- **Internal** - Source correction factors are used to provide a flat output power within specifications. NO internal circuitry is used to level the output power. Therefore, the source will NEVER become unlevelled.
- **Receiver Rx** - Receiver Leveling. Select a receiver to use for leveling the source. [Learn more.](#)

Channel Power Slope

Not available on M937x models.

Offset and Limits Launches the [Power Offset and Limits](#) dialog.

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.

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

Source power is NOT turned OFF during M937xA frequency band crossings or

during sweep retrace.

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

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

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

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

Using Menus

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

Sweep Type dialog box help

◀ Programming Commands ▶



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 analyzer measures during a sweep. Range: 2 to 20001.(Default is 201).
- **Power** - Sets the power level or the source. [Learn more.](#)
- **IF Bandwidth** - [Learn more.](#)

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 analyzer 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 analyzer remains during the measurement sweep.

CW Time Sets the analyzer to a single frequency, and the data is displayed versus time. [Learn more.](#)

- **CW Frequency** Sets the frequency where the analyzer 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 analyzer measures during a sweep. Range: 2 to 20001.(Default is 201).

Segment Sweep Sets the analyzer 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.

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 [analyzer ALC range](#).

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.
- Power Sweep is 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.

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 the **max number of data points per trace**.

How to make segment sweep settings	
Using front-panel HARDKEY [softkey] buttons	Using Menu
<ol style="list-style-type: none"> 1. Press SWEEP 2. then [Sweep Type] 	<ol style="list-style-type: none"> 1. Click Stimulus 2. then Sweep 3. then Sweep Type 4. then Segment
	

The Segment Table will be shown automatically when the **Segment Table** softkey menu is displayed. The Table will be hidden automatically after exiting the Segment Table menu.

To display the Segment Table while using a menu other than the **Segment Table** menu, use the 4th softkey on the menu to control when the table is shown. The softkey offers three behaviors based on the softkey's changing label.

- **Show Segment Table:** causes the Segment Table to appear but it will automatically hide when exiting the menu.
- **Hide Segment Table:** hides the Segment Table to appear but it will automatically reappear if the menu is exited then re-entered.
- **Pin Segment Table:** causes the Segment Table to appear and it will remain displayed after exiting the Segment Table menu.

To make the following menu settings available, you must first show the segment table. From the menus, click **Stimulus**, then **Sweep**, then **Segment Table**, then **Show Table**.

Then choose from the following:

Add Segment - adds a sweep segment at the end of the segments.

Insert Segment - adds a sweep segment before the selected segment.

Tip: 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

	STATE	START	STOP	POI	IFBW	P1 Pwr	P2 Pwr	TIME
1	ON	20.000000 MHz	1.000000 GHz	21	10.0 kHz	17.00 dBm	0.00 dBm	2.474 msec
2	ON	1.000000 GHz	4.000000 GHz	21	35.0 kHz	17.00 dBm	0.00 dBm	630.000 µsec

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

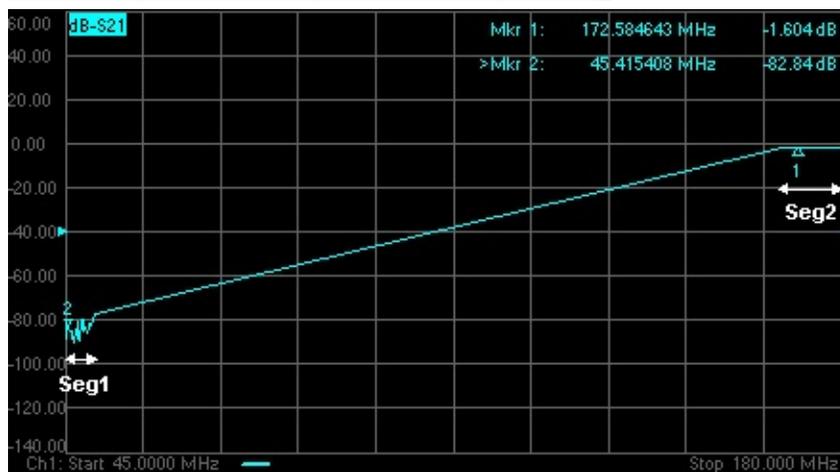
On the **Sweep Type** dialog box, click **Segment Sweep**

Then check **X-Axis Point Spacing**

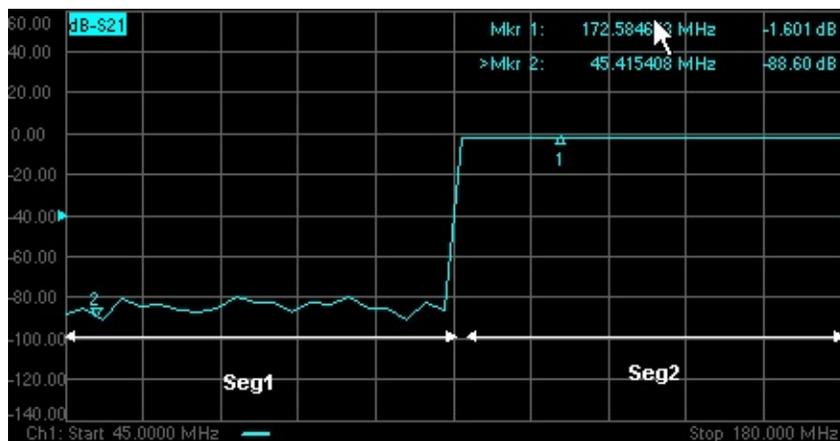
- **Without X-axis point spacing**, a multi-segment sweep trace can sometimes result in squeezing many measurement points into a narrow portion of the x-axis.
- **With X-axis point spacing**, the x-axis position of each point is chosen so that all measurement points are evenly spaced along the x-axis.

For example, given the following two segments:

	STATE	START	STOP	POINTS
1	ON	45.000000 MHz	50.000000 MHz	21
2	ON	170.000000 MHz	180.000000 MHz	21



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 analyzer automatically maintains the fastest sweep time possible with the selected measurement settings. However, you can increase the sweep time to perform a slower sweep.

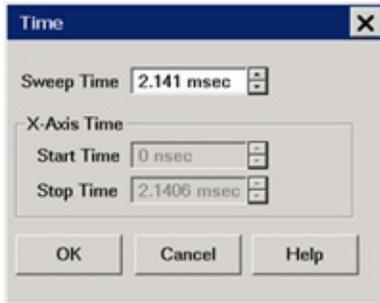
How to set Sweep Time

Using front-panel HARDKEY [softkey] buttons

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

Using Menus

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



Sweep Time Specifies the time the analyzer takes to acquire data for a sweep. The maximum sweep time of the analyzer is 86400 seconds or 1 day. [Learn about other settings that affect sweep speed.](#)

X-Axis Time Set Start and Stop time to be displayed on the X-axis. These settings are NEVER available.

Note: Ignore CW time as labeled on the X-axis and marker readout. It is NOT at all accurate.

Sweep Setup

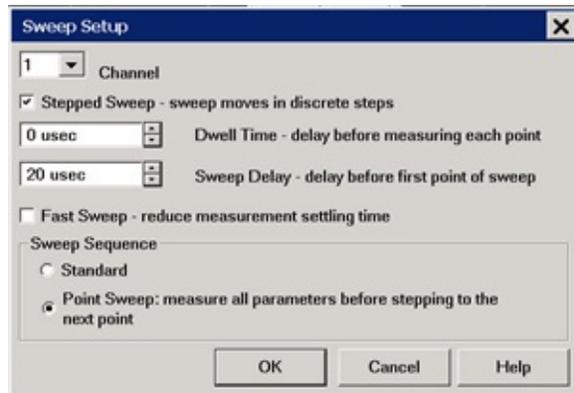
How to make Sweep Setup settings

Using front-panel HARDKEY [softkey] buttons

1. Press **STIMULUS**
2. then **[Sweep]**
3. then **[Sweep Setup]**

Using Menus

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



Channel Specifies the channel that the settings apply to.

Stepped Sweep When checked (Stepped Sweep) the analyzer 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 analyzer 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 check box is cleared, the analyzer could be in either Analog or Step mode. The mode can change from sweep to sweep. There is **NO way** to determine whether the analyzer 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.
- When step mode is a faster way to take the data.

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 Available ONLY with Opt. 103 When checked, in Analog Sweep mode the analyzer 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 NOT allowed with **Power Limit** enabled.
- **Note:** Performance specifications do NOT apply in Fast Sweep.

Sweep Sequence

Standard Sweep When checked, the analyzer sweeps all data points for each source port in turn. For a 2-port analyzer, 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.

Point Sweep Available ONLY on standard S-parameter channels. When checked, the analyzer 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 usually results in slower sweeps and is useful only in rare circumstances.
- Point sweep is the same as stepped sweep mode on the 8510 and 8530.

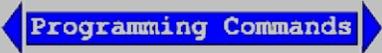
Trigger

A trigger is a signal that causes the analyzer to make a measurement sweep. The analyzer offers great flexibility in configuring the trigger function.

View the interactive [Trigger Model](#) animation to see how triggering works.

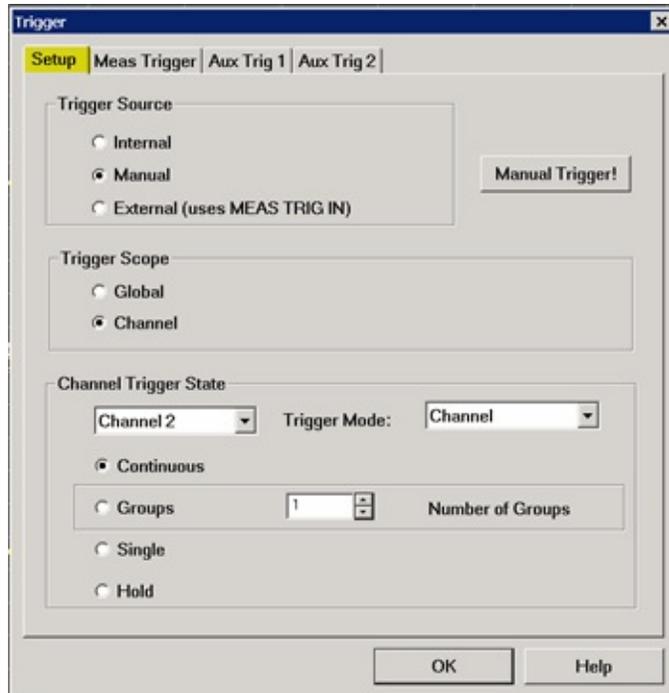
- [How to Set Trigger](#)
- [Source](#)
- [Scope](#)
- [Channel Settings](#)
- [Restart](#)
- [External Triggering](#) (separate topic)

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

How to set Triggering	
Using front-panel <small>HARDKEY</small> [<small>softkey</small>] buttons	Using Menus
<ol style="list-style-type: none">1. Press TRIGGER2. then [Trigger...]	<ol style="list-style-type: none">1. Click Stimulus2. then Trigger3. then Trigger
	

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.

Trigger Source

These settings determine **where** the trigger signals originate for all existing channels. A valid trigger signal can be generated only when the analyzer is not sweeping.

Internal Continuous trigger signals are sent by the analyzer 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 analyzer. 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 Triggering.](#)
- Interactive [Trigger Model](#) animation

External Triggering

External triggering is used to synchronize the triggering of the analyzer with other equipment.

- [Overview](#)
- How to make Trigger Settings:
 - [Meas Trig \(IN\) Dialog](#)

See Also

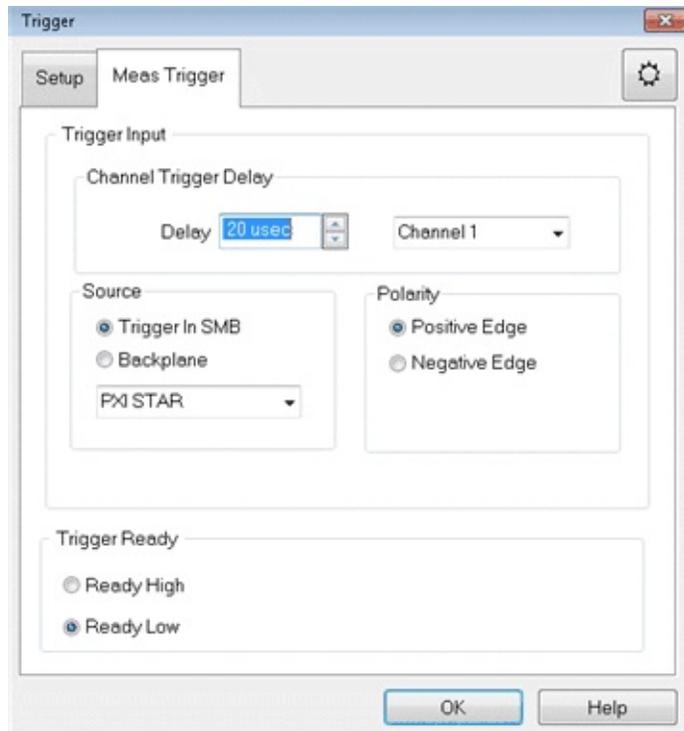
- [Controlling a Handler](#)
 - [Synchronizing an External Source](#)
 - [Internal Triggering](#)
-

Overview

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, or has been, made. Usually the slower instrument sends the trigger signal.

Meas (External) Trigger dialog box help



See how to access the Trigger Dialog

Trigger Ready and Trigger IN

The Trigger connectors are located on the front-panel.

These signals can be used when the VNA is communicating with a slow mechanical device. A material handler is very mechanical and takes a relatively long time to load and discharge parts. Here is how these signals work together to communicate:

1. The VNA sends a 'Ready' signal when it is ready to make a measurement.
2. The external device sends a trigger signal to the VNA when it is ready for a measurement.

Dialog Settings

To cause the VNA to respond to Meas Trig IN or Handler I/O signals, select **External** on the **Trigger Setup** tab, **Source** setting.

Also on the Trigger **Setup** tab, Scope setting, choose whether one external trigger signal will apply to ALL channels (Global) or one trigger signal per Channel. The following settings apply accordingly.

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 VNA accepts Trigger IN signals through the following connectors:

- Trig In SMB (on front panel of module)
- Backplane (on front panel of embedded controller to use backplane trigger lines)

Level / Edge

Positive Edge After the VNA arms, it will trigger on the next positive edge.

Negative Edge After the VNA arms, it will trigger on the next negative edge.

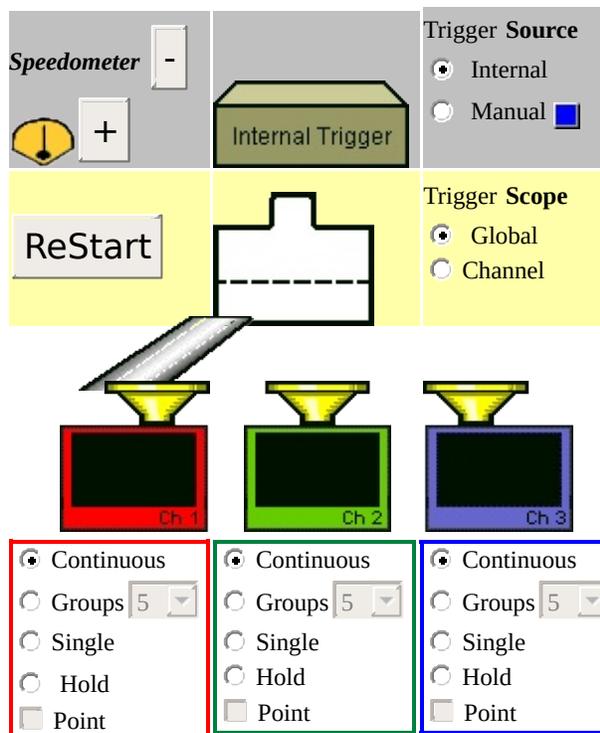
Trigger Ready

Choose Polarity of the 'Ready OUT' signal.

- **Ready High** - TTL High indicates the VNA is ready for trigger.
- **Ready Low** - TTL Low indicates the VNA is ready for trigger (default setting).

See Also

- [Learn how to External Trigger during Calibration](#)



About the trigger model

Read [Text description](#) of triggering behaviors.

This model does not include [Sweep trigger mode](#).

Data Format

A data format is the way the analyzer 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**

Using Menus

1. Click **Response**
2. then **Format**

[Programming Commands](#)

Format dialog box help



Click a link to learn about that format:

Log Mag	Polar	Kelvin
Phase / Unwrapped Phase	Linear Mag	°F
Group Delay	SWR	°C
Smith / Inverse Smith Chart	Real	
	Imaginary	

Format Unit

Only the following Formats allow a Unit selections:

Log Mag - Choose from:

- **dBm** (Power)
- **dBmV** (dB milli Volts) - used for unratiod receiver measurements.
- **dBmA** (dB milli Amps) - used for unratiod receiver measurements.

Lin Mag - Choose from:

- W (Watts), V, (volts), A (amps)

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, then 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 analyzer 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 $(1+\rho)/(1-\rho)$ where ρ 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:
 - time domain
 - auxiliary input voltage signal for service purposes

Imaginary Format

- Displays only the imaginary (reactive) portion of the measured data.
- Y - axis: Unitless
- Typical Measurements:

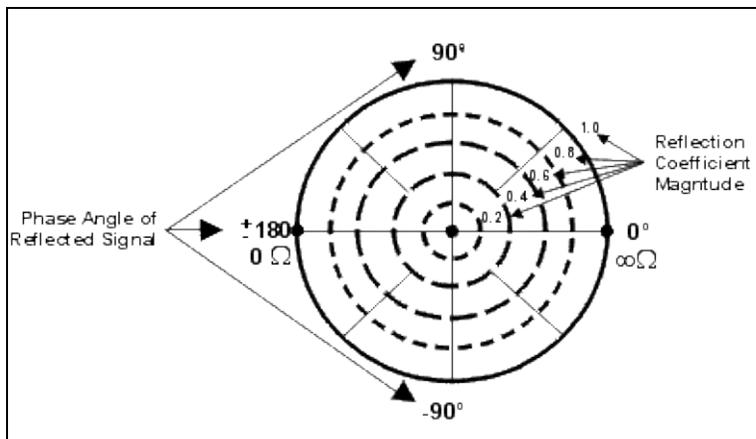
- impedance for designing matching network

Polar Format

Polar format is used to view the magnitude and phase of the reflection coefficient (Γ) from your S_{11} or S_{22} 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 (Γ) of 1, or total reflected signal. The center of the circle represents a reflection coefficient (Γ) 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° , $\pm 180^\circ$, and -90° correspond to the top, left-most, and bottom positions on the polar display, respectively.

Smith Chart Format

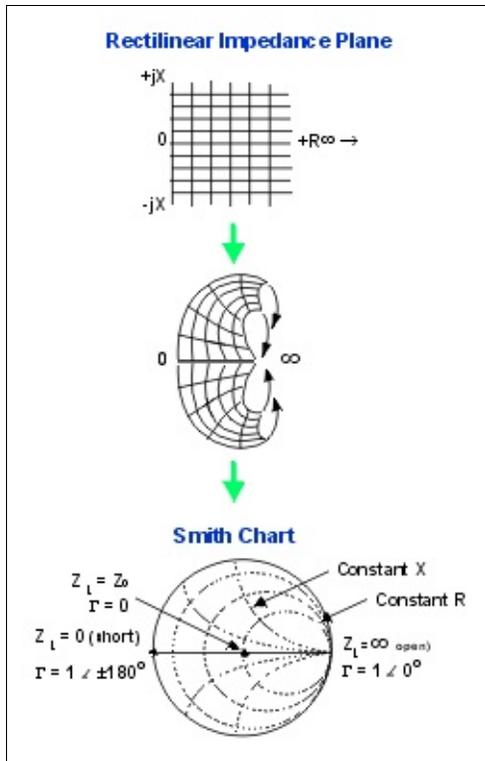
The Smith chart is a tool that maps the complex reflection coefficient (Γ) to the test device's impedance.

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)

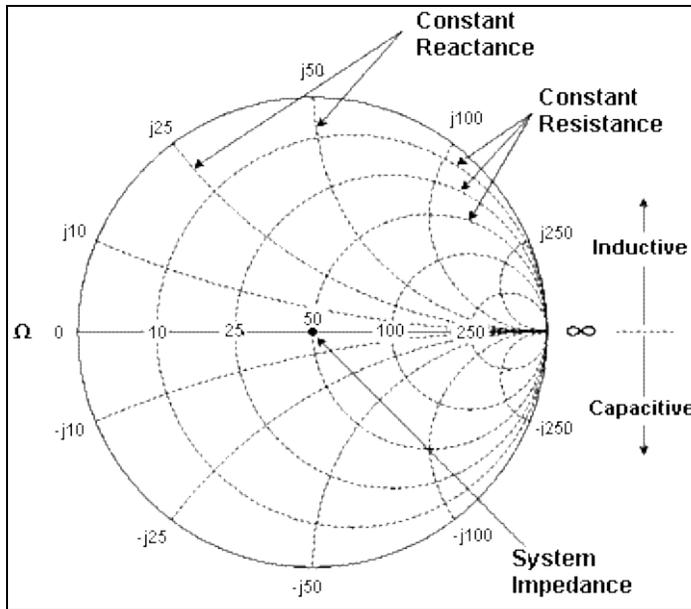


Inverse Smith Chart (also known as Admittance)

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.

Kelvin, °F, and °C

Used to display temperature, primarily with the Noise Figure application. Learn more.

Scale

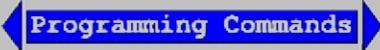
The Scale, Reference Level and Reference Position settings (along with **Format**) determine how the data trace appears on the PNA screen.

- [Scale, Reference Level and Position](#)
- [Scale Coupling](#)
- [Magnitude Offset](#)

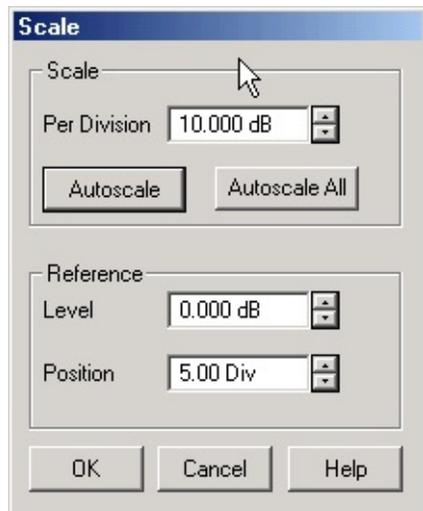
[See other 'Setup Measurements' topics](#)

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	
Using front-panel <code>HARDKEY</code> [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press <code>SCALE</code>	<ol style="list-style-type: none">1. Click Response2. then Scale3. then Scale
	

Scale dialog box help



Note: 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.

Tip: Click on the Y-axis labels, then use a mouse scroll wheel to change scale in preset increments. Or Right-click on **Y-axis** annotation to change Scale.

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.

Tip: Double click on the Y-axis labels to autoscale the active trace.

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  on the screen. Range: -500 dB to 500 dB.

In Polar and Smith chart formats, reference level is not applicable.

Tip: Click on the Y-axis labels, then drag up or down to change the reference level in preset increments.

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.

Tip: Click on the triangle , then drag up or down to change the reference position in preset increments.

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**

Using Menus

1. Click **Response**
2. then **Scale**
3. then **Scale Coupling**

 Programming Commands 

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]**

Using Menus

1. Click **Response**
2. then **Scale**
3. then **Magnitude Offset**

Programming Commands

Magnitude Offset dialog box help



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.

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.

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](#).

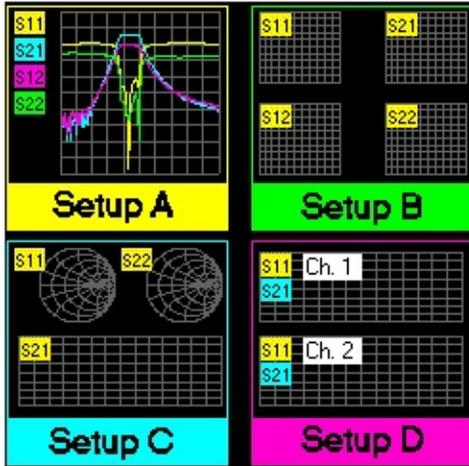
[See other 'Setup Measurements' topics](#)

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	
Using front-panel <code>HARDKEY</code> [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press <code>DISPLAY</code>2. then <code>[Measurement Setups]</code>	<ol style="list-style-type: none">1. Click Response2. then Display3. then Meas Setups
No programming commands exist for this feature	

The following are the four pre-configured measurement setups:



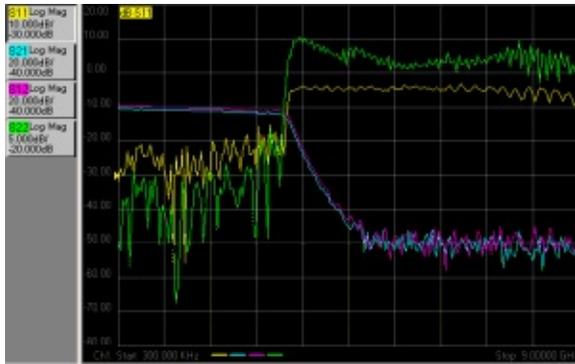
Arranging Existing Measurements

The following arrangements place EXISTING measurements into pre-configured Window arrangements using a sort algorithm.

How to select an Existing measurement arrangement	
<p>Using front-panel HARDKEY [softkey] buttons</p> <ol style="list-style-type: none"> 1. Press DISPLAY 2. then [Layout] 3. then choose from the following... 	<p>Using Menus</p> <ol style="list-style-type: none"> 1. Click Response 2. then Display 3. then choose from the following...
<p>◀ Programming Commands ▶</p>	

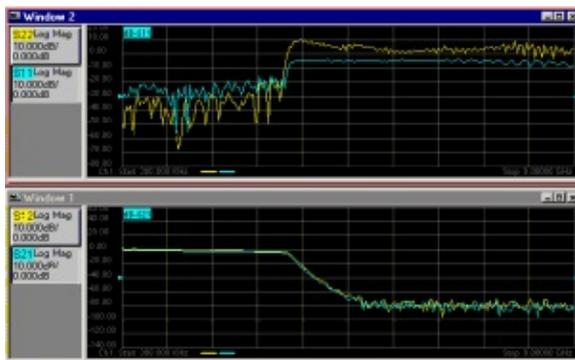
Overlay 1x

This configuration places all existing traces in a single window, all overlaid on each other.



Stack 2x

This configuration places all existing traces in two "stacked" windows.



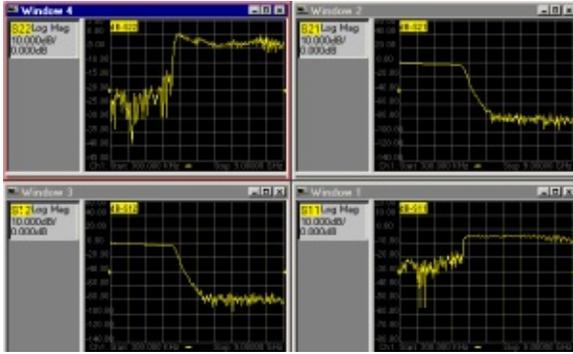
Split 3x

This configuration places all existing traces in three windows, two on top and one below.



Quad 4x

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 **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.

Customize the Analyzer Screen

You can customize your analyzer screen by showing or hiding the following display elements. All of these selections are made from the **Response > Display** menu.

- **Layout** (Separate topic)
- **Windows** (Separate topic)
- **Measurement Setups** (Separate topic)
- **Display Labels**
- **Marker Display** (Separate topic)
- **Tables**
- **Toolbars**
- **Status and Title Bars**
- **Display Colors** (Separate topic)
- **Grid: SOLID | Dotted**
- **System Date and Time**
- **Minimize Application**

See Also

Traces, Channels, and Windows

See other 'Setup Measurements' topics

How to show/hide the Status Bar and Title Bars

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

1. Press **DISPLAY**
2. then [**Tables and Toolbars**]
3. then [**Status Bar or Title Bars**]

Using Menus

1. Click **Response**
2. then **Display**
3. then **Toolbars**
4. then **Status Bar or Title Bars**

Status Toolbar

Status CH 1: S11 No Cor

When enabled, the status bar is displayed along the bottom of the screen. The primary status bar shows the following:

TIP: Right-click on many of these items in the status bar for quick access to settings.

- **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)
- 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)
- System Date and Time - Can be set ON or OFF.

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 analyzer is Preset.

Title Bars

The Title bar shows the window number and Minimize / Maximize icons.

- Checked - Title bars for all windows are shown.
- Cleared - Title bars for all windows are hidden. This allows more room to display measurement results.

Toolbars

You can display different toolbars to allow you to easily set up and modify measurements.

Your toolbar choices are reset to default settings only on analyzer startup (NOT on Preset).

How to display Toolbars	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press DISPLAY 2. then [Tables and Toolbars] 3. then [Toolbars] 	<ol style="list-style-type: none"> 1. Click Response 2. then Display 3. then Toolbars
	

List of toolbars

- **Active Entry**
- **Softkeys**
- **Markers**
- **Time Domain**
- **Port Extension**
- **All Off** (does NOT appear on softkeys)

Note: There is also a Cal Set toolbar available for **Monitoring Error Terms**

Active Entry Toolbar

Marker 1 10.0000000000 GHz

When used with softkeys, this area allows numeric values to be entered for settings. From the keyboard, enter G for Giga, M for Mega or milli, K for kilo, and so forth.

Softkeys

Softkeys are automatically turned ON when one of the 'function' hardkeys is pressed. This setting allows you to turn the softkeys OFF to show more measurement space on the screen. The softkeys will reappear when another function hardkey is pressed.

Markers Toolbar

Marker 1 Delta Start/Stop Center/Span

The markers toolbar allows you to set up and modify markers. It shows:

- Marker number
- Stimulation value
- Marker functions:
 - Delta
 - Start/Stop
 - 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](#)

Time Domain toolbar

Transform Gating Transform Start/Stop -10.000000 nsec 10.000000 nsec More... ? X

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 toolbar



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. NOT available on softkeys.

Tables

Tables are displayed at the bottom of the selected window.

How to display tables

Each window can display only one table at a time.

Click **Hide Table** to turn OFF the table.

Using front-panel

HARDKEY [softkey] buttons

1. Press **DISPLAY**
2. then **[Tables and Toolbars]**
3. then **[Tables]**

Using Menus

1. Click **Response**
2. then **Display**
3. then **Tables**

Programming Commands

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 Labels

How to show and hide Display items	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press DISPLAY2. then [Labels]	<ol style="list-style-type: none">1. Click Response2. then Display3. then Labels
	

List of Display Labels

- Window Title
- Trace Title
- Trace Status
- Frequency, Stimulus

Window Title

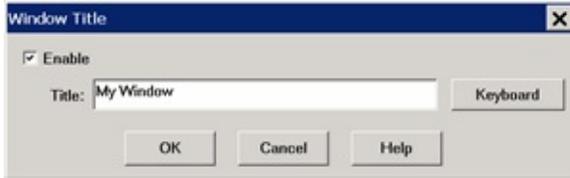
You can create and display a title for each **window**.

- The limit is set by the number of windows that are displayed.
- The title (My Window) is annotated in the upper-left of the window as follows:



How to enter a Window Title

1. Click in the window to make the window active.
2. Click **Response**, then **Display**, then **Labels**, then **Window 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.

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 New Trace LogM 10.00dB/ 0.00dB

How to enter a Trace Title

1. Click the **Trace Status** label to select a trace.
2. Click **Response**, then **Display**, then **Labels**, 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.

Trace Status



Tr 1 S12 LogM 10.00dB/ 0.00dB
Tr 3 S12 LogM 10.00dB/ 0.00dB

Trace status is annotated at the top of each window.
The highlighted trace number indicates the Active Trace.
Click the title 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** (see below).
- 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

How to set Grid and Clock display settings

Using front-panel

Using Menus

HARDKEY [softkey] buttons	
<ol style="list-style-type: none">1. Press DISPLAY2. then [More]	<ol style="list-style-type: none">1. Click Response2. then Display3. then 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 analyzer is Preset.

Set the color of the grid using [Display Colors](#).

[How to display grid settings](#)

System Date and Time (also known as Clock)

The system date and time can be shown in the far right corner of the status bar. The format is: year-mo-day hr:min and can NOT be changed.

[Learn how to set the time](#)

[How to show/hide the clock](#)

Minimize Application

The Network Analyzer application can be minimized to show the desktop and Windows taskbar.

1. Click **File**
2. then **Minimize Application**

To restore the analyzer application, click the analyzer application on the Windows taskbar.

Last modified:

8-Nov-2013 Added clock

21-Jun-2013	Removed 3 toolbars (10.0)
3-May-2012	Removed C models
21-Jul-2011	Several changes (9.5)
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 with the same stimulus conditions as an existing channel.

- [Why Copy Channels](#)
- [How to Copy Channels](#)
- [List of Channel Settings](#)

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]**

Using Menus

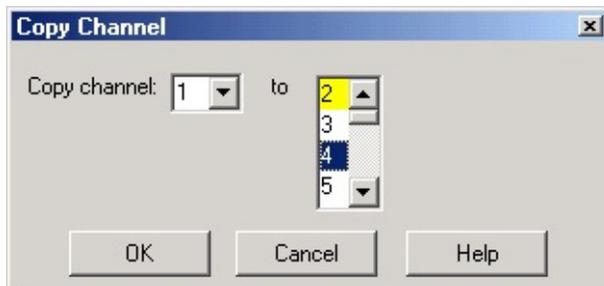
1. Click **Trace/Chan**
2. then **Channel**

3. then [More]
4. then [Copy Channel]

3. then Copy Channel

Programming Commands

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 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](#)

List of Channel Settings

- [Frequency Span](#)
 - [Power](#)
 - [Cal Set usage](#)
 - [IF Bandwidth](#)
 - [Number of Points](#)
 - [Sweep Settings](#)
 - [Average](#)
 - [Trigger \(some settings\)](#)
-

Undo/Redo Settings

If you make an incorrect setting, you can quickly recover by selecting Undo. If you then incorrectly Undo a setting, you can Redo the undone setting.

- Undo and Redo applies ONLY to **selected settings**.
- The Undo stack remembers 16 levels of Undo-able settings.

How to Undo or Redo a setting

Tips:

- Click or touch the Undo and Redo Icons:



Undo Redo

- Undo/Redo can be stored to a User Key or Favorite softkey. [Learn more](#).
- With a mouse, right-click on the Softkeys or on the Entry toolbar.
- With a keyboard:
 - Undo....Ctrl+Z
 - Redo....Ctrl+Y

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

1. Press **SYSTEM**
2. then **[Undo]**
3. then **[Undo] <setting>**
or **[Redo] <setting>**

Using Menus

1. Click **Utility**
2. then **Undo**
3. then **Undo<setting>**
or **Redo <setting>**

SCPI and COM programming and Undo/Redo:

- There are NO Programming commands to invoke Undo/Redo
- Programing commands are NOT Undo-able.

- The Undo stack is cleared when programming commands are sent to the PNA.

Clear Undo History

To clear the Undo stack, press **System**, then **Service**, then **Utilities**, then **Click Undo History**.

Undo and Security

- Undo/Redo is disabled with **High** and **Extra** security levels. [Learn more](#).
- State files that are saved for Undo/Redo purposes (for example: Preset) are deleted when any of the following occur:
 - The Security level is changed
 - The Network Analyzer App is started or closed.

Selected Undo-able settings

You can Undo or Redo the following **settings**:

Note: There are several settings that are NOT Undo-able. Because of this, when you attempt to Undo a long sequence of operations, it is unlikely that the original state can be recreated exactly.

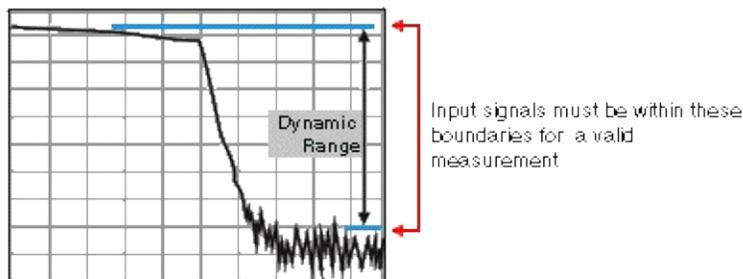
- **Preset**
- **File Recall**
- Frequency Settings
- **Turn off Marker** and **Marker All OFF**
- **Number of Points**
- **Power Level** - most applications and S-parameters
- **Turn OFF Channel**
- **Close Window**
- **New Channel , new Window, and new Trace.**
- Delete Trace
- Window Tile
- Change Layout (1x, 2x, 3x, 4x)
- Move Trace, Drag Trace

- Zoom XY, Zoom Out Full
 - Autoscale All, Autoscale
 - Scale, Reference Level, Reference Position
 - Scale Coupling dialog
 - Electrical Delay
 - Phase Offset
 - Measurement Setups dialog
 - Format
 - Sweep Type
 - Data->Memory
 - Single Marker Searches (Max, Min, Target, Peak...)
 - Multi-marker Searches (Bandwidth, Power Saturation, Normal Operating Pt)
 - Change a Marker's stimulus value: softkeys, dialog or drag
 - Change cell in Segment Table
 - Mechanical Settings dialog
-

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

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.
-

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 analyzer measures across a sweep. (A "sweep" is a series of consecutive data point measurements, taken over a sequence of stimulus values.)

The analyzer 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.

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

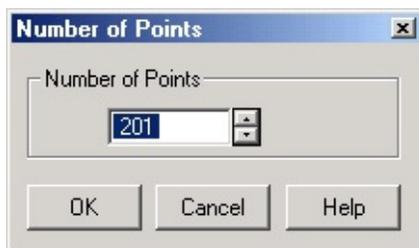
1. Press **SWEEP**
2. then **[Number of Points]**

Using Menus

1. Click **Stimulus**
2. then **Sweep**
3. then **Number of Points**

[Programming Commands](#)

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 **100,001**. The default value is 201.

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.
-
-

Phase Measurement Accuracy

You can increase the accuracy of phase measurements by using the following features.

- [Electrical Delay](#)
- [Phase Offset](#)
- [Spacing Between Frequency Points \(Aliasing\)](#)

See Also

[Port Extensions](#)

[Comparing the Delay Functions](#)

[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

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

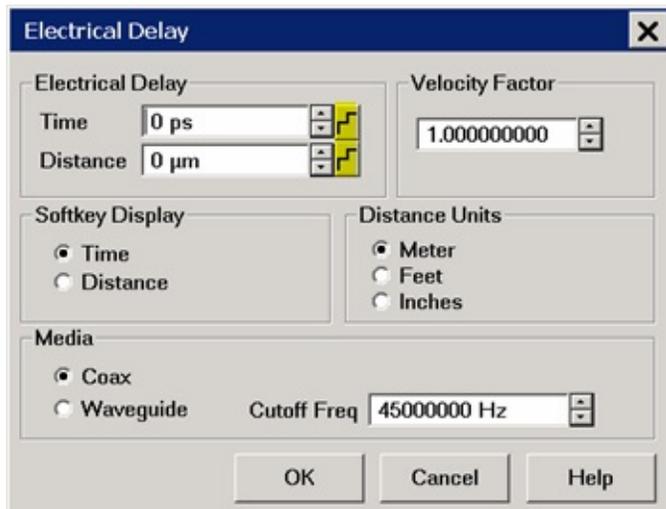
1. Press **SCALE**
2. then **[Electrical Delay]**

Using Menus

1. Click **Response**
2. then **Scale**
3. then **Electrical Delay**

[Programming Commands](#)

Electrical Delay dialog box help



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.

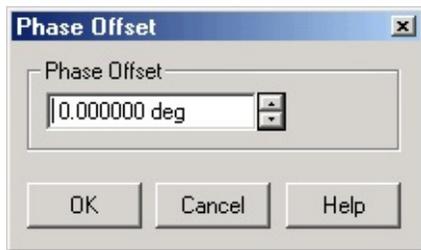
Phase Offset

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	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press SCALE 2. then [Phase Offset] 	<ol style="list-style-type: none"> 1. Click Response 2. then Scale 3. then Phase Offset

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)

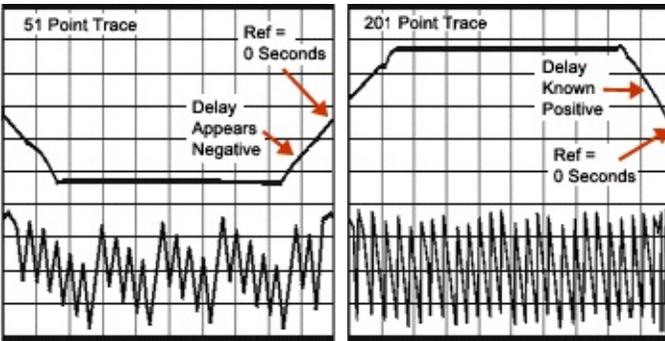
Spacing Between Frequency Points (Aliasing)

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 $>180^\circ$ 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**.

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.

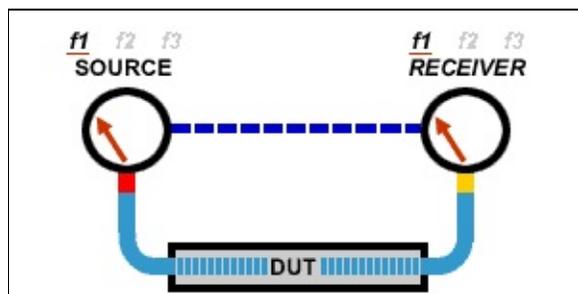
- why Device Delay May Create Inaccurate Results
- Solutions to Increase Measurement Accuracy
 - Slow the Sweep Speed
 - Add Electrical Length to the R Channel

Other topics about Optimizing Measurements

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 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 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

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](#)

Other topics about Optimizing Measurements

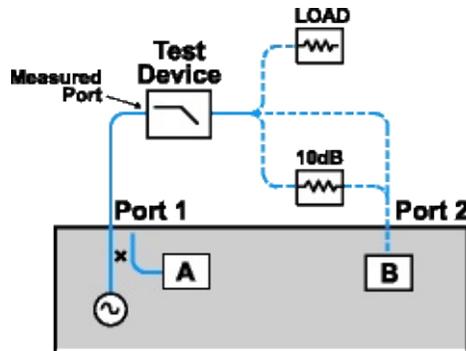
Why Terminate the Unmeasured Port

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

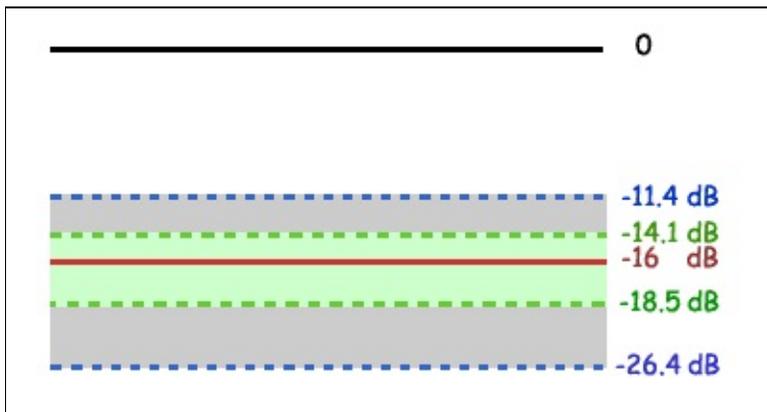
Use one of the following methods:

- 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:

$$= 18 \text{ dB } (.126)$$

Load match (NA_{LM})

$$\text{Directivity } (NA_D) = 40 \text{ dB } (.010)$$

Filter:

$$\text{Insertion loss } (FIL) = 1 \text{ dB } (.891)$$

$$\text{Return loss } (FRL) = 16 \text{ dB } (.158)$$

Attenuator:

$$\text{Insertion loss } (A_{IL}) = 10 \text{ dB } (.316)$$

$$\text{SWR } (A_{SWR}) = 1.05 (.024)$$

32.26 dB Return Loss

Calculations:

	Without Attenuator	With Attenuator
ρ_{NA}	$= (FIL) * (NA_{LM}) * (FIL)$ $= (.891) * (.126) * (.891)$ $= .100$	$= (FIL) * (A_{IL}) * (NA_{LM}) * (A_{IL}) * (FIL)$ $= (.891) * (.316) * (.126) * (.316) * (.891)$ $= .010$
$\rho_{Attenuator}$	NA	$= (FIL) * (A_{SWR}) * (FIL)$ $= (.891) * (.024) * (.891)$ $= .019$
Worst Case Error (E_{WC})	$= \rho_{NA}$ $= .1$	$= \rho_{NA} + \rho_{Attn.}$ $= .01 + .019$ $= .029$

Uncertainty Adds	$= -20\log(FRL) + (E_{WC}) + (NA_D)$ $= -20\log(.158) + (.100) + (.010)$ $= \mathbf{11.4 \text{ dB}}$	$= -20\log(FRL) + (E_{WC}) + (NA_D)$ $= -20\log(.158) + (.029) + (.010)$ $= \mathbf{14.1 \text{ dB}}$
-------------------------	---	---

Uncertainty Subtracts	$= -20\log(FRL) - (E_{WC}) - (NA_D)$ $= -20\log(.158) - (.100) - (.010)$ $= \mathbf{26.4 \text{ dB}}$	$= -20\log(FRL) - (E_{WC}) - (NA_D)$ $= -20\log(.158) - (.029) - (.010)$ $= \mathbf{18.5 \text{ dB}}$
------------------------------	---	---

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 features help reduce trace noise and the noise floor which can lead to better dynamic range and more accurate measurements.

- [Averaging](#)
- [IF Bandwidth](#)
- [Trace Smoothing](#)

See Also

[Group Delay](#)

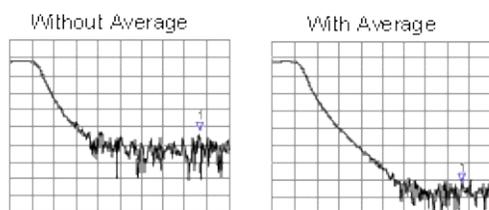
[Increase Dynamic Range](#)

Other topics about Optimizing Measurements

Averaging

Averaging is a feature that reduces the effects of random noise on a measurement. The analyzer 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



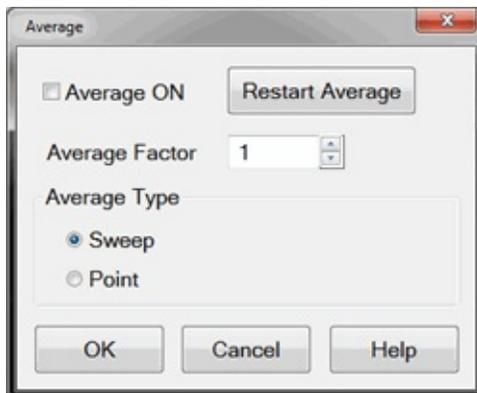
Both **Averaging** and **IF Bandwidth** can be used for the same benefit of general noise reduction. For minimizing very low noise, Averaging is more effective than reducing IF bandwidth. Generally, Averaging takes slightly longer than IF bandwidth reduction to lower noise, especially if many averages are required.

Also, changing the IF bandwidth after calibration results in **uncertain accuracy**.

How to Set Averaging	
Using front-panel <small>HARDKEY</small> [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press Avg2. then [Averaging]	<ol style="list-style-type: none">1. Click Response2. then Avg3. then Average

Average dialog box help

Programming Commands



Average ON Check to enable Averaging.

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 consecutive sweeps. When the number of sweeps = Average Factor, the averaging continues following the **Sweep Averaging formula**.

(Sweep) Restart Begins a new set of measurements that are used for the average. Applies only to Sweep averaging - NOT Point.

Point Each data point is measured the number of times specified by the Average Factor, and then averaged, before going to the next data point.

- On subsequent sweeps, averaging is automatically restarted by measuring each data point again the number of times specified by the Average Factor.
- Because measurements occur quickly in the background, the Average Counter is NOT updated.

Notes

- 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. The Average Counter is NOT updated for **Point** averaging.
- **Channel-wide scope-** Averaging is enabled and the factor is set for all measurements in a channel. The Average counter is displayed for each channel.
- **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 data processing map.](#)
- **Triggering** is implemented separately from Averaging. For example, setting averaging factor to 4 has NO effect on the number of triggers that are required to achieve 4 sweeps or 4 data points.
- **Unratioed** measurements - Although averaging unratioed (single receiver) measurements is allowed, you may see unexpected results.
 - The noise floor does not drop when averaging unratioed measurements as on ratioed measurements.
 - 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.

Sweep Averaging Formula

NewAvg = (NewData/n) + [OldAvg*(n-1/n)] 'where n = 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 clicking **Restart**.

[Learn more about Averaging](#) (scroll up)

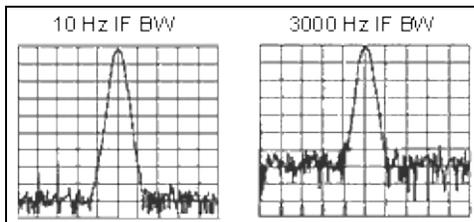
IF Bandwidth

The received signal is converted from its source frequency 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.

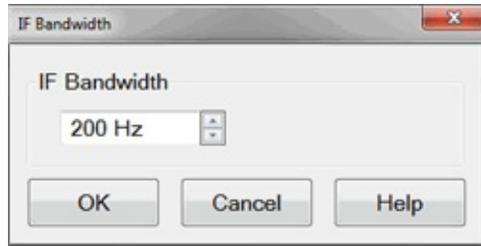
Reducing the IF receiver bandwidth reduces the effect of random noise on a measurement. Each tenfold reduction in IF bandwidth lowers the noise floor by 10 dB. However, narrower IF bandwidths cause longer sweep times.

- **Channel** - 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



How to set IF Bandwidth	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press Avg2. then [IF Bandwidth]	<ol style="list-style-type: none">1. Click Response2. then Avg3. then IF Bandwidth
IF Bandwidth dialog box help	Programming Commands



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 following is a list of selectable IF Bandwidths:

10 | 20 | 30 | 50 | 100 | 200 | 300 | 500 | 1k | 2k | 3k | 5k | 10k | 20k | 30k | 50k | 100k | 300k | 600k | 1.2M

[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 smooths trace noise and does not increase measurement time significantly.

Because Trace Smoothing follows Format in the 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.](#)

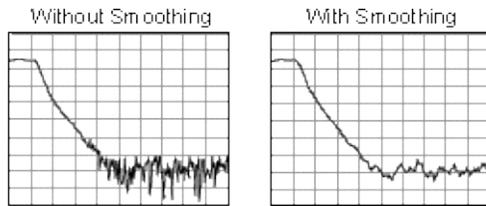
[See the data processing map.](#)

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



How to set Trace Smoothing	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press AVG 2. then [Smoothing] 	<ol style="list-style-type: none"> 1. Click Response 2. then Avg 3. then Smoothing

Smoothing dialog box help
Programming Commands

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 analyzer 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 no 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 these analyzer features that reduce or remove the effects of accessories.

- [power Slope to Compensate for Cable Loss](#)
- [Gating to Selectively Remove Responses](#)
- [De-embedding a 2-port device](#) (separate topic)

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.

Speed up PXIe VNA Measurements

The following features can be used together, or separately, to make faster VNA measurements:

Option 103 - Fast sweep mode.

- [Learn more about the feature.](#)
- [See the SCPI command.](#)

Use PXI-specific 'Trigger' commands

- [SCPI command to set index line logic.](#)
- [SCPI command to efficiently trigger multiple channels.](#)

Shared Memory Data Transfer

- [See SCPI commands](#)

See Also

- [General VNA Measurement Speed Techniques](#)

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 analyzer 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:

- $N * (N-1)$ 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 analyzer. This knowledge allows you to set up the measurement scenarios that are best for your applications.

[Learn more about Traces, Channels, and Windows](#)

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....**
3. Under **Trigger Source**, click **Internal**.
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 window.
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 SCPI, the following techniques can be used to transfer data quickly between the analyzer 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 Shared Memory Data Transfer.** [Learn how.](#)

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

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

1. Press **MACRO**
2. then [**Macro Setup**]

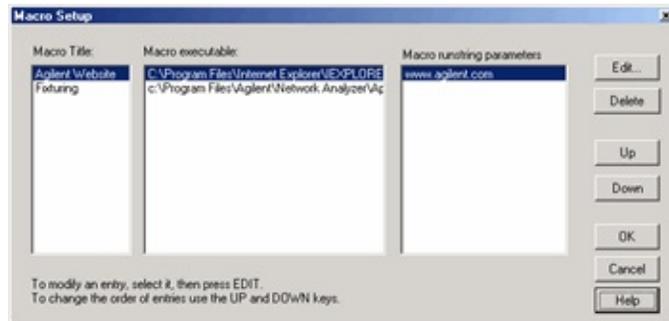
Using Menus

1. Click **Utility**
2. then **Macro**
3. then **Macro Setup**

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



Macro setup allows you to create 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 Keysight 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.Keysight.com/find/pna".

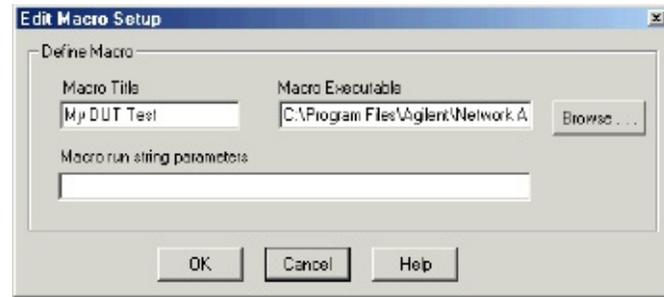
Edit Invokes the **Macro Edit dialog box**.

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 Add a title that appears in the softkeys and menu.

Macro Executable Set the complete path to the macro executable file. Click **Browse** to navigate to the macro executable file and establish the complete path to the file.

Macro run string parameters Optionally add parameters that are passed to the program referenced in the executable file.

[See Macro Setup dialog box](#)

How to Run Macros

Using front-panel HARDKEY [softkey] buttons

1. Press **MACRO**
2. then select the macro to run

Using Menus

1. Click **Utility**
2. then **Macro**
3. then select the macro to run

← Programming Commands →

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:

3-May-2012	Removed c models
17-Feb-2009	Added 25 limit
4-Sep-2008	Removed legacy content

Select a Calibration Type

The following calibration types are available in the PNA.

Cal Type	Interface	Accuracy	Thru Methods allowed
TRL Family	Both	Very High	All except Unknown Thru
SOLT	Both	High	All
Enhanced Response	SmartCal	High	Defined Thru or Flush Thru
QSOLT (Quick SOLT)	SmartCal	Medium	Defined Thru or Flush Thru
1-Port Reflection	Both	High	Not Applicable
Open/Short Response	Unguided	Low	Not Applicable
Thru Response	Unguided	Low	Known Thru or Flush Thru

[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 may be required.

- [Learn more about TRL family cal](#)
- For more information on modifying standards, see [Calibration Standards](#).

Calibration Method: [SmartCal](#), [Unguided Calibration](#)

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 calcs:

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**, **ECal**

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**, **ECal**

General Accuracy: High

Standards Required: (SHORT, OPEN, LOAD) or ECal 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 / Transmission 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](#) and Guided Cal from the 'Select DUT Connectors page', check [Modify Cal](#), then click **Next**.

General Accuracy: Low

Standards Required: THRU

Isolation: One LOAD for each PNA test port.

Systematic Errors Corrected:

- Frequency response transmission tracking
 - Isolation
-
-

Last modified:

24-May-2013 Edit transmission resp

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

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

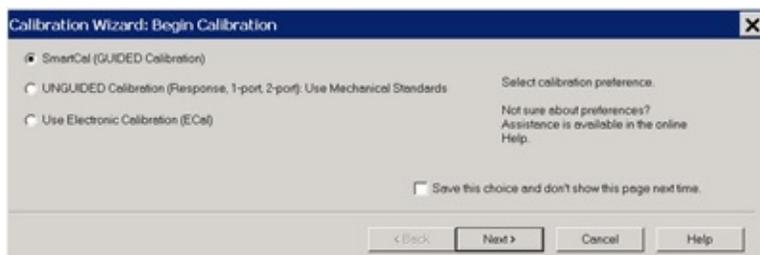
1. Press **CAL**
2. then **[Start Cal]**
3. then **[Cal Wizard]**

Using Menus

1. Click **Response**
2. then **Cal Wizard**

[Programming Commands](#)

Calibration Wizard Begin dialog box help



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.

- 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.
- Only one Cal Kit can be used.
- Can NOT use Offset Load standard.

Why do we offer the Unguided Cal?

The Unguided Cal is familiar to legacy 8510 and 8753 customers. However, in those analyzers, you selected the cal kit, but not the connector type or gender. Therefore, the calibration engine did NOT know if the connectors were [insertable](#) or non-insertable. In the case of non-insertable, many people would use an undefined 'bullet' THRU and ignore the measurement inaccuracy.

The Guided Cal Wizard asks for the DUT connectors and guides you through the most accurate calibration possible.

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.
- See Also: [Perform a 4-Port Cal with a 2-Port ECal Module](#)

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 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.

The calibration standards can be measured while viewing any 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](#).

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:

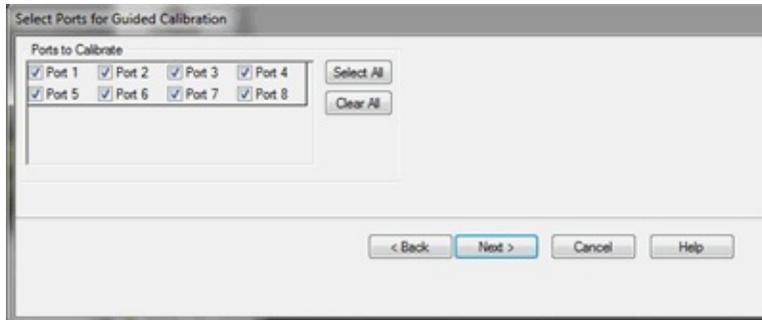
- ALL Cals **EXCEPT Open, Short, and Thru Response** Cals.
- ECal on one or more ports.
- TRL - [Learn how to do TRL cals](#)

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 following dialog boxes appear when performing a Guided calibration on standard channels.

Select Ports for Guided Calibration dialog box help



Allows you to select ports to calibrate.

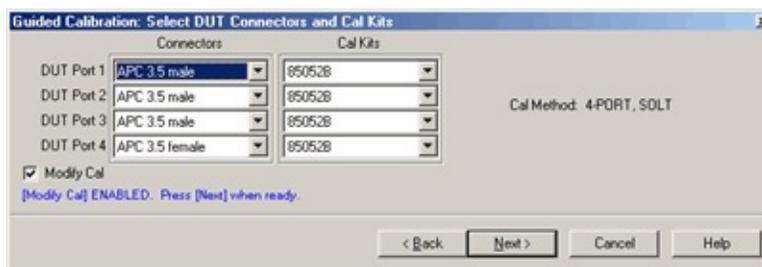
Cal Type Selection Select the number of ports to calibrate.

N Port Cal Configuration If not calibrating all ports, specify which ports to calibrate.

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.

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.

Cal Kit Notes

85056K

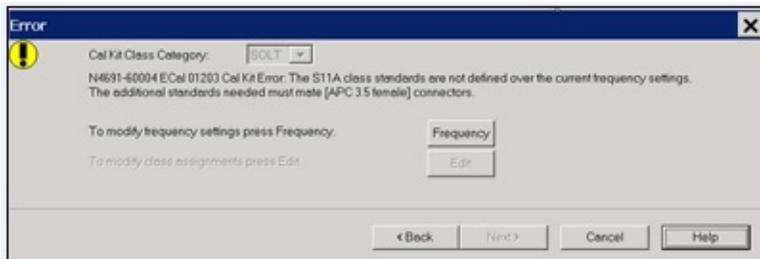
The 85056K definitions in the analyzer are for 2.92mm standards (2.4mm plus 2.92 adapters). To calibrate 2.4 mm connectors using the 85056K cal kit, select 85056A as the cal kit when you need the sliding load. Otherwise, select 85056D as the cal kit. Both the 85056A and the 85056D kits contains exactly 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.

Modify Cal Check, then click Next, to **Modify Cal** (Standards AND Thru Method).

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.

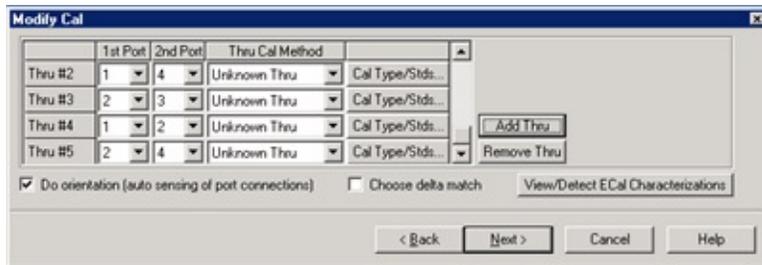
Frequency Change the frequency range of the active channel.

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/ Std

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 analyzer automatically senses the model and direction in which an ECal module port is connected to the test 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.

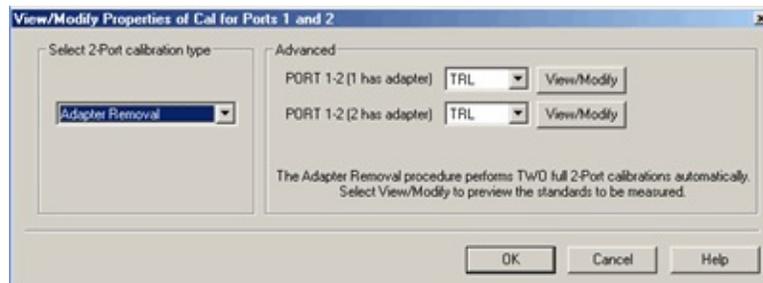
Orientation occurs first at the middle of the frequency range that you are calibrating. If

a signal is not detected, it tries again at the lowest frequency in the range.

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 connected ECal modules.

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** - Available ONLY when a TRL cal kit was selected for the lowest port number of the port pair.
- **QSOLT** Available ONLY 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** Available ONLY when "Defined Thru" or "Flush Thru" is selected. "EnhResp 2 <= 1" refers to the receive port 2 and source port 1.
- **Transmission Response** Available ONLY when "Defined Thru" or "Flush Thru" is selected, when Mechanical Cal is selected, and when 2 ports are being calibrated. "TransResp 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.

Calibration Steps dialog box help



Note: 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.

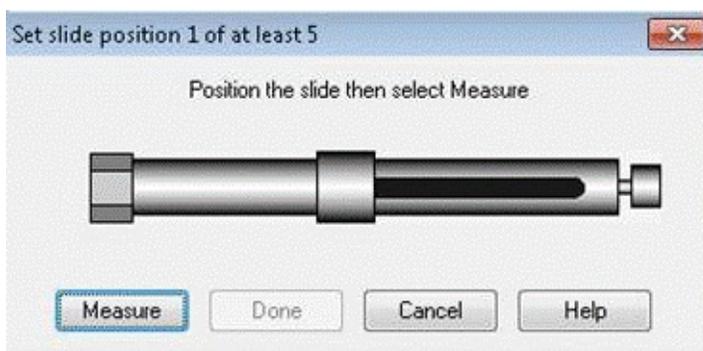
Done Click **after** a standard is re-measured and all measurements for the calibration are complete.

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.

To Measure a Sliding Load:

1. Connect the sliding load to the measurement port following the procedure described in the Calibration Kit User's and Service Guide.

Note: Do NOT set the center conductor to be an interference fit with the center conductor of the testport.

2. Position the sliding element, then click **Measure**. Do not move the sliding element until measurement is complete.

Note: The direction in which the slide moves is NOT important. You can start with the slide at the front and move it backward or start at the back and move it forward. To minimize stability errors it is important to start at one end and move it in the same direction for each of the measurement steps.

3. Measure the sliding load for at least **five** positions for best accuracy.

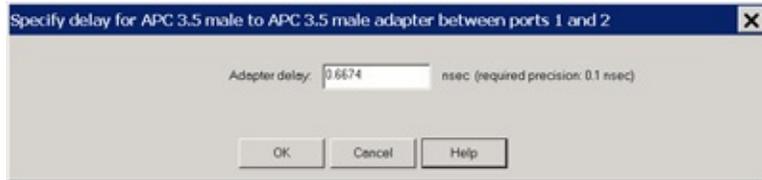
Note: The positions of the sliding element should cover the full length of the slide, but be unequally spaced to reduce the possibility of overlapping data points. Most sliding loads have marks for each slide position.

4. Click **Done** after the final measurement.
5. Disengage sliding load lock (if available), and remove sliding load from the measurement port.
6. Measure the remaining standards.

How to Verify Sliding Load Calibration Measurements

Once the calibration is completed, the sliding load can be measured again. The magnitude of the return loss should remain nearly constant as the slide is moved. If the slide spacing was not adequate due to slide position selections, there will be frequency ranges where the magnitude will not remain nearly constant.

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.

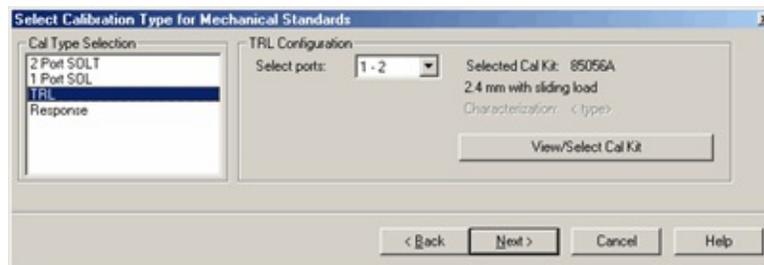
The **Calibration Complete** dialog box appears after all standards are measured.

Unguided Calibration

Programming Commands

The following dialog boxes appear 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.

Calibration Type Selection

- **2-Port SOLT**
- **1-Port SOL**
- **TRL** - [Learn more about TRL](#)
- **Response** - Reflection and Thru (if the active measurement is transmission)

Cal Configuration If not calibrating all test ports, specify which ports to calibrate.

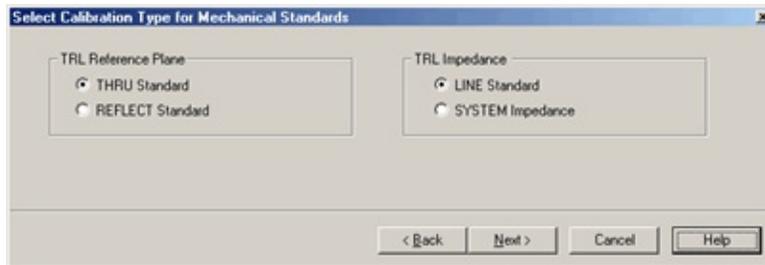
Back Return to [Cal Wizard Begin](#) dialog. If checked, you can clear the [Save Preferences](#) checkbox to see the Begin page when the Cal Wizard begins.

View/Select Cal Kit Click to invoke the [Select Cal Kit dialog box](#).

Note: When selecting a Cal Kit with an impedance other than 50 ohms (Waveguide = 1 ohm), it is **NO LONGER NECESSARY** to change the [System Impedance](#) setting before performing a calibration. The impedance for the calibration is now derived from the Cal Kit impedance.

Next Click to continue to [Measure Mechanical Standards](#) dialog box.

Select Cal Type dialog box help



This dialog box appears ONLY when 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

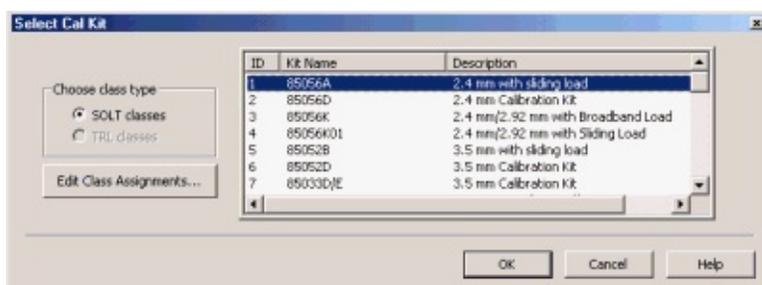
LINE Standard Specifies that the characteristic impedance of the LINE standard should be used as the system impedance. This ignores any difference between Offset Z0, Offset Loss, and System Z0.

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

Edit Class Assignments Allows modification of the selected Cal Kit class assignments.

- To learn to substitute other calibration kits, see [Advanced Modify Cal Kits](#)
- Unguided Cal can access only mechanical cal kits #1 through #95, although more cal kits can be imported. [Learn how.](#)

Note: When selecting a Cal Kit with an impedance other than 50 ohms (Waveguide = 1 ohm), it is **NO LONGER NECESSARY** to change the **System Impedance** setting before performing a calibration. The impedance for the calibration is now derived from the Cal Kit impedance.

Measure Mechanical Standards dialog box help



Note: 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. [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.

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 previously-saved Cal Set names.

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 typing tool that can be used from the front panel.

Save Saves the Cal Set to the new Cal Set name.

Learn about [User Cal Sets](#)

Calibrate All Channels

"Cal All" allows you to calibrate multiple channels in a single calibration session. This not only reduces the number of connections that need to be made, but also the number of cal standard measurements that must be performed.

In this topic:

- [Features](#)
- [Limitations](#)
- [How to perform a Cal All Channels Calibration](#)
 - [Select Channels dialog](#)
 - [Calibration Attenuator Settings dialog](#)
 - [Select DUT Connectors and Cal Kits dialog](#)
 - [Power Cal Settings dialog](#)
 - [Cal Steps dialog](#)
 - [Finish](#)

Other Cal Topics

Features

Cal All offers a single, optimized calibration procedure for all channels (with some limitations, see below). The optimizations include:

- Minimizing the number of physical connection of standards.
- Minimizing the number of power meter calibration sweeps.
- User-settable power levels for S-Parameter as well as power calibration steps.
- Accounting for different switch and attenuator settings among different channels. This reduces the number of measurements required to characterize different switch/attenuator settings (channel setup differences).
- Cal All will produce the same number and format of Cal Sets (error terms) that would be realized had the calibrations been performed one at a time.

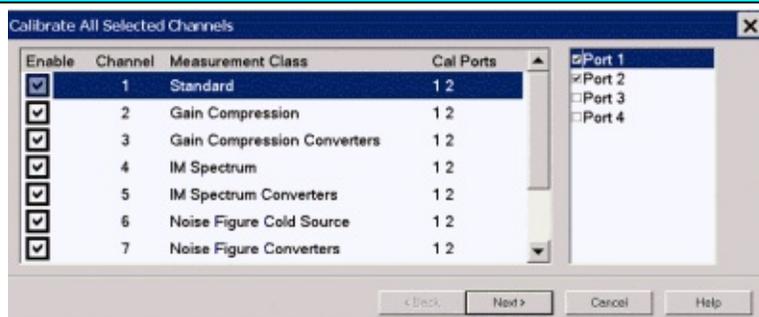
- Calibrate External Sources that are connected to the analyzer using Configure an External Source.

Limitations

- Cal All is performed at one IFBW.
- All channels that are calibrated are forced into **stepped sweep mode**.
- All channels to be calibrated **MUST** have the same **cal reference plane**. In other words, Cal All cannot compensate for any path changes that occur external to the analyzer.

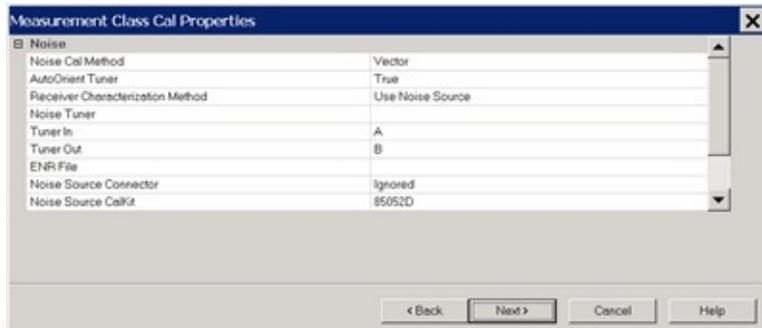
How to perform a Cal All Channels Calibration	
<p>Using front-panel HARDKEY [softkey] buttons</p> <ol style="list-style-type: none"> 1. Press CAL 2. then [Start Cal] 3. then [Cal All Channels] 	<p>Using Menus</p> <ol style="list-style-type: none"> 1. Click Response 2. then Cal 3. then Start Cal 4. then Cal All Channels
<p>◀ Programming Commands ▶</p>	

Selected Channels dialog box help



1. Check the channels to be calibrated.
2. Check the ports to be calibrated.
3. Click **Next>**

Measurement Class Cal Properties dialog box help



Confirm or change the following unique cal properties for each channel to be calibrated. Click a link to learn about these properties.

The properties with **(NOT available in Cal All)** are NOT available in a Cal All calibration as they are in a stand-alone calibration.

Gain Compression and GCX

- No unique properties.

Noise Figure and NFX

Swept IMD and IMDX

IMSpectrum

- No unique properties.

SMC (VMC is NOT offered)

Standard Channel

The power cal is optional only if none of the selected channels require a power cal.

Calibration Attenuator Settings dialog box help



This dialog shows the Power, Attenuator, and IFBW settings for the Cal All calibration. The default values for the Cal All session are the preset values of a standard S-parameter channel. These values are not necessarily the same as those of the channels that are selected for calibration. When there are differences in measurement path (switch) settings between the Cal All channel and the selected channels, these differences are detected by Cal All and additional measurements are made for each path condition. These additional measurements allow Cal All to

produce error terms appropriate for each of the selected channels. In general, the Cal All session should be performed at a power level that is high enough to prevent noise in the error terms. However, an increase in power could cause compression or damage to the analyzer receivers. The following settings allow you to increase the power level ONLY during the Cal All session.

Power Limit (Disable)

Cal All shows you when power limits are enabled. This setting provides you a convenient way to TEMPORARILY disable these limits in order to take advantage of the power settings available in Cal All. If power limits are on, your DUT is probably a high-gain device and the attenuator settings in your channels are high resulting in lower power at the cal reference plane. This lower signal can result in noisier measurements during the acquisition of cal. This situation is precisely what Cal All is intended to improve. Cal All allows you to configure the calibration conditions for better signal-to-noise performance during the cal while leaving your DUT conditions alone. You can elect to clear the “Disable Power Limits during cal” checkbox when you prefer to calibrate at a higher power level than is allowed by your limit. The limit is restored after the Cal All session.

Source / Receiver Attenuator

By default, the Cal All calibration is performed with Source and Receiver attenuators set to 0. Change the Source or Receiver attenuator settings when external hardware (such as a booster amplifier) would cause the analyzer receivers to be compressed or damaged.

You may also want to change the attenuator or path configuration settings to force the cal channel to match settings of the selected channels. If all of the selected channels are set to identical hardware settings, it may be better to apply these settings to the cal channel. For example, if your channels all use a 5 or 10 dB attenuator step at port 1, you might elect to change the Cal All channel to use the same low attenuator settings. This will result in the cal measurements being made under the same path conditions as the channel and it will eliminate the need to mathematically compensate for the difference. However, if large attenuator values are used, the default Cal All settings will likely improve your results.

S-Parameter Cal Port Power

Set the power level at which the S-Parameter cal is performed.

Power Offsets

Power Offsets are channel-scoped. Consequently, offsets that you already set are NOT automatically copied to the Cal All session. This setting allows you to also apply a Power Offset during the Cal All session. [Learn about Power Offsets.](#)

Noise Reduction

This button accesses the following dialog for settings that help reduce trace noise and the noise floor which can lead to better dynamic range and more accurate

measurements. [Learn more.](#)



IF Bandwidth

Set the IFBW used to perform the Cal All calibration. The default IFBW setting of 1 kHz is a good nominal setting for most measurements. Lowering the IFBW removes noise from the calibration measurement, but also causes slower sweeps.

Always ON

Check to enable averaging.

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 consecutive sweeps. [Learn more.](#)

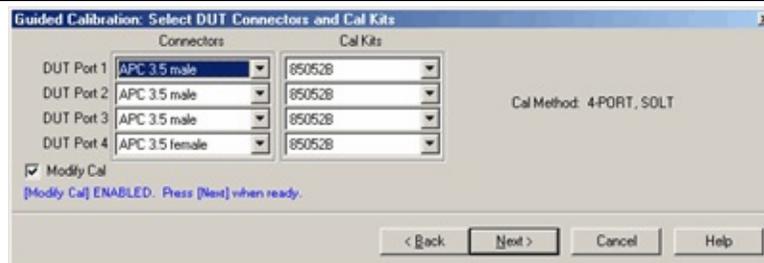
(Sweep) Restart Begins a new set of measurements that are used for the average. Applies only to Sweep averaging - NOT Point.

Point Each data point is measured the number of times specified by the Average Factor, and then averaged, before going to the next data point. [Learn more.](#)

Mechanical Devices

This button accesses a dialog that shows the settings for all active channels. These settings are shown side-by-side for easy comparison. [Learn more.](#)

Select DUT Connectors and Cal Kits dialog box help

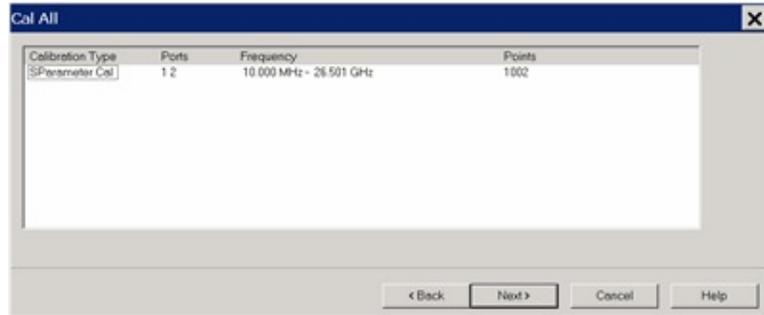


For each DUT port:

- Select the connector at the calibration reference plane (where the cal standards will be connected).
- Select the cal kit to be used.

Check **Modify Cal** to change the Thru method. An Unknown Thru cal is performed by default. [Learn about THRU methods.](#)
[Learn more about this dialog.](#)

Cal All Summary dialog box help



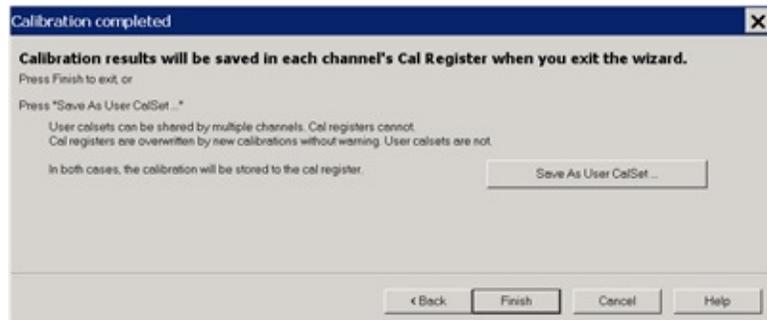
This page is a summary of the Cal All settings. Confirm the settings, then click **Next >** or **< Back** to change settings.

Cal Steps dialog box help



Follow the prompts to connect each standard. Then click **Measure**.
Click **Re-measure** if necessary.
Then click **Next >**

Finish Cal dialog box help



Click **Finish** to save the Cal All session results to Cal Registers.

Or click **Save As User CalSet**, then enter a prefix title. The Meas Class and channel number are appended to this prefix to save to a User Cal Set for each calibrated channel.

[Learn more about this dialog.](#)

Using Calibration Sets

- [What are Cal Sets](#)
- [Cal Registers and User Cal Sets](#)
- [How to Manage and Apply Cal Sets](#)
 - [Calibration Selection dialog box help](#)
 - [Cal Set Properties dialog box help](#)
 - [Select Cal Set -- Choose Stimulus Settings dialog box help](#)
- [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 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 - *The “Standards data” container in the Cal Set is intended for internal use only. External access is provided for use in diagnosing calibration problems. Users should not form any expectations as to the presence of the data or the naming conventions used.*
- Error term data

- GUID (**G**lobally **U**nique **I**Dentifier)

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.

Calibration Registers

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 'legacy' Vector Network Analyzers.

- Every channel has ONE dedicated Cal Register. They are named CH n _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**.

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 analyzer 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

The analyzer 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 VNA Application.

How to select and apply a Cal Set to the active channel

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

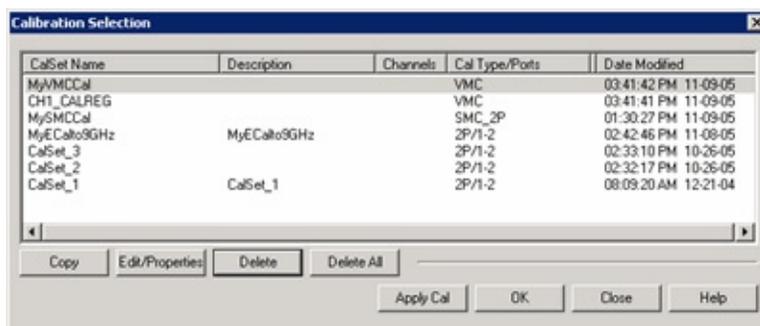
1. Press **CAL**
2. then **[Manage Cals]**
3. then **[Cal Set]**

Using Menus

1. Click **Response**
2. then **Cal**
3. then **Manage Cals**
4. then **Cal Sets**

◀ Programming Commands ▶

Calibration Selection dialog box help



This dialog allows you to manage and apply Cal Sets

Although the number of Cal Sets you can have is limited only by the amount of analyzer 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 calibration

+ - Indicates a Power Correction is included in the Cal Set

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.

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 Starts the [Cal Set Properties](#) dialog box. This allows you to view all of the Cal Set properties.

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](#).

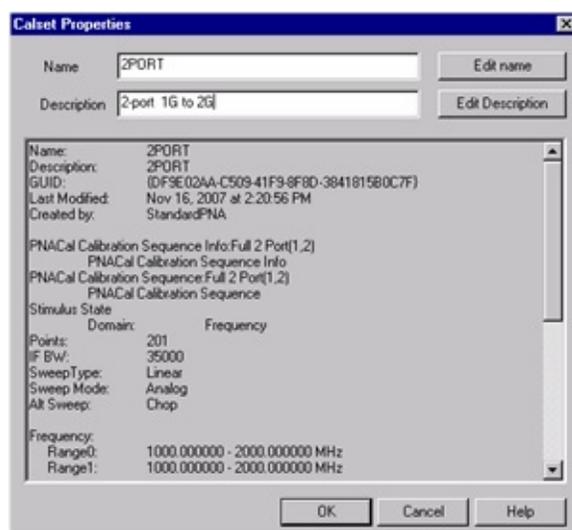
Unselect Available ONLY if the selected Cal Set is being used by the active

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.

This dialog box is started in two ways:

1. From the Edit / Properties button on the Calibration Selection dialog box.
2. From the main menu/softkeys: Click **Response**, then **Cal**, then **Properties** (this method does not allow you to change the Cal Set name).

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.

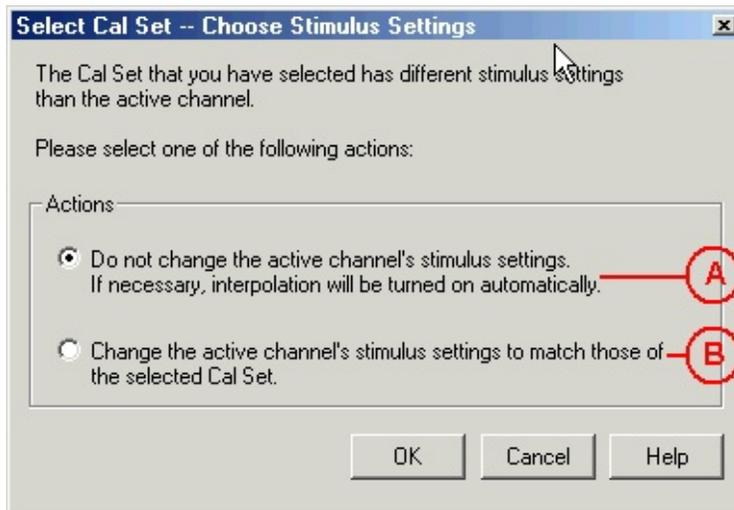
Cal Set Properties Lists descriptive information and stimulus conditions of the Cal Set.

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 dialog box help



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 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

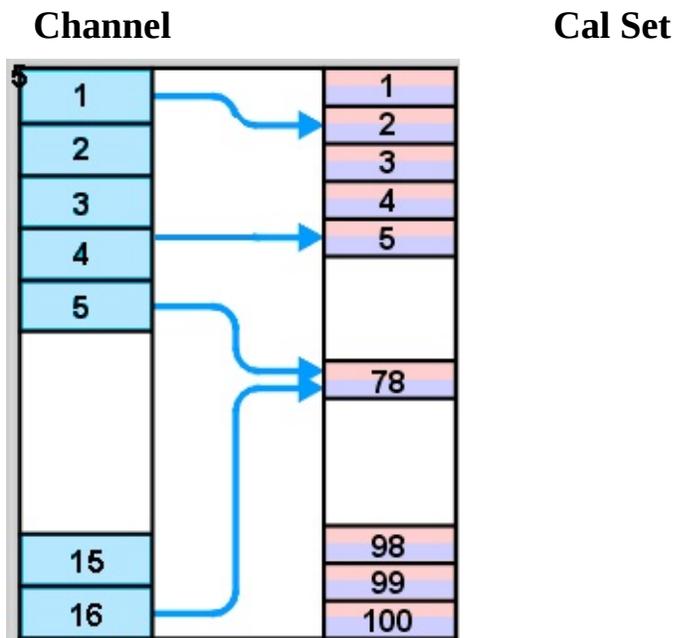
It is possible to do one calibration, then apply it to several channels.

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.

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 Accuracy](#)

[See other Calibration Topics](#)

How to set Error Correction

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

1. Press **CAL**
2. then **[Correction ON/off]**

Using Menus

1. Click **Cal**
2. then **Correction ON/off**

[Programming Commands](#)

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:

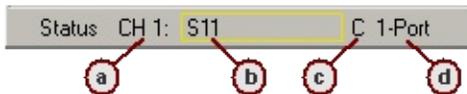
- If Interpolation can NOT be performed, a dialog box will ask if you would like to **change the stimulus settings** to those of the applied calibration. Click OK or Cancel.
- If Interpolation can be performed, the stimulus setting will change and correction turned ON.

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:



- Active Channel
- Measurement parameter
- Correction Level (see description below)
- Calibration type

Correction Level	Accuracy
C Full	Highest
C* Interpolated	Uncertain
CΔ Changed	Uncertain
No No Correction	Lowest

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](#).

CΔ 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

Using front-panel

Using Menus

HARDKEY [softkey] buttons

1. Press **CAL**
2. then **[More]**
3. then **[Interplotion ON/off]**

1. Click **Cal**
2. then **More**
3. then **Interplotion ON/off**

← Programming Commands →

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. In general, the magnitude and phase stimulus from the PNA and the response from the DUT need to be smooth and continuous for measurement interpolation to give accurate results.

Significant measurement inaccuracy WILL occur when the phase shift response between measurement points increases changes more than 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 frequency span between measurement points becomes much greater.
- when measurement frequencies are above 10 GHz where phase changes happen more rapidly.
- when interpolating across frequency band crossings. [Learn more about band crossings](#).

Note: When the interpolation algorithm encounters an abrupt or large change in the response magnitude or phase, such as can occur at band crossings, large interpolation errors can be included in the displayed data. These errors can be seen as steps or spikes. If this occurs, consider turning off interpolation, changing the measurement parameters, or creating [sweep segments](#) that skip over the band crossings.

Last modified:

- 19-Feb-2013 Added Accuracy note (SS)
- 3-Sep-2008 Removed legacy content
- 12-Sep-2006 Added link to programming commands

Using ECal

This topic discusses all aspects of performing an ECal:

- [ECal Overview](#)
- [Connect ECal Module to the Analyzer](#)
- [How to Perform a Calibration Using ECal](#)

See Also:

[ECal User-Characterization](#)

[Perform a 4-Port Cal with ONE 2-Port ECal Module](#)

[Restore ECal Module Memory](#)

[See other Calibration Topics](#)

ECal Overview

ECal is a complete solid-state calibration solution. Every ECal module contains electronic standards that are automatically switched into position during a measurement calibration. These electronic standards have been measured at the factory and the data stored within the memory of the ECal module. The analyzer uses this stored data, along with the 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
- And so forth...

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:

Model	Compression level	Damage level
N469x series	-5 dBm	+10 dBm
N4432A series N4433A series	-7 dBm	+20 dBm
N4431x series	+7 dBm	+20 dBm
8509x series	+9 dBm	+20 dBm

The power level can be increased after calibration with minimal impact on measurement accuracy.

Connect ECal Module to the Analyzer

ECal modules are controlled and powered through a USB connection. When you connect the module, the type of module, frequency range, and connector type are automatically recognized.

For PXI modules, connect the ECal module to a USB port on the remote or embedded controller.

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](#). See Also: [Perform a 4-Port Cal with ONE 2-Port ECal Module](#)

Connect the ECal module ports to the analyzer ports. During the calibration process the analyzer 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 to 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 Menus

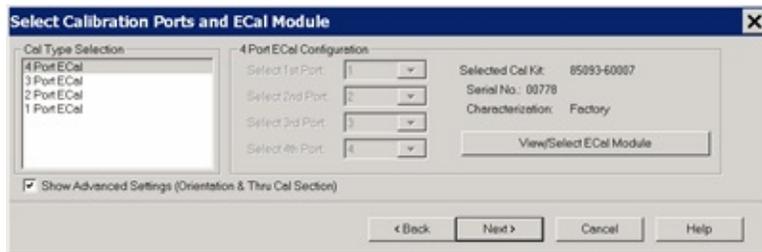
1. Click **Response**
2. then **Cal**
3. then **Cal Wizard**

Programming Commands

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

3 Port ECal

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 USB. Also, **Select the User Characterization** within the module. Learn more about **User Characterization**.

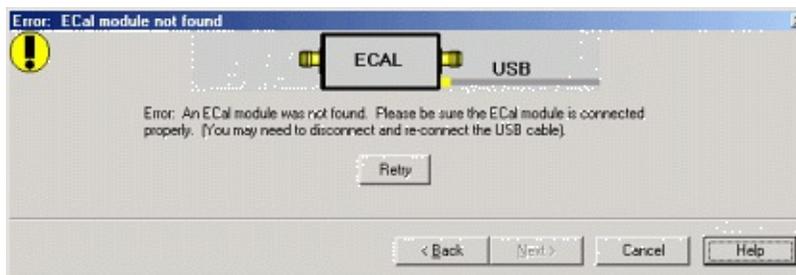
Show Advanced Settings Check to display the **Advanced Settings** when **Next** is clicked.

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: ECal isolation is not performed. The inherent isolation of the analyzer 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



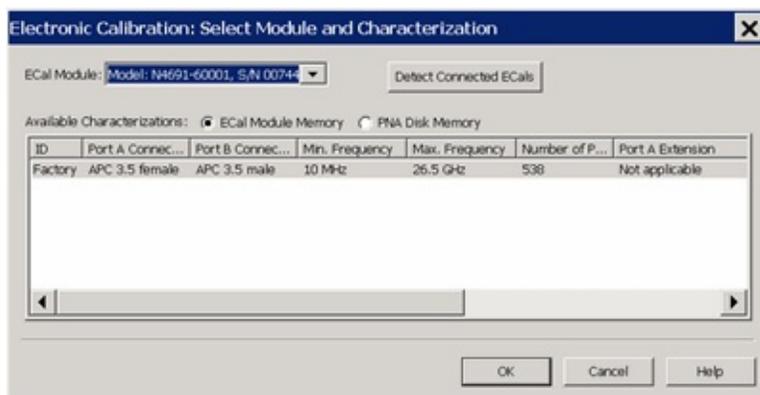
When this dialog appears, the ECal module is not connected or has not been recognized by the network analyzer.

Retry Check the USB connections and click to continue.

Notes:

- If your ECal module is not detected, try to unplug, then reconnect to the USB.
- 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.
- See [Connect ECal Module to USB](#).

Select Module and Characterization dialog box help



ECal Module Select one of the ECal modules that are connected to the analyzer.

Detect Connected ECals Click to rescan the USB for ECal modules.

Available Characterizations

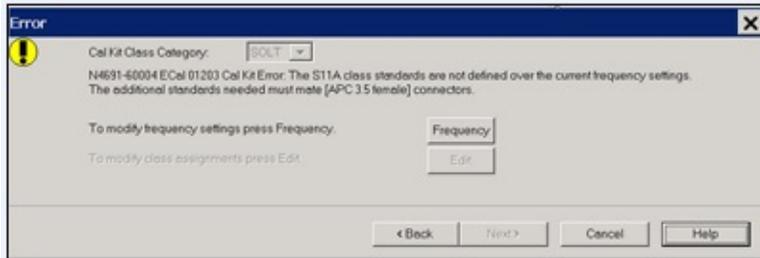
ECal Module Memory - Displays the factory and [user characterizations](#) that are stored in the ECal module.

Disk Memory - Displays the user characterizations that are stored in Disk Memory.

[Learn more User Characterizations in Disk Memory.](#)

Select either the characterization data to use for the calibration. Once selected, that characterization becomes the default selection until the analyzer is turned OFF and restarted. When restarted, **Factory** again becomes the default selection.

Error: Frequency Range dialog box help



When this dialog appears, 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.

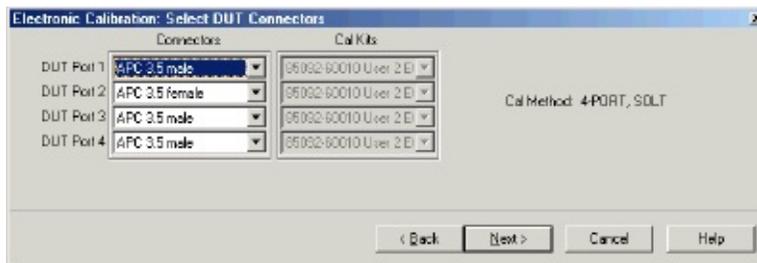
Frequency Change the frequency range of the active channel.

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

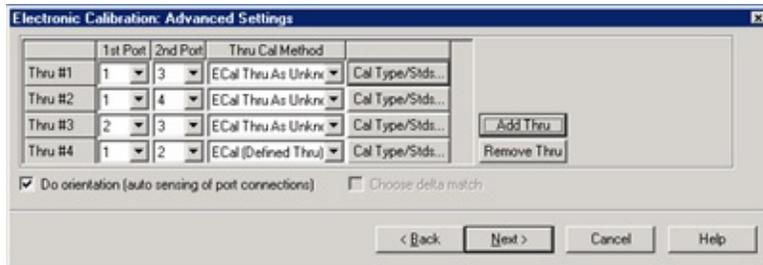


If the ECal module or selected User Characterization has more than one connector type, then the following dialog box is presented which allows you to describe the DUT connector type. Otherwise, click next to proceed to **Advanced Settings** (if checked) or **ECal Steps**.

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](#).
- For Balanced measurements, [learn which Thru paths to select](#).

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

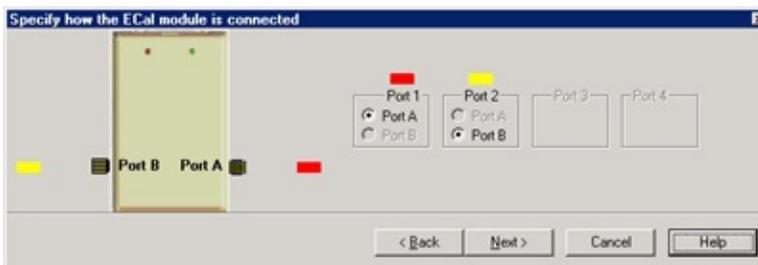
When this box is checked (the default setting) the PNA automatically senses the model and direction in which an ECal module port is connected to the PNA ports. If power to the ECal module is too low, it will appear as if there is no ECal module connected. If you use low power and are having this problem, 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.

Choose delta match

Available only when a Delta Match Cal is required.

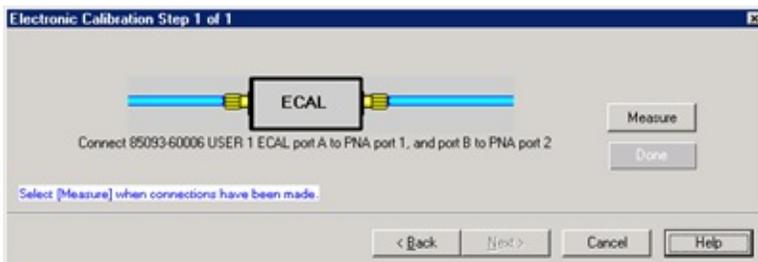
- Check, then click **Next** to invoke the **Select Cal Set for Delta Match** dialog box.
- Clear - The Cal Wizard uses the Global Delta Match Cal if available.



Specify how the ECal 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.](#)

Last modified:

2-Aug-2011	Added 4-port Cal with 2-port ECal
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](#)

[Perform a 4-Port Cal with a 2-Port ECal Module](#)

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 disk memory. 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 analyzer 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 analyzer 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.support.keysight.com/pna/apps/applications.htm>.

- 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 with a mechanical cal kit calibration versus an ECal extended User Characterization calibration.
- You can save up to 12 User Characterizations in a single ECal module. Previous releases allowed up to 5. There are memory limitations. The analyzer will determine if the contents of a User Characterization will fit inside the module before it is performed.
- A User Characterization can be performed remotely. [See programming commands.](#)

User Characterizations can be saved to **Disk Memory**. [Learn how.](#)

This feature provides the following benefits:

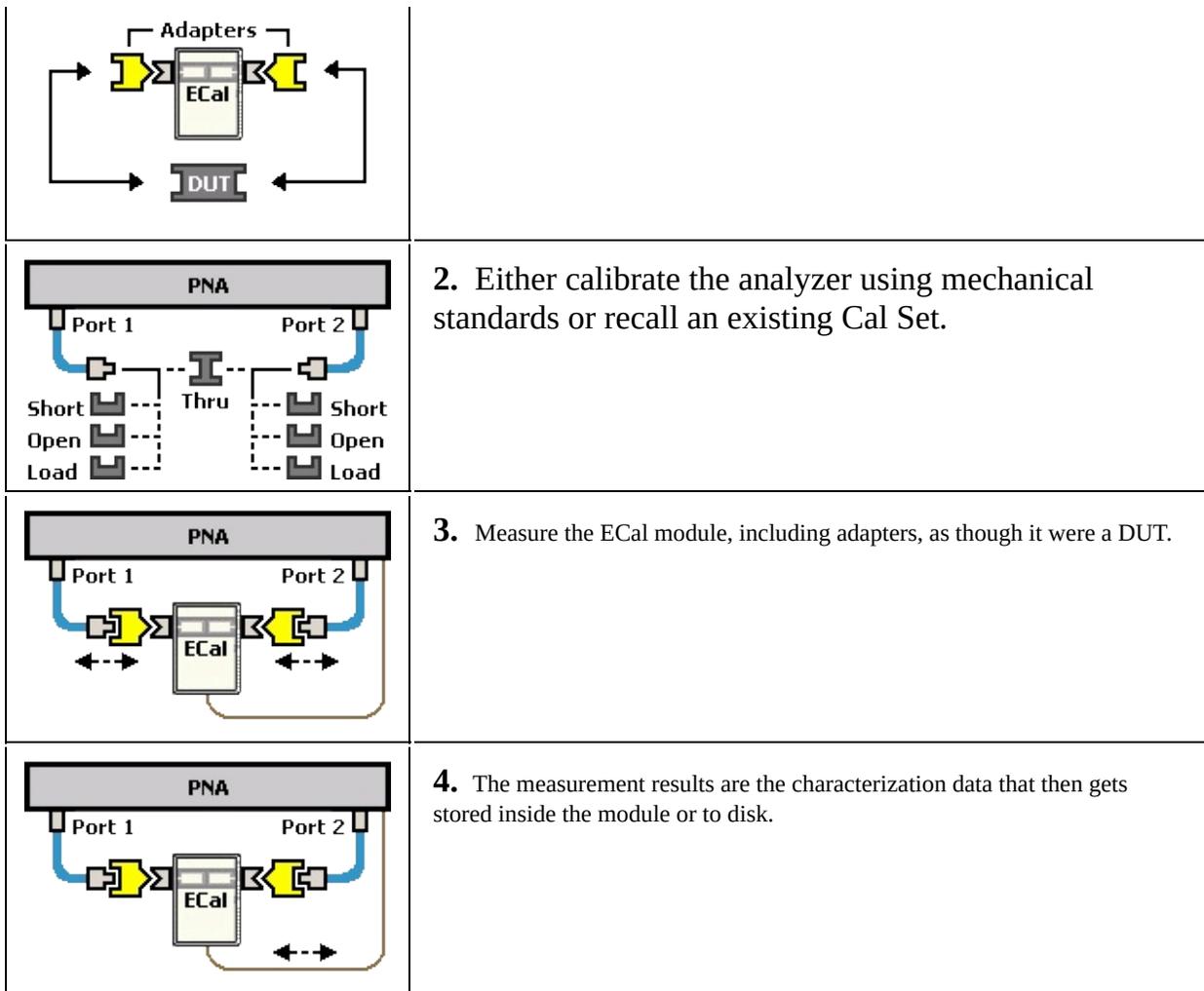
- A User Characterization using connectors that are NOT included in the [supported connector table](#) can NOT be stored to the ECal module. But when stored to disk memory, ANY connector type is allowed by firmware using a description of any length for the User Characterization.
- There is NO limit to 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 analyzer.
- 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 analyzers.

[Learn how to Manage User Characterization in Disk Memory.](#)

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.



Note

A 2-port analyzer 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 analyzer 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 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 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 the analyzer 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

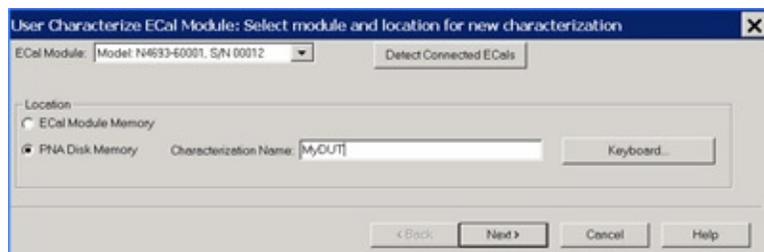
1. Press **CAL**
2. then **[More]**
3. then **[ECal]**
4. then **[Characterize ECal Module]**

Using Menus

1. Click **Response**
2. then **Cal**
3. then **More**
4. then **ECal**
5. then **Characterize ECal Module**

◀ **Programming Commands** ▶

Select Module and Location dialog box help



ECal Module Select one of the ECal modules that are connected to the analyzer.

Detect Connected ECals Click to rescan the USB for ECal modules.

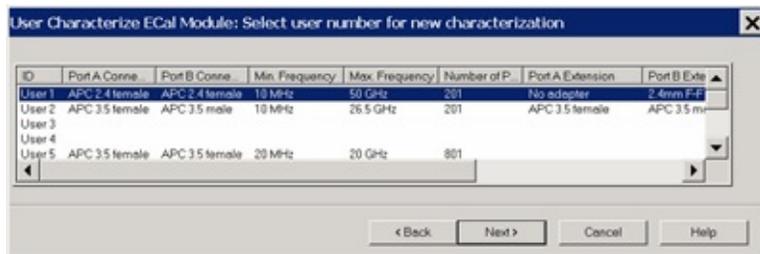
Location

- **ECal Module Memory** Click Next to see the following dialog.
- **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 disk memory.
 - See the benefits of storing the User Characterization to disk Memory.

Keyboard Launches a keypad that can be used to type a characterization name from the analyzer front panel.

Next Click to continue to the **Select Connectors for the Characterization** dialog box. See note regarding extended frequency use.

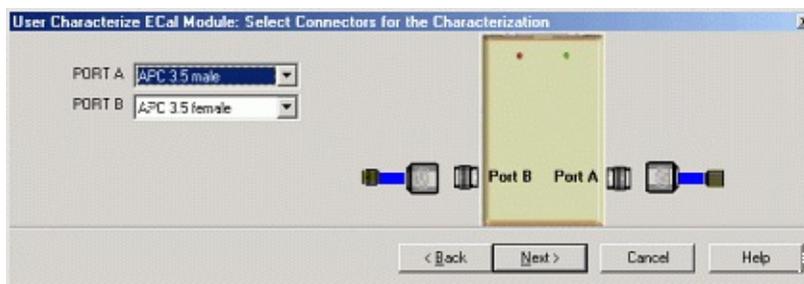
Select User Number for new characterization dialog box help



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](#) can NOT be stored to the ECal module. But when stored to disk memory, ANY connector type is allowed. [Learn more about storing to 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.

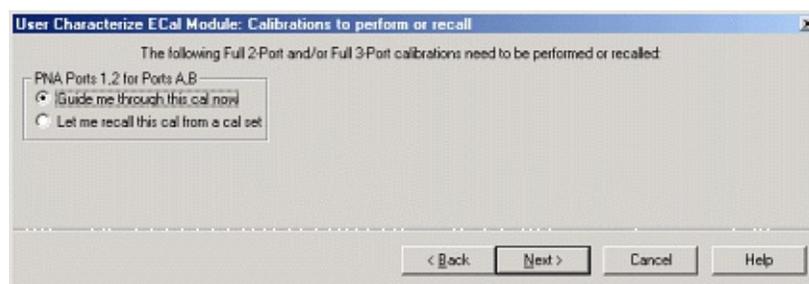
PORT B Lists the connector types available for Port B.

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 analyzer 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.

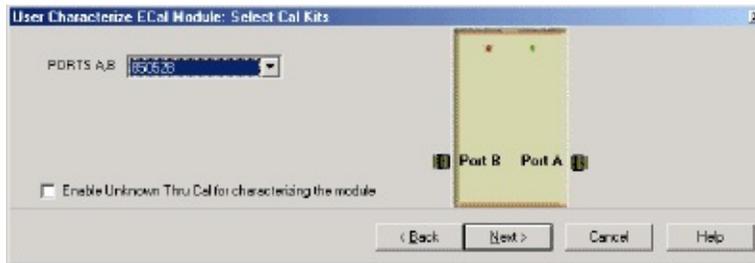
Guide me through this cal now Click to perform a Guided calibration. A calibration kit is required for each connector type.

If more than one calibration is required, the following 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.

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.

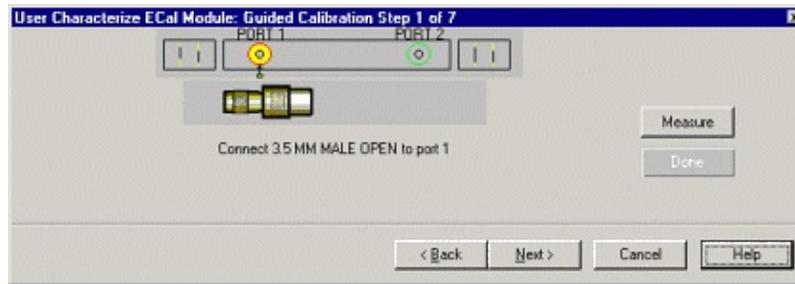
Select Cal Set Click to open the **Select A Cal Set** dialog box.

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.

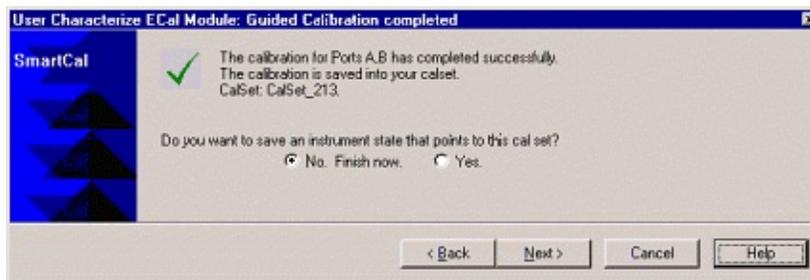
The following values were estimated from the measurement. Most of the time, they are adequate. However, for CW sweep or frequency sweep with large step sizes, the accuracy of the values may be improved.

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



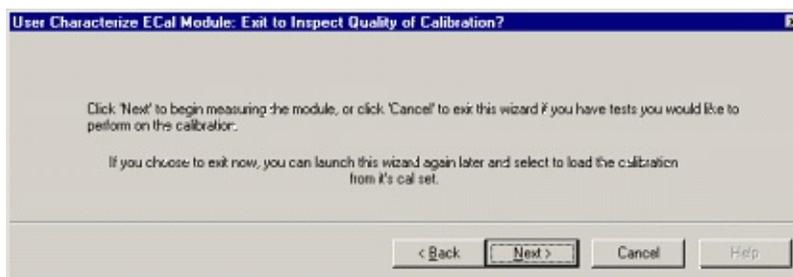
Allows you to finish the calibration and continue to the next characterization steps.

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.

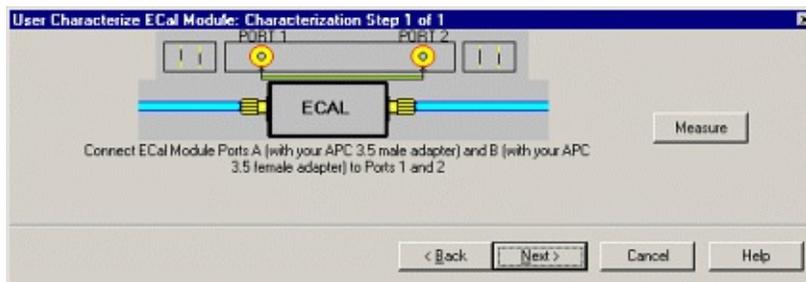
Next Click to continue to the **Characterization Steps** dialog box.

Cancel Exits the Calibration.

To return to the current step:

1. Start User Characterization.
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

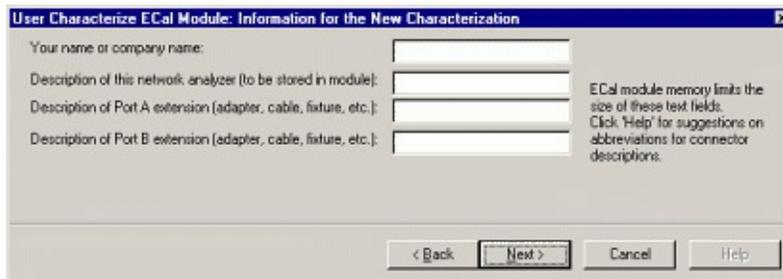


Describes the instructions for each measurement required for characterization.

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.

Suggestions for connector abbreviations

To minimize the number of characters, we suggest using 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 Disk Memory.](#)

Connector Type	3-Character Code
1.0 mm female	10F
1.0 mm male	10M
1.85 mm female	18F

1.85 mm male	18M
2.4 mm female	24F
2.4 mm male	24M
2.92 mm female	29F
2.92 mm male	29M
3.5 mm female	35F
3.5 mm male	35M
7-16 female	16F
7-16 male	16M
Type F female	F7F
Type F male	F7M
N50 female	N5F
N50 male	N5M
N75 female	N7F
N75 male	N7M
APC 7	7MM
K-band waveguide	KBW
P-band waveguide	PBW
Q-band waveguide	QBW
R-band waveguide	RBW
U-band waveguide	UBW
V-band waveguide	VBW
W-band waveguide	WBW
X-band waveguide	XBW

Next Click to continue to the **Write Characterized Data to the ECal module** dialog box.

Write Characterized Data dialog box help



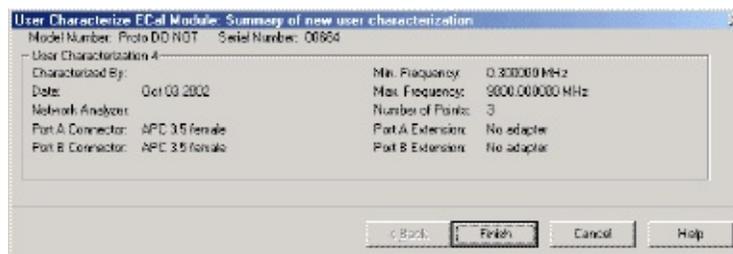
User Characterization and factory characterization data is written to either the disk memory or the ECal module memory.

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 if 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.support.keysight.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).

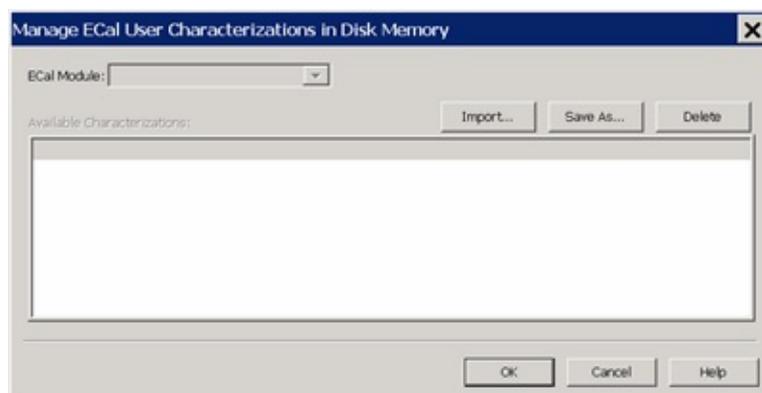
Manage ECal User Characterizations in Disk Memory

Normally, User Characterizations that are stored in 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 analyzer.

How to Manage ECal User Characterizations in Disk Memory	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press CAL 2. then [More] 3. then [ECal] 4. then [Manage ECal Disk Memory] 	<ol style="list-style-type: none"> 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 do either of the following:

- Save an existing User Characterization in disk memory to an *.euc file.
- Load a previously saved *.euc file for use on the analyzer with the specified ECal module.

Learn more about User Characterizations stored to Disk Memory.

ECal Module Select an ECal Module from the list for which User Characterizations are currently stored in disk memory.

Save As Saves a User Characterization that is currently in 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 analyzer to be used with the selected ECal Module.

Import Loads a previously saved *.euc file for use on the analyzer with the specified ECal module.

Delete Removes a User Characterization from 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 disk memory, 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 disk memory, 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.support.keysight.com/pna/apps/applications.htm>

How to Restore ECal Module Memory

Using front-panel HARDKEY [softkey] buttons

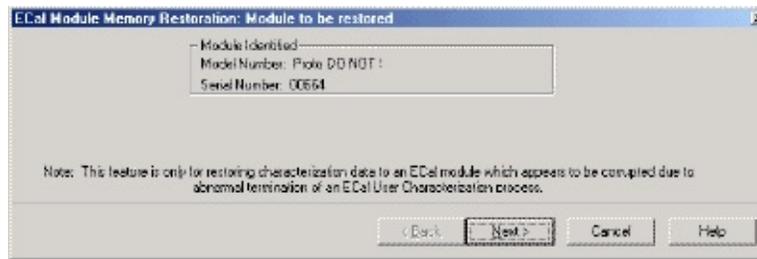
1. Press **SYSTEM**
2. then **[Service]**
3. then **[Utilities]**
4. then **[Restore...]**

Using Menus

1. Click **Utility**
2. then **System**
3. then **Service**
4. then **Utilities**
5. then **Restore...**

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, choose the one to have data restored.

Next Click to write data to the module.

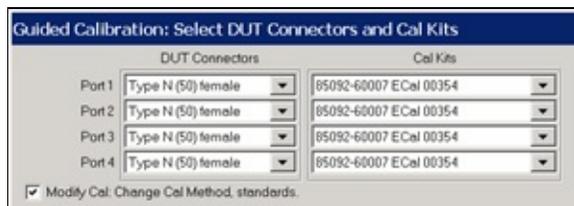
Perform a 4-Port Cal with One 2-Port ECal Module

You can perform a 4-port calibration with a 2-Port ECal Module. When all four DUT connectors are the same type and gender, the calibration can occur with only four connections, the same number of connections you would make with a 4-port ECal module.

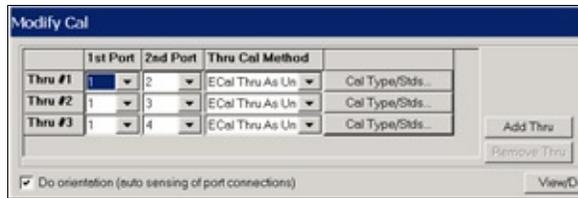
- The ECal module must span the frequency range of the measurement.
- The ECal module must have connectors that match the DUT connectors. Because we are using a 2-port ECal module, this means that the DUT must have only TWO unique connector types and gender. When the DUT has more than two connector types/genders, you can select a different cal kit for each port using SmartCal.

Procedure

1. Connect the 2-port ECal module to a PNA USB port.
2. Press **Cal**, then **Start Cal**, then **Cal Wizard**
3. Select either **SmartCal** or **Use Electronic Calibration** then click **Next**.
4. Select **4 Port Cal**, then click **Next** to see the following dialog:



5. Select the DUT Connectors for each port. In this example, all four DUT connectors are Type N, female.
6. Select the attached ECal module. We are using a **85092-60007 ECal** module.
7. Select **Modify Cal (Show Advanced Settings for ECal)** then click **Next** to see the following dialog:



1. Do orientation (auto sensing of port connections)
8. For the fewest number of physical connections, select the default port assignments.
 - The **1st Port** selection for each port pair is 1.
 - For single-ended (standard) measurements, THREE is the minimum number of Thru connections. For Balanced measurements, FOUR Thru connections should be made. [Learn more.](#)
 - For higher accuracy, select **Add Thru**. The Cal Wizard will add another port pair which results in more physical connections.
9. Select **ECal Thru as Unknown**. This is the most accurate and easiest Thru Cal Method. [Learn more.](#)
10. You may need to clear **Do Orientation** when calibrating at low power levels. [Learn more.](#) This will add additional connection steps.
11. Follow the prompts to complete the calibration:
 1. Connect ECal to ports 1 and 2. Click **Measure**.
 2. Connect ECal to ports 1 and 3. Click **Measure**.
 3. Connect ECal to ports 1 and 4. Click **Measure**.
12. At the **Specify Delay** dialogs, click **OK**. This is the measured delay for each of the Thru connections in the ECal module. [Learn more.](#)
13. Click either **Save As User Cal Set**, or **Finish**.

Last modified:

2-Aug-2011 New topic

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 on 4-port PNA-L and ALL Models with an External Test Set](#)

[See other Calibration Topics](#)

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

Keysight 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. 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 $f_1 \times f_2$).

Note: The TRL LINE standard must have a delay that is greater than 0 (zero) ps. Otherwise, calibration correction calculations will contain unpredictable results.

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 Keysight Vector Network Analyzers \(Application Note 1287-11\)](#)

TRL on a 4-port PNA-L and ALL PNA Models with an External Test Set

Beginning with the PNA code revision 5.25, TRL CAN be performed on a 4-port PNA-L and ALL PNA Models 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.

Note: See Delta Match Calibration to learn which models require this.

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 in these cases

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:

13-Dec-2012	Fixed models for GDM cal.
3-Aug-2012	added line std note
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

CalPod

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](#)

See Also

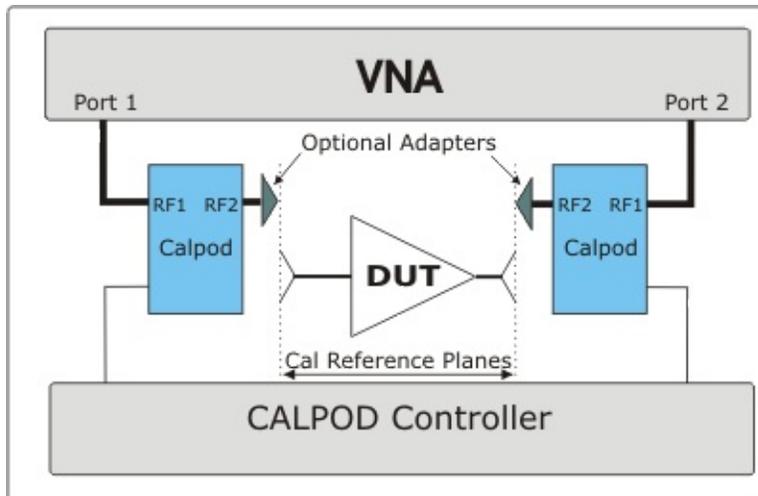
CalPod as ECal

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://literature.cdn.keysight.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:



The **Blue** boxes represent CalPod modules with internal Thru, Short, Open, and Load states.

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 recorection:
 - 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 recorection 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

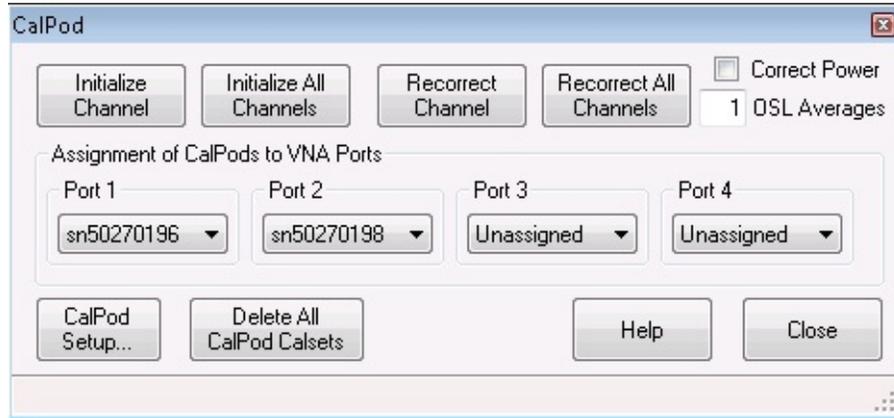
1. Press **CAL**
2. then **[More]**
3. then **[CalPod]**
4. then **[CalPod...]**

Using 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

This checkbox causes power to be recorrected ONLY when source power correction data is stored as error terms in the CalSet. This occurs only when a Guided Power Cal is performed and when an app channel is calibrated such as a FCA, GCA, IMD, and Noise Figure channel. This checkbox has NO effect when a S-parameter Cal or a standard Source Power Cal has been performed, because source power correction data is not stored in the CalSet.

When any of the above power calcs have been performed, and when this box is 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 re correction. Do this when you add a significant amount of loss in the calibration path, or when the power level at the DUT is important.

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.

OSL Averages Controls the number of sweeps worth of raw measurements to be measured and averaged together for the recorection computations for each state of each CalPod.

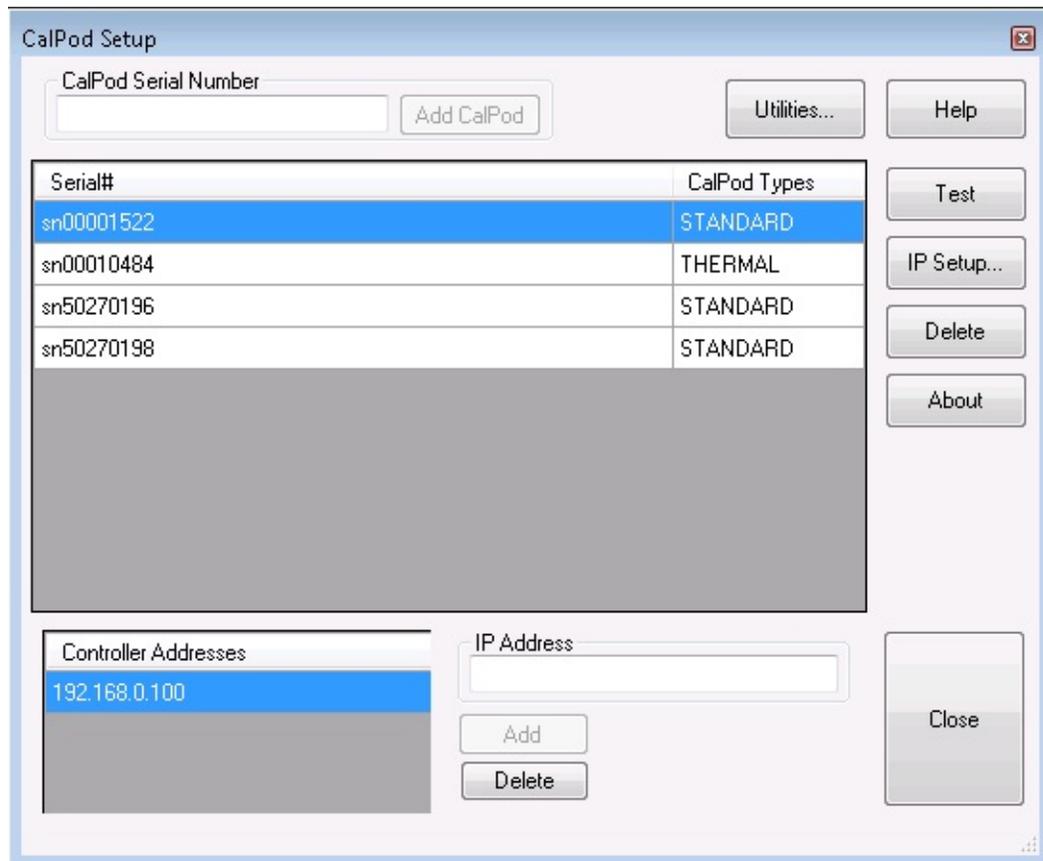
CalPod Assignments

For each PNA port, select a CalPod module.

CalPod Setup Starts the **CalPod Setup** dialog

Delete All CalPod Cal Sets Deletes all recorection 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.)

Serial # and CalPod Types

Shows the list of available CalPod modules. A CalPod module type may be STANDARD or THERMAL.

Buttons

Utilities Launches the VNA CalPod Utilities used to configure the CalPod Controller and VNA over LAN.

Note: Before using a CalPod Controller, the LAN MUST be set up using the CalPod Utilities or an error message will be displayed indicating that the VNA is unable to communicate with the CalPod Controller.

Keysight Technologies
PXI Vector Network Analyzer
CalPod Utilities

Agilent/Keysight M9036A/M9037A PXI Controller Network Setup.

- M9036A/M9037A PXI Controller Local Area Connection LAN to Static**
Use this option to set the Local Area Connection adapter to a Static IP address of 192.168.0.101
- M9036A/M9037A PXI Controller Local Area Connection 2 LAN to Static**
Use this option to set the Local Area Connection2 adapter to a Static IP address of 192.168.0.102
- M9036A/M9037A PXI Controller Local Area Connection LAN to Obtain an IP address automatically**
Use this option to set the Local Area Connection adapter to Obtain an IP address automatically
- M9036A/M9037A PXI Controller Local Area Connection 2 LAN to Obtain an IP address automatically**
Use this option to set the Local Area Connection 2 adapter to Obtain an IP address automatically

CalPod Literature

- User's Guide
- Calpod Controller Performance Characteristics
- Calibration Refresh Modules Flyer
- Network Setup for Non Agilent/Keysight PXI Controllers

KEYSIGHT TECHNOLOGIES

Exit this program PDF Reader required for CalPod Literature

M9036A/M9037A PXI Controller Local Area Connection LAN to Static Sets the Local Area Connection adapter to a static IP address of 192.168.0.101.

M9036A/M9037A PXI Controller Local Area Connection 2 LAN to Static Sets the Local Area Connection 2 adapter to a static IP address of 192.168.0.102.

M9036A/M9037A PXI Controller Local Area Connection LAN to Obtain an IP address automatically Sets the Local Area Connection adapter to obtain an IP address automatically.

M9036A/M9037A PXI Controller Local Area Connection 2 LAN to Obtain an IP address automatically Sets the Local Area Connection 2 adapter to obtain

an IP address automatically.

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 recorection.

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.support.keysight.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.

See the **CalPod User's Guide** for instructions at:

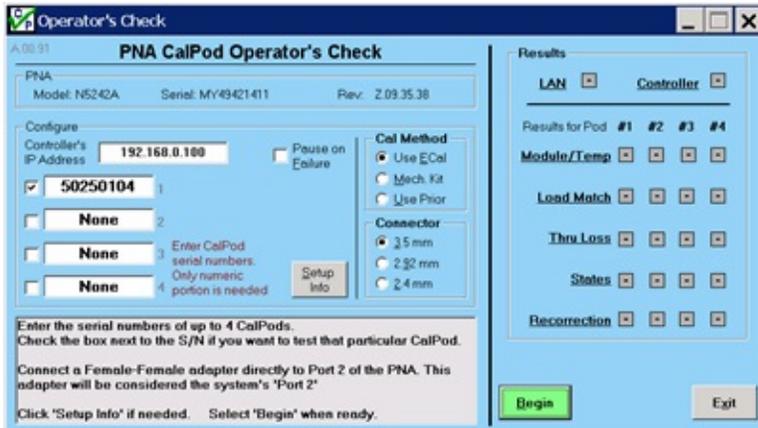
<http://literature.cdn.Keysight.com/litweb/pdf/85523-90001.pdf>

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**.



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/Keysight/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.support.keysight.com/pna/calpod>

Last Modified:

24-Apr-2013	Updated with SPC info (BH)
29-Mar-2013	Added link to Calpod as ECal
15-Jun-2011	Updated per LH
5-May-2011	New topic

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.

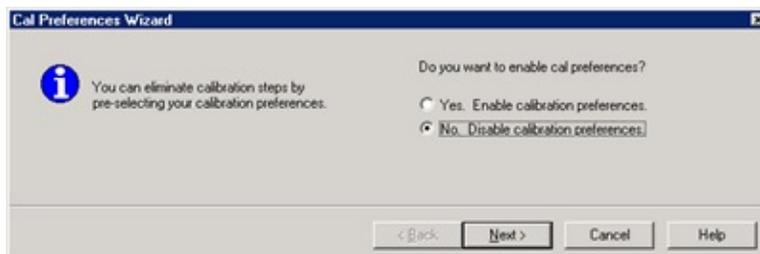
How to change Cal Preferences

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	Using Menus
<ol style="list-style-type: none">1. Press CAL2. then [Start Cal]3. then [Preferences]	<ol style="list-style-type: none">1. Click Response2. then Cal3. then Start Cal4. then Preferences

Programming Commands

Cal Preferences Wizard dialog box help

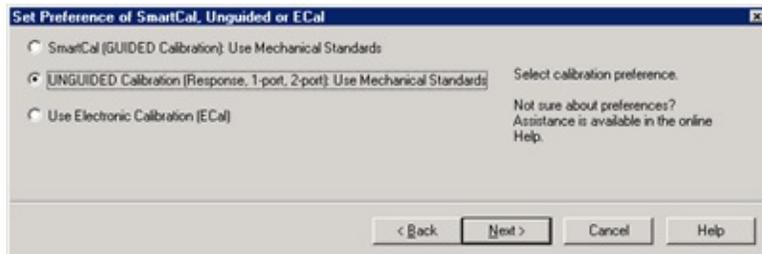


Use this dialog to change either of the following preferences:

- 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



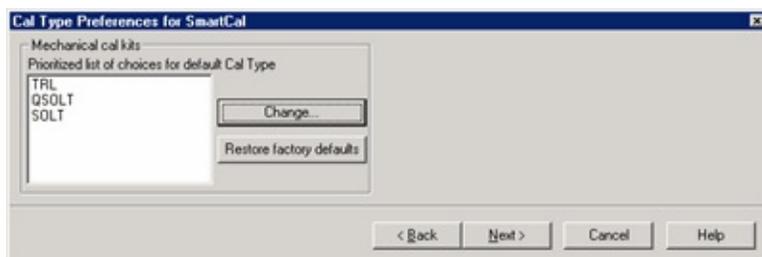
Use this dialog to change which Cal method to perform.

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.

Note: Your Cal Type settings are saved only until the NA application is closed. When

re-opened, the factory default settings are restored.

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.

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.

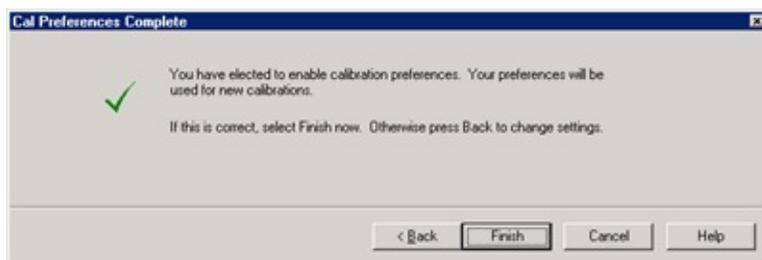
Unselected Cal Types Cal Types in this list will not be presented as a choice during a Calibration.

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.

Cal Plane Manager (CPM)

Adapters, fixtures, and probes are often used for DUTs that have non-coaxial interfaces. This could make it difficult to calibrate with traditional cal standards. Cal Plane Manager (CPM) allows you to mathematically remove (de-embed), a characterized adapter, test fixture, or probe head from measurements.

CPM is an enhancement of the existing Characterize Adaptor Macro.

In this topic:

- **Features**
- **Using Cal Plan Manager**
 - **Cal Plane Manager**
 - **Characterize Adapter/Fixture and Apply**
 - **Calset Selection**
 - **Port Selection**
 - **Phase Pivot**
 - **Select Files**
 - **Apply De-embedding**
 - **Select Channels to De-embed**
 - **Select Calsets to De-embed**
- **Other Actions**
 - **Reverse Port Order**
 - **Create a Transmission Only S2P File**
 - **Cascade Two S2P Files**

Other Cal Topics

Features

- Characterizes adapters and fixtures in SnP files.
- Applies the characterizations to existing Cal Sets and channels.

- Writes to PNA power loss table using the S2P files of fixtures/adapters.
- Reverses the port order of an existing S2P file.
- Creates a forward-only S2P file from an existing S2P file.
- Cascades two S2P files.

Important Notes

- **Adapter/fixture** definition: Any physical 2-port device or component that is to be mathematically removed from channel measurements or Cal Sets.
- The adapter/fixture to be characterized **MUST** be reciprocal ($S_{21} = S_{12}$).
- Two Tier-1 cals must be performed and saved to Cal Sets **BEFORE** performing the CPM characterization.

Using Cal Plane Manager

How to start Cal Plane Manager

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

1. Press **CAL**
2. then **[Fixtures]**
3. then **[Cal Plane Manager]**

Using Menus

1. Click **Response**
2. then **Cal**
3. then **[Fixtures]**
4. then **[Cal Plane Manager]**

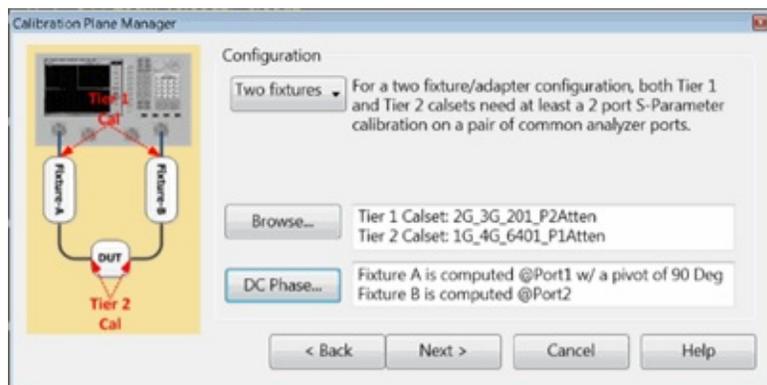
Cal Plane Manager dialog box help

Programming Commands

Choose from the following, then click **Next** >:

- **Characterize Adapter/Fixture and Apply** - Given that you have already performed calibrations both before and after the adapter/fixture, SNP files are generated which characterize the adapter/fixture. These files are then used to de-embed the fixture from the channel and a new calset. Learn how.
- **Apply Adapter/Fixture** - The *.SNP files are already saved. Use these files to de-embed the fixture from the channel and a new Cal Set.
- **Other Actions**

Characterize Adapter/Fixture and Apply dialog box help



Requirements

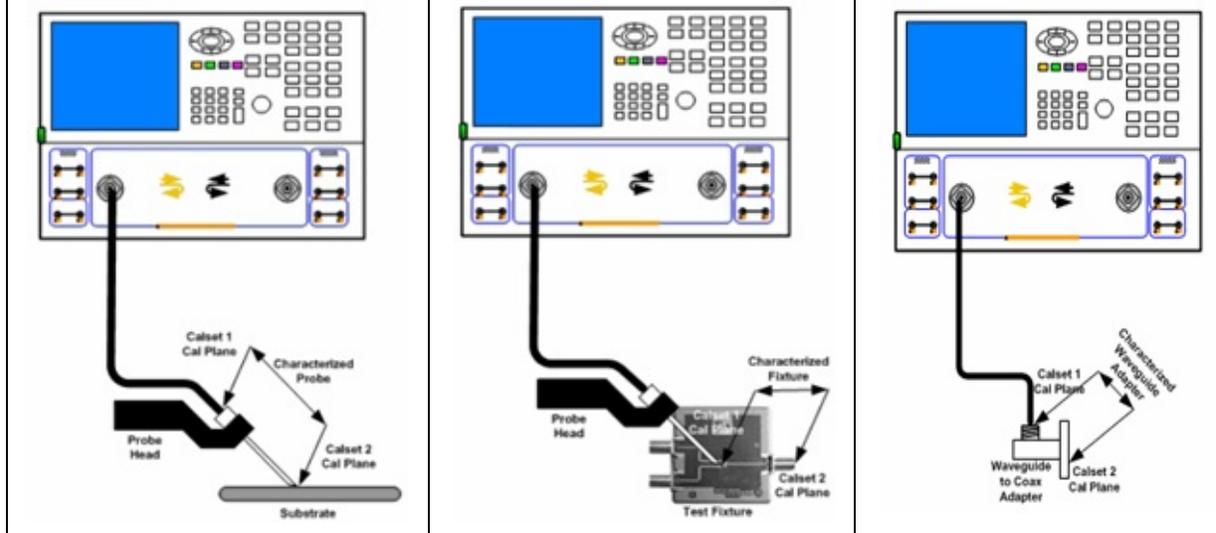
- You must have already performed a Tier 1 calibration at the input of the fixture/adapter(s) AND a Tier 2 calibration at the output of the fixture/adapter(s) as in the above image.
- In addition, the calibrations must have been saved to Cal Sets on the analyzer.

Note: The mechanical switch / attenuator settings of the Tier 1 and Tier 2 cals for CPM MUST be the same settings. Also, when the span or number of points are different between the two cal tiers, there must be sufficient data points to ensure that phase wrapping does NOT occur. This is accomplished when the delta frequency for either calset is less than $12/\text{combined length of the test port cables in meters}$.

Applications

The following images show the calibration planes of the Tier 1 and Tier 2 calibrations:

Probe Head on Wafer	Probe Head on Fixture	Waveguide Adapter



Procedure

Configuration Select the number of adapter/fixtures to be characterized and de-embedded.

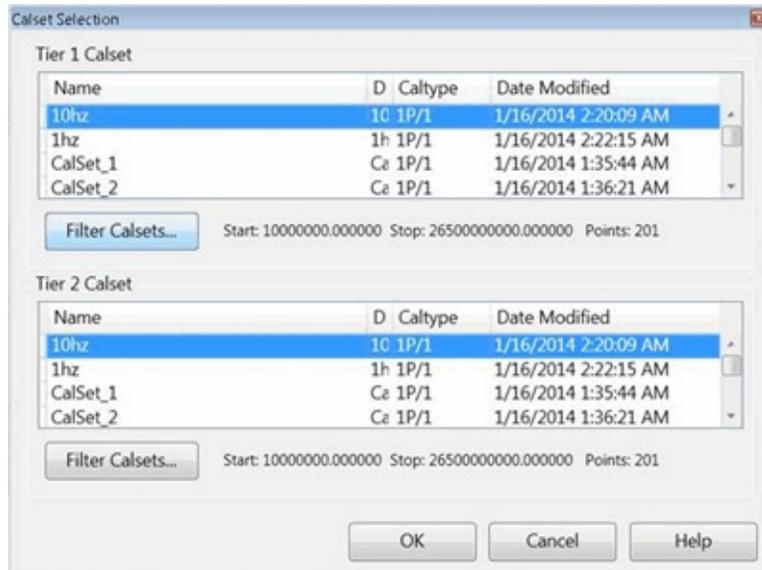
- Choose **One fixture** when you have a single fixture/adapter on either the input or output of the DUT.
- Choose **Two fixtures** when you have a fixture/adapter on BOTH the input AND output of the DUT.

Browse - Starts the following **Calset Selection** dialog.

DC Phase - Starts the **Phase Pivot** dialog.

Click **Next** >

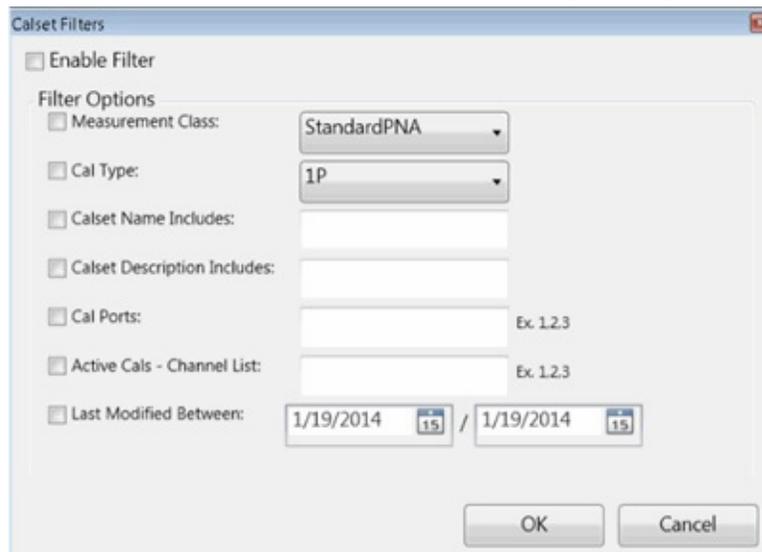
Calset Selection dialog box help



Choose from the listed Cal Sets on the analyzer to use for the Tier 1 and Tier 2 calibrations.

- The Tier-1 calset **MUST** be from a calibration that was performed at the input to the adapter/fixture.
- The Tier-2 calset **MUST** be from a calibration that was performed at the DUT reference plane.

Click **Filter Calsets** to start the following dialog.



Check **Enable Filter**, then provide advanced filter requirements to narrow the search for appropriate Cal Sets.

Filter Options

Measurement Class - The classes listed are those that are enabled on the analyzer.

Cal Type - Filter for 1P (one-port) or 2P (two-port) Cal Sets.

Calset Name Includes - Filter to include any text that appears in the calset name.

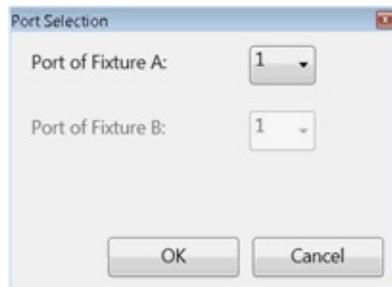
Calset Description Includes - Filter to include any text that appears in the calset description.

Cal Ports - Filter to include only the analyzer ports to be de-embedded.

Active Cals - Filter to include only the Cal Sets that are currently in use on the analyzer.

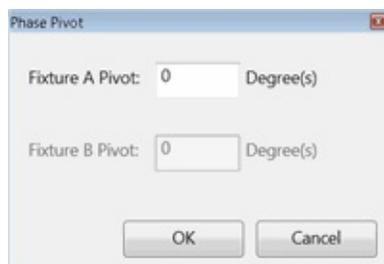
Last Modified Between - Filter to include only the Cal Sets that were last modified between the two specified dates.

Port Selection dialog box help



In the previous dialog, when a multiport calset is selected for a characterization that involves fewer ports, then select the port in the calset that is used to characterize the fixture/adaptor.

Phase Pivot dialog box help



For most devices, the projected phase of S21 at DC crosses the X-axis between 0° and -180°.

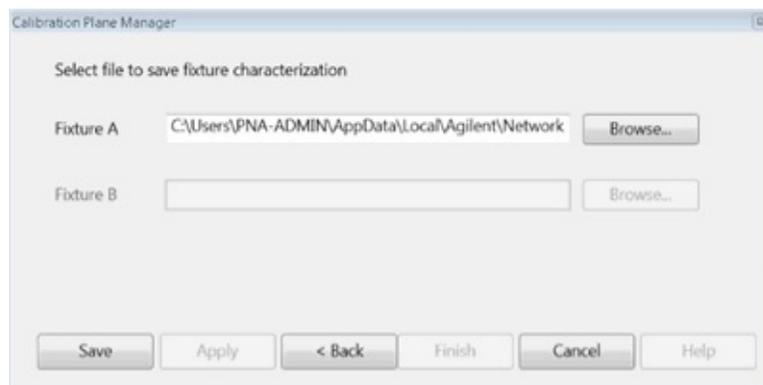
The phase pivot point specifies the center of the phase window. It is normally 1 Pi wide. The default value of 0° should be adequate for the majority of adaptors.

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 CPM versus the results you might get by measuring the same adapter with a 2-port calibration.

In these cases, you may have to change the default value to capture the projected phase of S21 at DC.

Select Files dialog box help



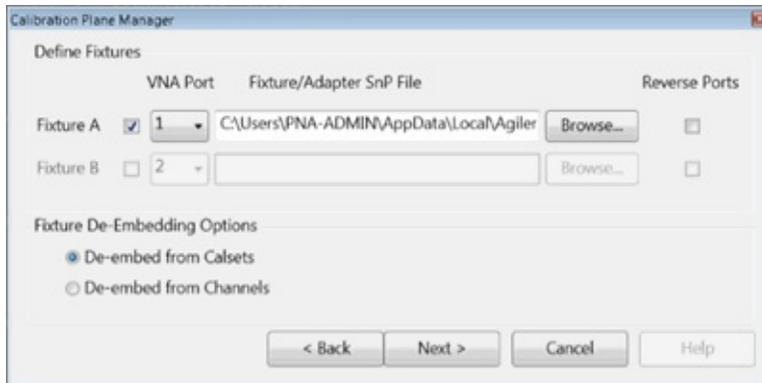
For each Fixture (A and B):

- Click **Browse**, then navigate to the folder to where the S2P files are to be saved.
- Enter a filename, then click **S2P Format**, then choose the format in which the data is to be saved:
 - Log Magnitude & Angle (default)
 - Lin Magnitude & Angle
 - Real & Imaginary

Click **Apply** to continue to de-embed the fixture.

Click **Finish** to end with the characterization and close the dialog.

Apply De-embedding dialog box help

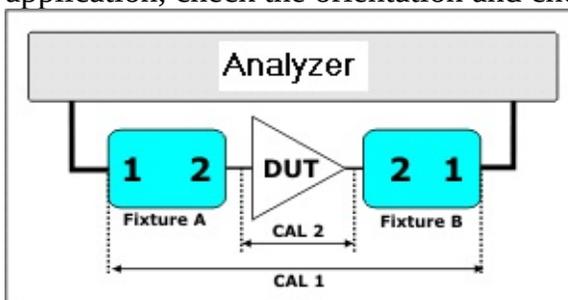


Given one S2P file for each fixture/adaptor, this dialog will remove the effects of the fixture/adaptor from either:

- one or more Calsets
- or one or more channels.

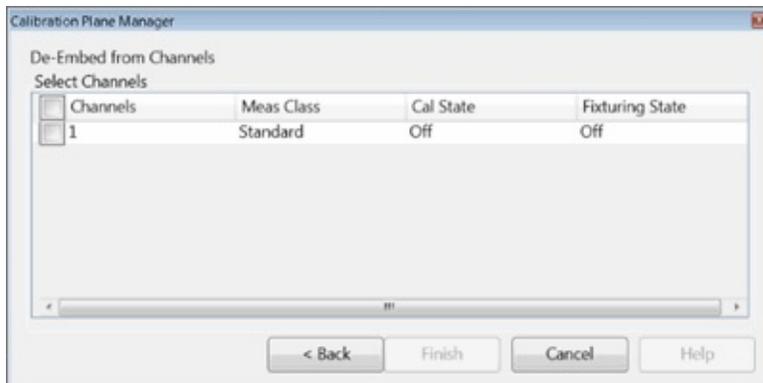
For each Fixture (A and B)

1. Check to enable fixturing.
2. Select the VNA port the fixture is connected to.
3. Click **Browse**, then navigate to the S2P file that represents the fixture/adaptor.
4. **Reverse Ports** - S2P files that are created using CPM ALWAYS reference port 1 of the fixture/adaptor on the side closest to the analyzer and port 2 of the fixture/adaptor ALWAYS on the DUT side of the device as in the following image. The application of the S2P file (this dialog) assumes this same orientation. If your S2P files were created using a different (external) application, check the orientation and check **Reverse Port Order** if necessary.



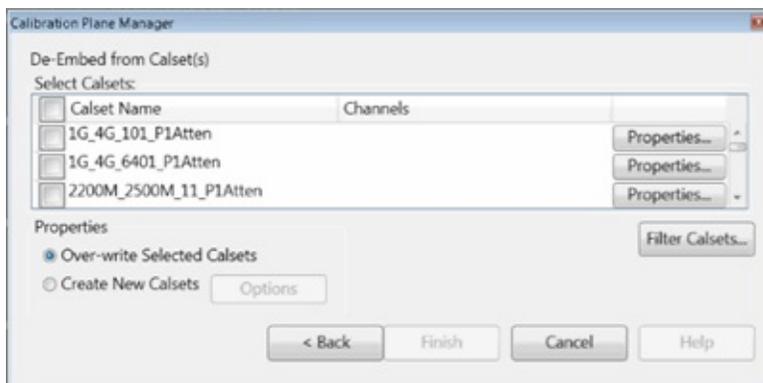
5. Choose one of the following De-embed Options
 - **De-embed from Calsets** - Starts the **Select Calsets to De-embed** dialog box.
 - **De-embed from Channels** - Starts the following **Select Channels to De-embed** dialog.

Select Channels to De-embed dialog box help



De-embedding is performed and applied to specified channels on the analyzer. Select one more channels currently displayed on the analyzer from which to de-embed the adapter/fixture.

Select Calsets to De-embed dialog box help



This dialog appears when **De-embed from Calsets** is selected in the previous dialog. De-embedding is performed and applied to specified Cal Sets. This allows you to easily apply de-embedding in the future by simply applying the de-embedded calset to any channel.

Select Calsets: Select the Cal Sets to which de-embedding will be applied.

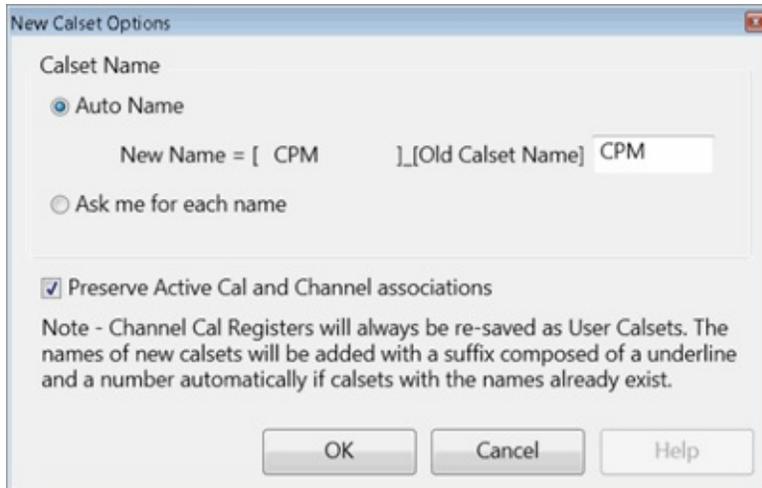
Properties View information about the corresponding calset.

Properties

- **Overwrite Selected Calsets** - The selected Cal Sets are overwritten with the adapter/fixture de-embedded.

- **Create New Calsets** - Select the Cal Sets from which new Cal Sets will be created.

Click **Options** to start the following dialog.

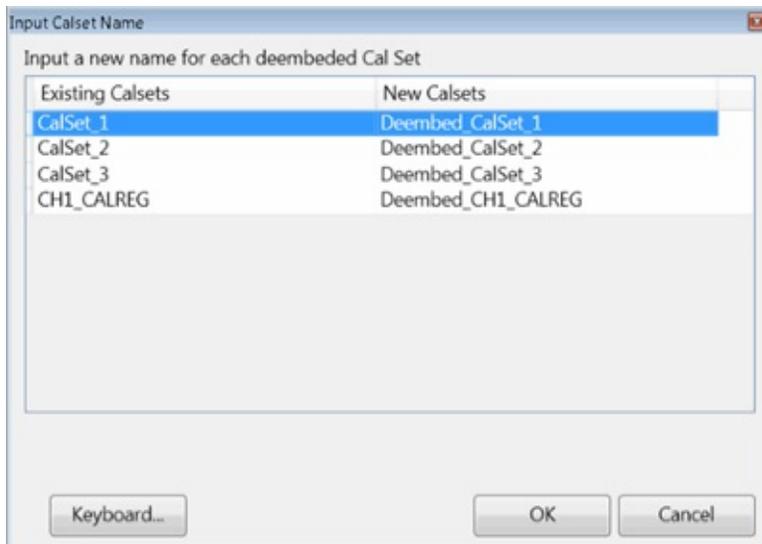


Calset Name

Auto Name - By default, a new calset will be created using the old calset name with the specified text ("CPM" by default) appended to the beginning of the name. You can change the specified text.

Ask me for each name - Starts the following dialog when **OK** is pressed.

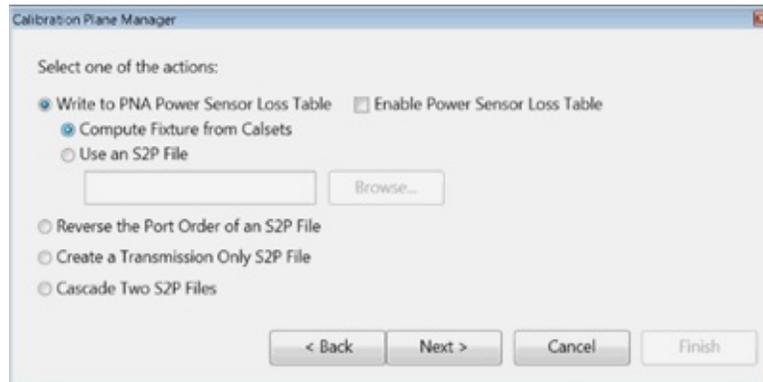
Preserve Active Cal and Channel associations - When checked (default) the new de-embedded Cal Sets will be used to correct the same displayed channels as the current Cal Sets.



The Existing Calsets that you selected for de-embedding appear in the left column. The proposed New Calset names appear in the right column.

To change the new Calset name, select, then edit the name.
When finished, click **OK**.

Other Actions dialog box help



Select one of the actions:

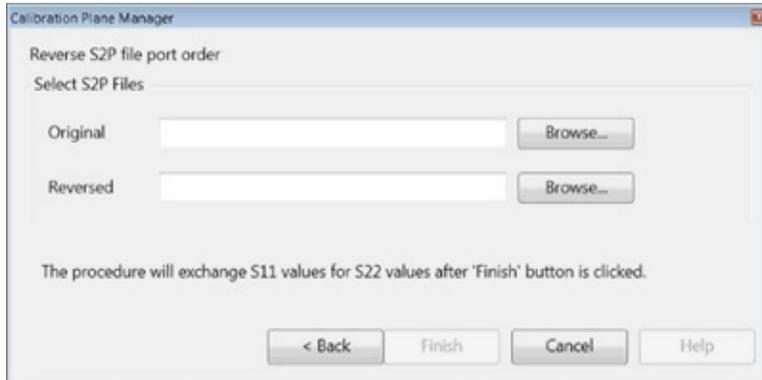
- **Reverse the Port Order of an S2P File.**
- **Create a Transmission Only S2P File.**
- **Cascade two S2P files.**

Write to PNA power sensor loss table. 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.

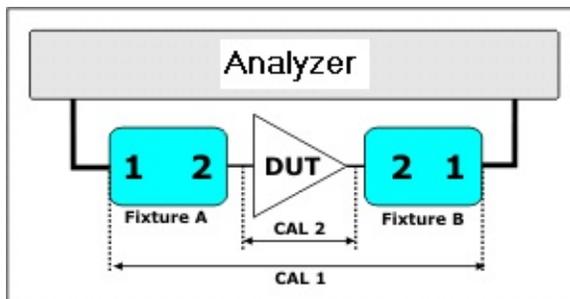
- **Enable Power Sensor Loss Table**
- Then choose from the following:
 - **Compute fixture from Calsets.** Computes the transmission loss of the fixture based on the selected Cal Sets. This choice is NOT available until two valid Cal Sets are selected.
 - **Use an S2P file.** Uses the S21 data in an existing S2P file to build the PNA's power loss table. Select, then click **Browse**, then navigate to the S2P file, then click **Next >**.

Note: In the PNA Power Loss Compensation table, loss is expressed as a positive number. CPM 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.

Reverse Port Order dialog box help



S2P files that are created using CPM ALWAYS reference port 1 of the fixture/adaptor on the side closest to the analyzer and port 2 of the fixture/adaptor ALWAYS on the DUT side of the device as in the following image.



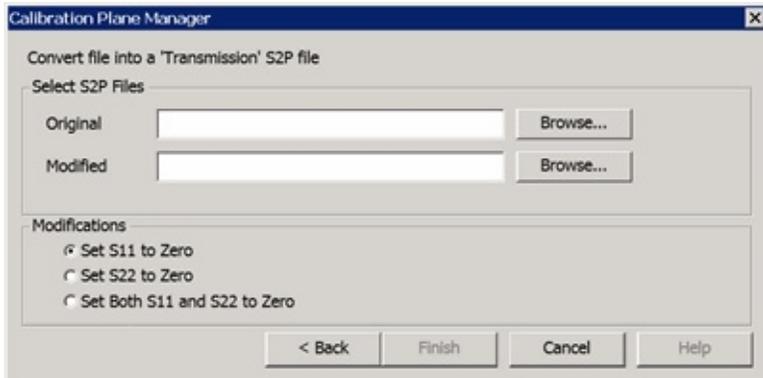
This action causes ports to be reversed on an existing S2P file.

- 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 **S2P file format**.

1. **Original** - Navigate to the S2P file to be reversed.
2. **Reversed** - Navigate to the folder where the new reversed S2P file will be saved. Enter a filename. By default, the file is saved to the same folder using the filename: <old filename>_Reversed.s2p
3. Click **Finish**. The Reversed file is saved to the specified location.

Create a Transmission Only S2P File dialog box help



From an existing S2P file, this feature allows you to zero the S11, S22, or both data columns. The original S21 and S12 data are preserved. This is useful for Enhanced Response calibration / de-embedding.

Original - Click **Browse**, then navigate to the file to be modified.

Modified - Click **Browse**, then navigate to the folder and enter or change the filename of the resulting S2P file. The file select dialog allows you to change the format of the data. Click **Format**, then choose from the following:

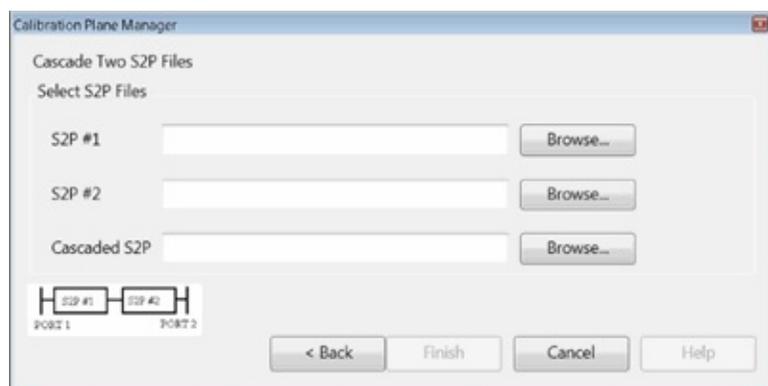
- Log Magnitude & Angle (default)
- Lin Magnitude & Angle
- Real & Imaginary

Modifications

Choose to Zero the S11, S22, or both data columns.

Click **Finish**. The transmission only file is saved to the specified location.

Cascade Two S2P Files dialog box help



This dialog combines the losses and phase shift of two S2P files into a single S2P file.

The stimulus settings of the two input S2P files need not be identical. The frequency range of the cascaded S2P file will be the frequency range that is common between the two input files. In addition, the cascaded S2P file will use the data points of the input file with the denser data points.

For example:

S2P #1: Frequency range = 1 GHz to 5 GHz; 201 pts.

S2P #2: Frequency range = 2 GHz to 6 GHz; 1001 pts.

Cascaded S2P: Frequency range = 2 GHz to 5 GHz using the data points of S2P #2.

S2P #1 - Click **Browse**, then navigate to one of the S2P files to be cascaded.

S2P #2 - Click **Browse**, then navigate to the other S2P file to be cascaded.

Cascaded S2P - Click **Browse**, then navigate to a folder and enter the filename of the resulting S2P file.

Click **Finish**. The cascaded file is saved to the specified location.

Fixture Simulator

The following features allow you to mathematically add (embed) or remove (de-embed) circuits to, or from, your measurements. The mathematical models are applied to specific ports for all measurements on the channel.

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

Click to learn more about each operation.

First, the following **Single-ended** measurement functions are processed in this order:

1. [Port Extensions](#)
2. [Ground loop de-embedding / embedding](#)
3. [2-Port De-embedding](#)
4. [Port Matching Circuit Embedding](#)
5. [Port Z \(Impedance\) Conversion](#)
6. [4-Port Network \(single-ended\) Embed/De-embed](#)

Then, **Balanced** measurement functions are processed in this order:

7. [Balanced Conversion](#)
8. [Differential / Common Mode Port Z Conversion](#)
9. [Differential Port Matching](#)

- **Source power compensation** is then optionally applied to compensate for the aggregate loss through all enabled fixturing operations.

Notes

- 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 **Data processing chain**, the Fixture Simulator functions occur at the same time as the **Apply Error Terms** block.
- When fixturing is enabled, all of the enabled fixturing features are applied when **snp files are saved**.

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**.

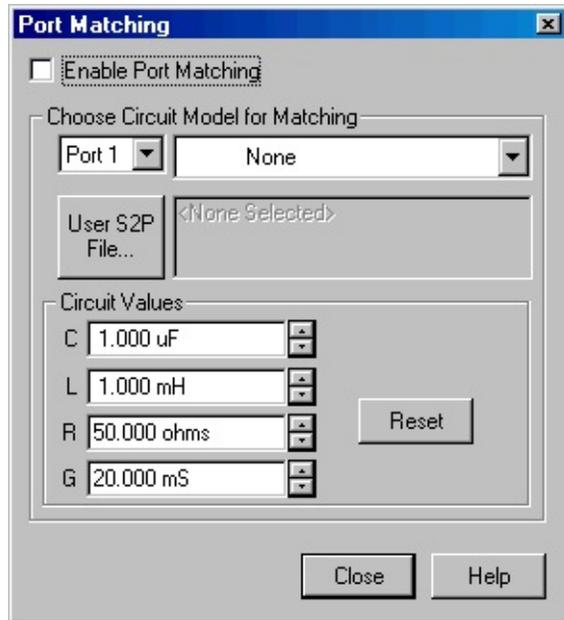
1. Check **Fixturing ON/off**
Port Extensions is NOT affected by Fixturing ON/off.
2. Check **Enable** on the individual fixturing selection dialog box.

Using Menus

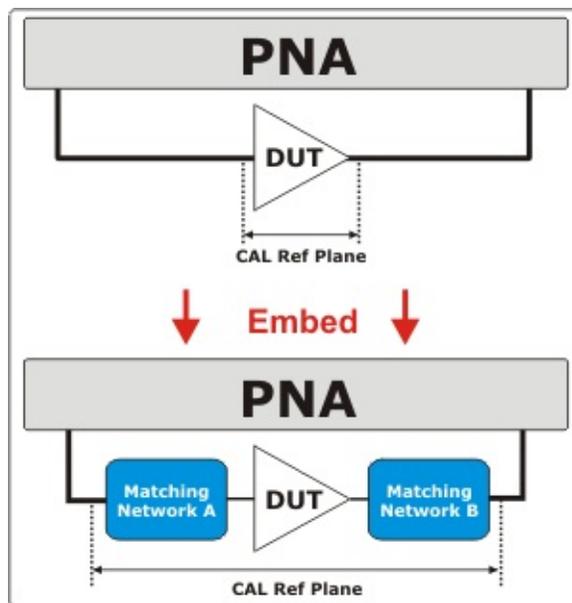
1. Click **Cal**
2. then **More**
3. then **Fixtures**

[Programming Commands](#)

[Port Matching dialog box help](#)



This function specifies a circuit to embed (add) to the measurement results. See [Order of Fixture Operations](#).

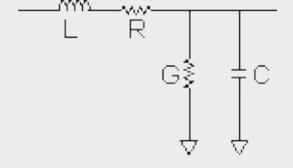
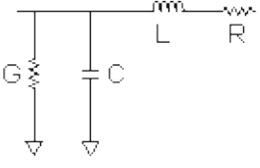
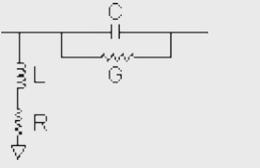
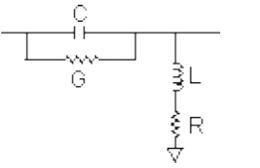
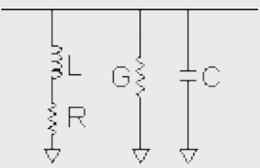


Enable Port Matching Check to apply the settings to the measurement results. Must also enable [Fixturing ON/off](#).

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:

Series L - Shunt C	PNA		DUT
---------------------------	-----	--	-----

			
Shunt C - Series L	PNA		DUT
Shunt L - Series C	PNA		DUT
Series C - Shunt L	PNA		DUT
Shunt L - Shunt C	PNA		DUT

User Defined (S2P File) Load a file that is specified with **User S2P File** button.

None Use no circuit model.

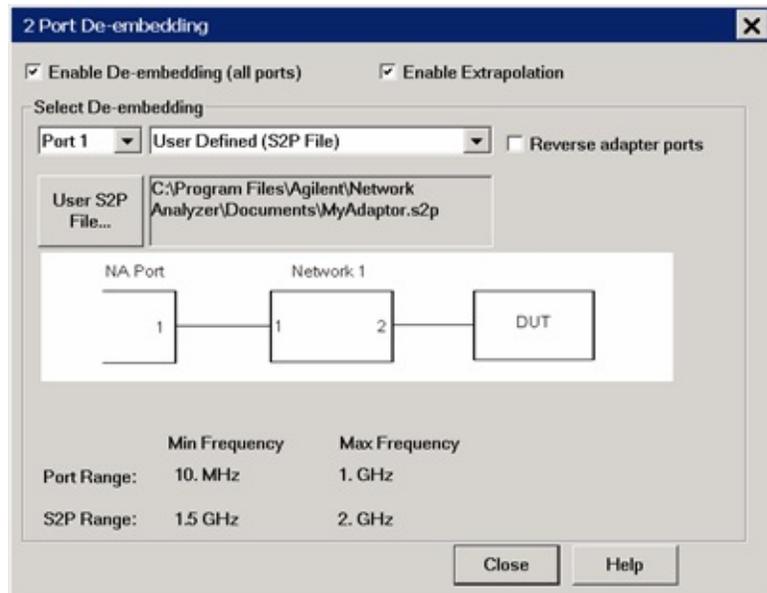
User S2P File Click to specify an S2P file of the circuit model to embed at the selected port. If the normalized impedance value in a recalled User .S2P file is different from the port reference impedance setting of the PNA, the PNA setting is used. Characterize Adaptor Macro can be used to create S2P 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.

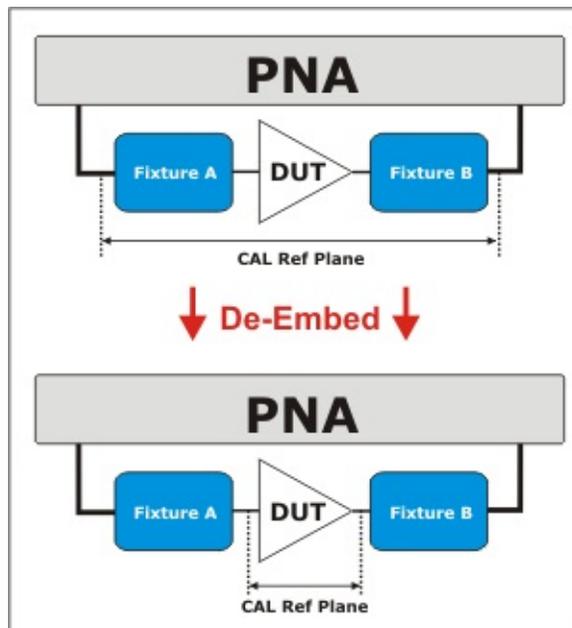
Reset Restores the default values.

2 Port De-embedding dialog box help



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 component or test fixture from the measurement results.

Note: De-embedding a component with more than 20 dB of loss becomes impractical because of an inability to accurately measure the match of the DUT through such a device.



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



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.

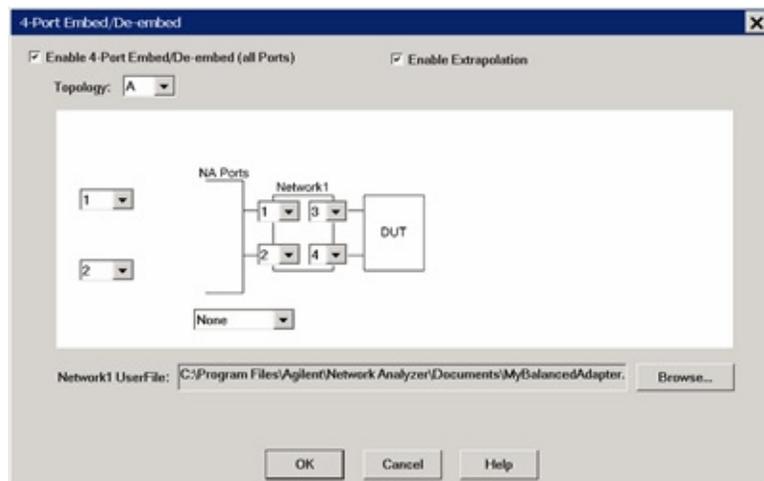
jX Imaginary part of the impedance value.

Close Applies the entries and closes the dialog box

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.

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.

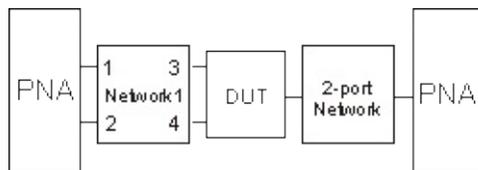
Topology

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

Note: Not all possible DUT topologies are addressed with this dialog box.

To embed or de-embed other topologies, use this dialog in conjunction with the **2-port embed/de-embed dialog**. For example, if you have a 3-port Bal-SE DUT and have networks to de-embed as shown here:



1. In this dialog specify **Topology B**.
2. De-embed the 4-port Network1 on the Balanced input.
3. Use the **2-port dialog** to de-embed the 2-port network on the Single-ended output.

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.

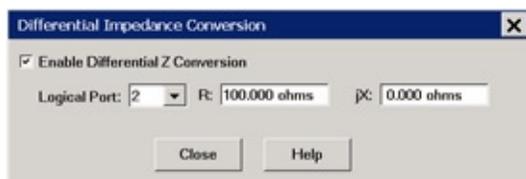
- **De-embed** - Remove the specified network circuit from the measurement results. See 2-port De-embed image.

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 Z0** 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.

$$(Z1 * Z2) / (Z1 + Z2)$$

Where ports 1 and 2 comprise the logical port:

Z1 = the Port Impedance values for port 1

Z2 = the Port Impedance values for port 2

If **Port Z Conversion** is not enabled, then **System Z0** 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](#).

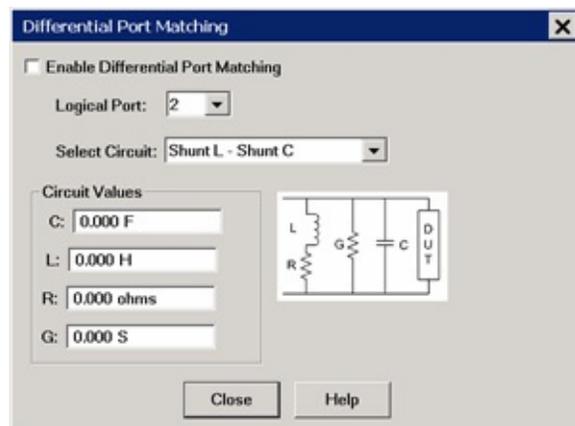
R Real part of the impedance value.

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.

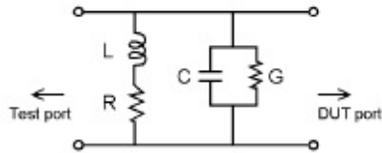
See [Order of Fixture Operations](#).

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.

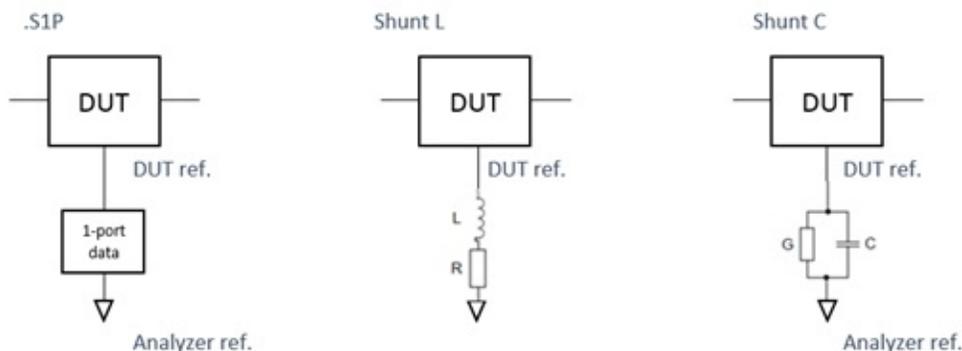
Note: Use caution when applying power compensation. Always test your setup without a DUT in a place. If you are using S2P files, **Recall** your S2P file into the PNA so you can verify that the device your S2P file describes is what you intended it to be. It is too easy to misalign data in S2P files if they are constructed by hand.

Ground Loop De-embedding / Embedding

Ground loop de-embedding removes the effect of a non-ideal ground connection between the DUT's ground and the analyzer's ground reference. Typically, the non-ideal component is the parasitic inductance of the ground contacts.

Ground loop embedding adds the effect of a non-ideal component on the ground contacts.

The Ground Loop De-embedding / Embedding can be specified by circuit model type or touchstone file.

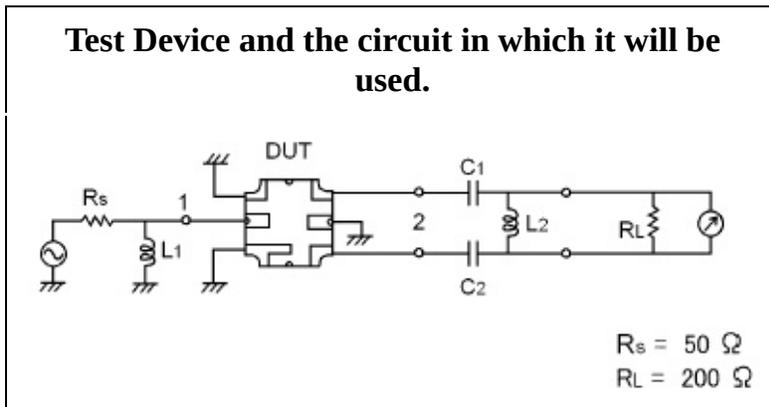


Ground loop de-embedding / embedding is only available from **SCPI remote interface**.

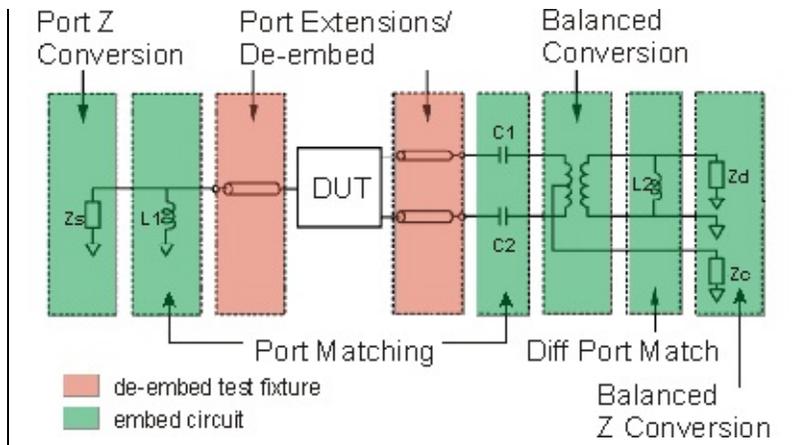
Fixture Simulator Example

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.



Circuit Simulation



This diagram does NOT refer to the order in which operations are performed.

See [Order of Fixture Operations](#).

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 R_s 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**.
-

Port Extensions

Port extensions allow you to electrically move the measurement reference plane after you have performed a calibration.

- [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

[Data Flow Map](#)

[Fixture Compensation features](#)

Other Calibration Topics

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 set Port Extensions

Using front-panel HARDKEY [softkey] buttons

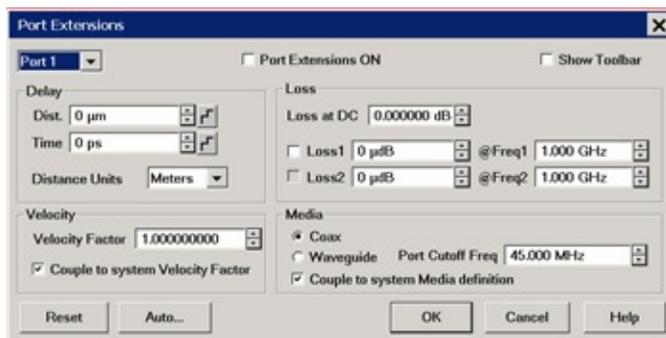
1. Press **CAL**
2. then **[Port Extensions]**

Using Menus

1. Click **Response**
2. then **Cal**
3. then **Port Extension**

◀ Programming Commands ▶

Port Extensions dialog and Toolbar help



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 Select a port for delay and loss values. Port Extensions settings affect ALL measurements on the active channel that are associated with a particular port.

Port Extension Turns ON and OFF port extensions on all ports.

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 screen. This is the only way to show or hide the toolbar.

Delay

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

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:

$$\text{Loss}(f) = \text{Loss1} * (f/\text{Freq1}) ^ 0.5$$

Loss1 and Loss2:

Set the lower frequency to Loss1, and the higher frequency to Loss2.

$$\text{Loss}(f) = \text{Loss1} * (f/\text{Freq1}) ^ n$$

Where:

$$n = \log_{10} [\text{abs}(\text{Loss1}/\text{Loss2})] / \log_{10} (\text{Freq1}/\text{Freq2})$$

Note: abs = 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 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.

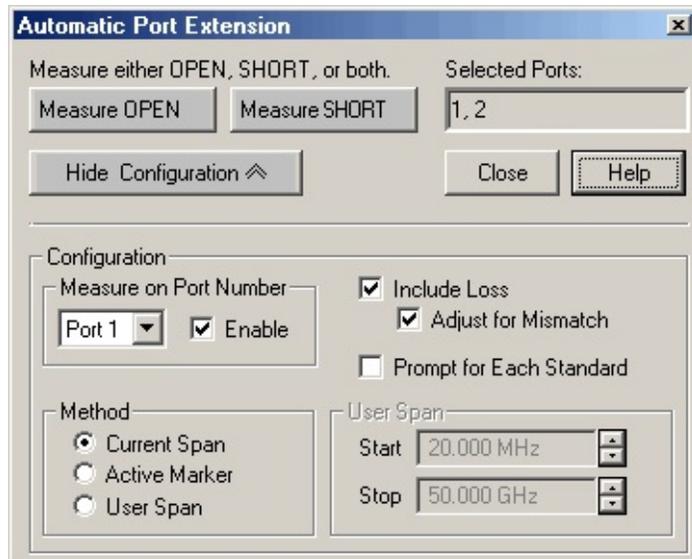
Auto Step Size is set to the default value.

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 analyzer 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. Mismatch adds ripple to the S11 and S22 traces. If the ripple is large, S11 and S22 can appear greater than 0 dB which leads to numeric instabilities in using the S-parameters. Adjust for mismatch increases the loss of the fixture so that the peak of the ripples is below 0 dB. While this adds more error (all the error is negative) it does allow the S-parameters to be used in simulators without numerical instabilities.

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 Delay Functions](#)

Swap Adapters Calibration Method

Although Swap Adapters calibration (also known as Swap Equal Adapters and Equal Length Adapters) method is NOT included in the PNA firmware, you can still perform this calibration.

Before we introduced the **Unknown Thru** method, the Swap Adapters method was often used as a quick alternative to the more tedious **adapter removal** method. In that case, you would be trading measurement accuracy for convenience. You might still want to perform Swap Adapters if you do NOT have calibration standards with the same connector type as your DUT. A procedure for this is shown below.

For any other reason, the Swap Adapters method is NOT recommended because the **Unknown Thru** method is more convenient AND more accurate.

There are many variations on the Swap Adapters calibration depending on the number of ports to be calibrated and whether the DUT is **insertable or non-insertable**. However, the concepts are the same: you perform connect and measure the reflection standards (OPEN, SHORT, and LOAD) with one adapter in place, then swap for a different adapter for the Thru measurement. The adapters must have the same delay, loss, and impedance. The better the adapters match, the better measurement results. Measure the adapters with a calibrated PNA to be sure.

Procedure

The following is an example procedure showing how to perform a Swap Adapters 2-port calibration for a non-insertable DUT. The DUT has 2.92 mm connectors. You do NOT have 2.92 mm calibration standards, but you DO have 2.4 mm standards and adapters that have the same electrical properties as the 2.92 mm adapters.

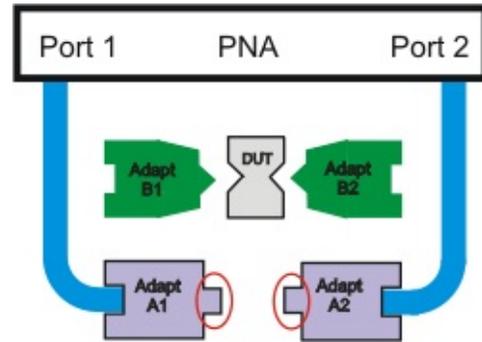
Adapters A1 and A2 = test port to 2.4 mm adapters

Adapters B1 and B2 = test port to 2.92 mm adapters

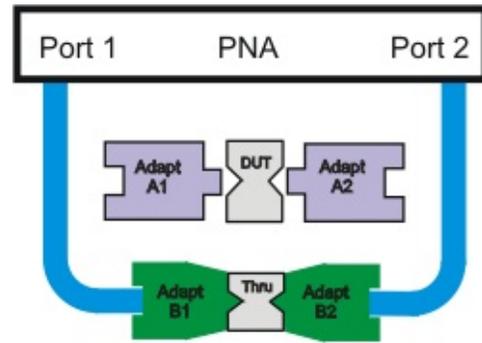
1. Start the Cal Wizard and select Guided (Smart) Cal. **Note:** The PNA will NOT prompt you to connect the adapters by name or when to swap the adapters.

2. Specify the connector type and gender and Cal Kit of the adapter that you will be using (2.4 mm) - NOT the connector type of the DUT (2.92 mm). By specifying the connector gender, you are also specifying the Thru method (flush thru for insertable and Unknown Thru for non-insertable.) For example, when both DUT ports have female connectors, we will perform an Unknown Thru cal.

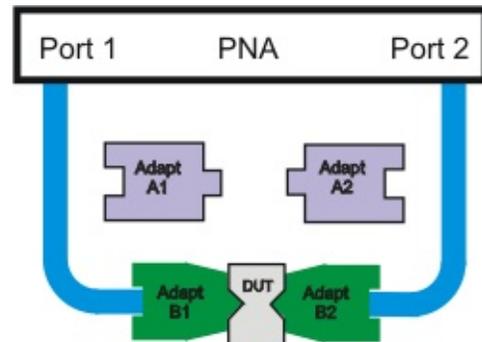
3. When prompted for reflection standards on port 1, connect the Open, Short, and Load standards to Adapter A1.
4. When prompted for reflection standards on port 2, connect the Open, Short, and Load standards to Adapter A2.



5. When prompted for a Thru connection, swap Adapter A1 and A2 for B1 and B2. Connect the Thru device. This could be any device that meets the requirements of the **Unknown Thru standard**. In the case of a non-insertable DUT, connect B1 and B2.



6. Make DUT measurements with Adapters B1 and B2 in place.



Last Modified:

8-Dec-2010 MX New topic

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?](#)

[See other Calibration Topics](#)

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 [Keysight Support Representative](#)

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

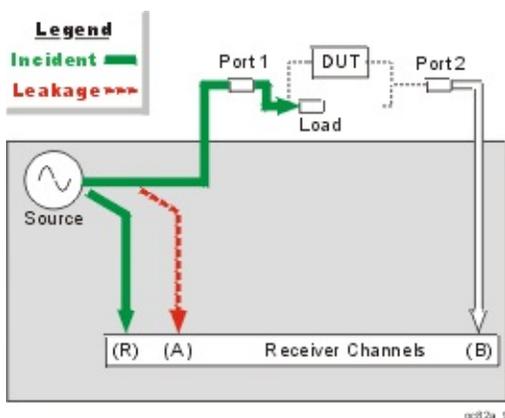
two port device.

- 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.

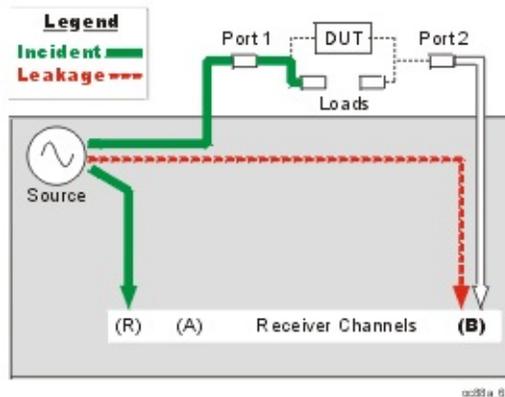
1. During calibration, a load standard is connected to Port 1. We assume no reflections from the **load**.
2. The signal measured at the 'A' receiver results from the incident signal leakage through the coupler and other paths.
3. Directivity error is mathematically removed from subsequent reflection 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

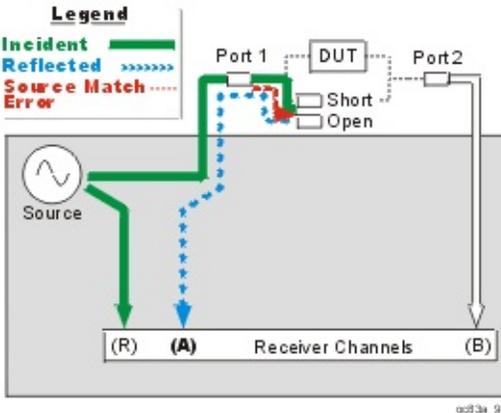
1. During calibration, load standards are connected to both Port 1 and Port 2.
2. The signal measured at the 'B' receiver is leakage through various paths in the analyzer.
3. This isolation error is mathematically removed from subsequent transmission measurements.

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

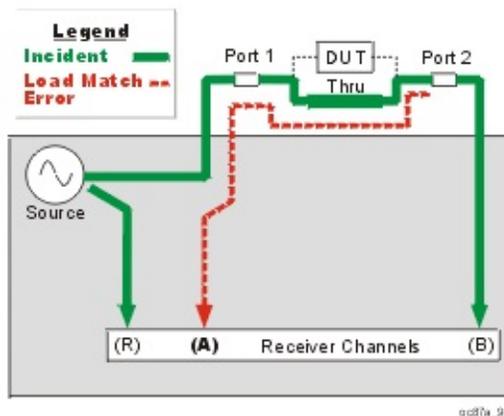
1. During calibration, all reflection standards are connected to Port 1. Known reflections from the standards are measured at the 'A' receiver.
2. Complex math is used to calculate source match error.
3. Source match error is mathematically removed from subsequent reflection and transmission measurements.

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

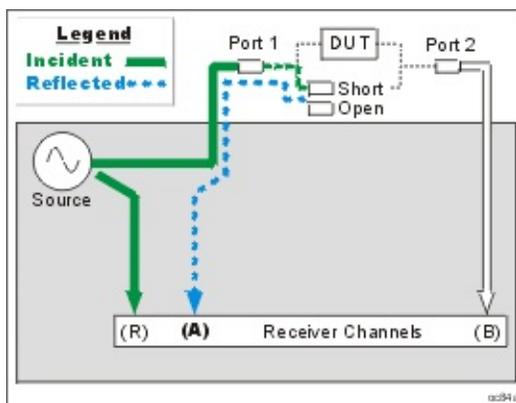
Reflection measurements are made by comparing signal at the 'A' receiver to signal at the 'R1' receiver. This is called a ratio measurement or "A over R1" (A/R1).

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:

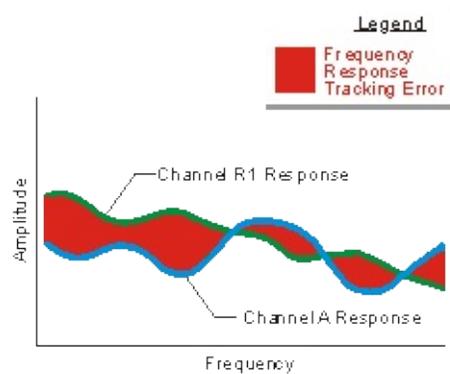
- signal-separation devices
- test cables
- adapters
- variations between the reference and test signal paths

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

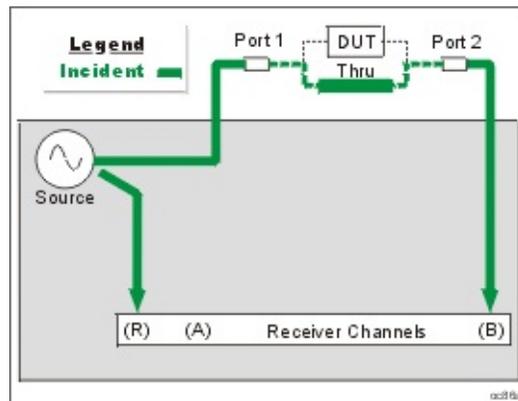
Transmission measurements are made by comparing signal at the 'B' receiver to signal at the 'R1' receiver. This is called a ratio measurement or "B over R1" (B/R1).

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:

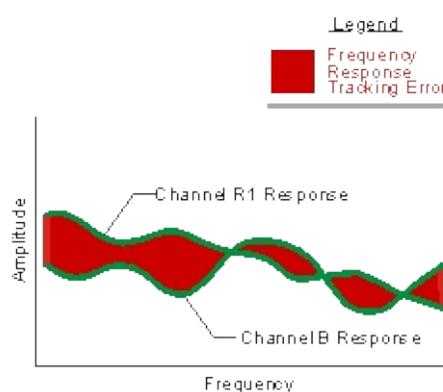
- signal-separation devices
- test cables
- adapters
- variations between the reference and test signal paths

Frequency response transmission tracking error can be characterized and reduced by the analyzer.



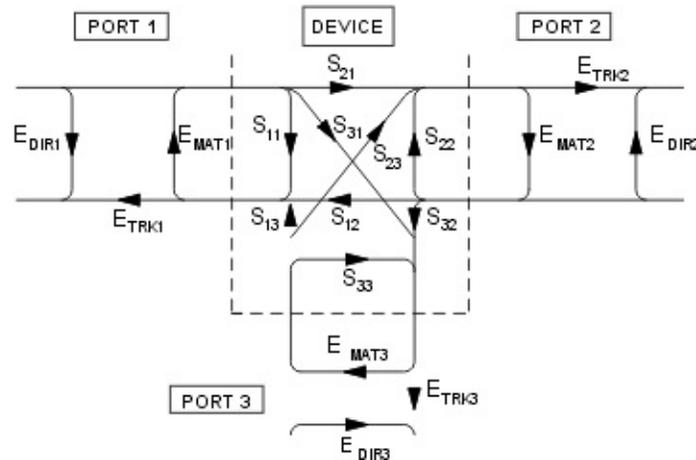
How the Analyzer Measures and Reduces Frequency Response Transmission Tracking Error.

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.

		Source Port			
		1	2	3	4
R e c o r d	1	DIR 1,1 RTRK 1,1 SRM 1,1	LDM 1,2 TTRK 1,2 XTLK 1,2	LDM 1,3 TTRK 1,3 XTLK 1,3	LDM 1,4 TTRK 1,4 XTLK 1,4
	2	LDM 2,1 TTRK 2,1 XTLK	DIR 2,2 RTRK 2,2 SRM 2,2	LDM 2,3 TTRK 2,3 XTLK 2,3	LDM 2,4 TTRK 2,4 XTLK

	2,1			2,4
3	LDM 3,1 TTRK 3,1 XTLK 3,1	LDM 3,2 TTRK 3,2 XTLK 3,2	DIR 3,3 RTRK 3,3 SRM 3,3	LDM 3,4 TTRK 3,4 XTLK 3,4
4	LDM 4,1 TTRK 4,1 XTLK 4,1	LDM 4,2 TTRK 4,2 XTLK 4,2	LDM 4,3 TTRK 4,3 XTLK 4,3	DIR 4,4 RTRK 4,4 SRM 4,4

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 on a PNA with four native ports can give higher accuracy under some conditions.

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.

	A	B	C	D
A	AA	AB	AC	AD
B	BA	BB	BC	BD
C	CA	CB	CC	CD
D	DA	DB	DC	DD

- 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 THRU's 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:

$$\frac{BA * AC}{AA} = \frac{B * \cancel{AA} * C}{\cancel{AA}} = BC$$

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.Keysight.com/find/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.

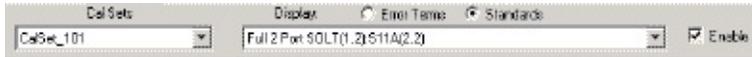
Viewing Cal Set Data

- 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.

See the error terms equations.

How to access Cal Set Viewer	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press CAL2. then [Manage Cals]3. then [Cal Set Viewer]	<ol style="list-style-type: none">1. Click Response2. then Cal3. then Manage Cals4. 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 multipoint calibrations.

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](#)

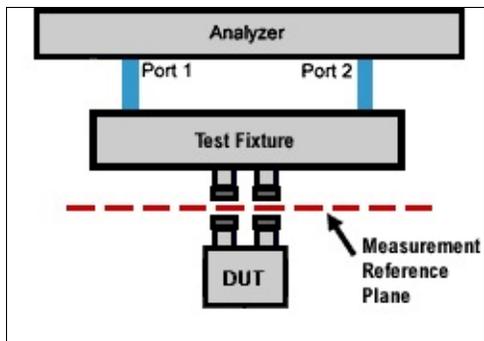
Learn how to [determine the validity of your calibration](#).

[See other Calibration Topics](#)

Measurement Reference Plane

Most measurement setups will NOT allow you to connect a device under test (DUT) directly to the analyzer front panel test ports. More likely, you would connect your device to test fixtures, adapters, or cables that are connected to the analyzer.

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 [Analyzer 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.

	Data-Based Model	Polynomial Model
How accurate is the model?	Provides highest calibration accuracy. Eliminates the errors that can be the result of polynomial model approximations.	Provides high calibration accuracy.
How does the model define calibration standards?	Uses S-Parameter measurements.	Uses traditional four-term polynomial calibration standard modeling parameters.

How do I manually edit the definitions of the calibration standards when using the model?	Use the Advanced Modify Cal Kit function.	Use the Advanced Modify Cal Kit function.
How do I use the Calibration Wizard with the model?	Use only the SmartCal (Guided) Calibration method.	Use the SmartCal (Guided) or the Unguided Mechanical Calibration methods.

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 remotely ONLY using SCPI or COM.

The uncorrected isolation between the test ports of the analyzer is exceptional (typically >100 dB). Therefore, you should only perform the Isolation portion of a 2-port calibration when you require isolation that is better than 100 dB.

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 analyzer 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 analyzer 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](#).

Calibration Thru Methods

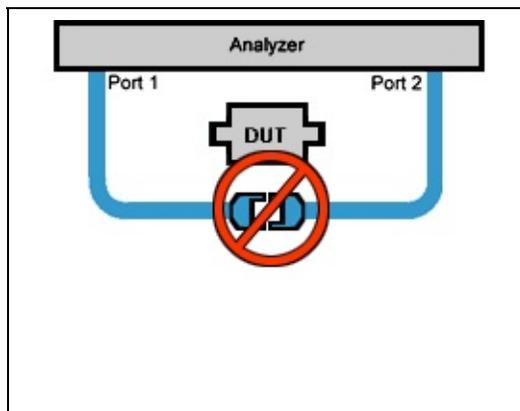
- [What is a Non-Insertable Device](#)
- [Choosing a Thru Method](#)
- [Flush Thru](#)
- [Adapter Removal](#)
- [Swap Adapters](#) (separate topic)
- [Defined Thru](#)
- [Unknown Thru](#)
- [ECal Thru Method Choices](#)

Other Cal Topics

What is a Non-Insertable Device

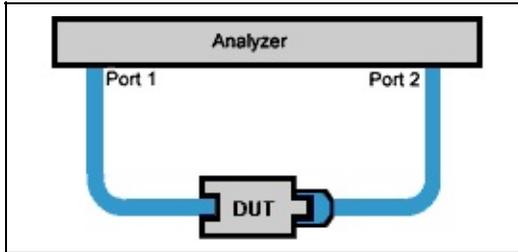
To understand the Thru method choices, you must first understand what is meant by "Non-Insertable device". These definitions also apply to ECal modules. Substitute "ECal module" for "device". Then see [ECal Thru Method Choices](#).

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. Click Response, then Cal, then **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 analyzer 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.
- NOT recommended when there is 40 dB or more of combined loss in the Unknown Thru and calibration path. This would NOT allow enough signal to accurately measure at the receiver.

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 analyzer.

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 40 dB of combined loss in the Unknown Thru and calibration path.
- Must be reciprocal: $S_{21}=S_{12}$.
- 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.

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 as accurate as 'ECal Thru as Unknown Thru'

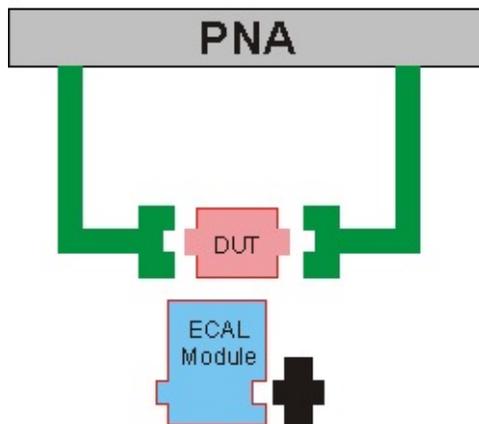
Unknown Thru

- 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.
- Adapter removal performs 2-port measurements on both sides of the adapter.



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.

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 S11 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 S11 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 S21 isolation, connect two loads: one on port 1 and one on port 2. Measure the magnitude of S21 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 Keysight-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

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

1. Press **CAL**

Using Menus

1. Click **Response**

2. then **[More]**
3. then **[ECal]**
4. then **[Confidence Check]**

2. then **Cal**
3. then **More**
4. then **ECal**
5. then **Confidence Check**

Programming Commands

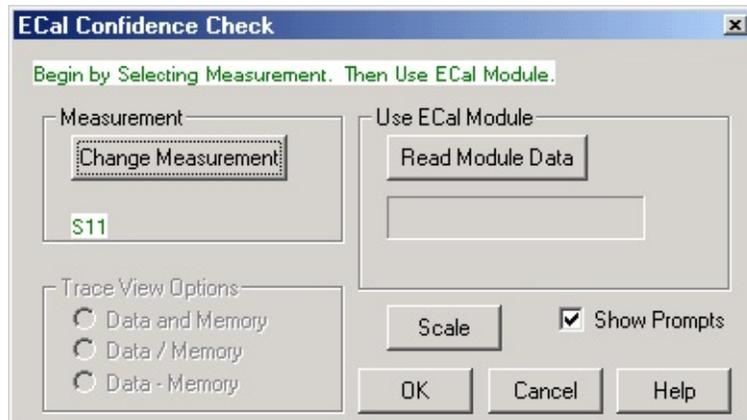
On the following **ECal Confidence Check dialog box**:

4. Click **Read Module Data**. The following occurs:
 - ECal module is set to "confidence state".
 - Analyzer reads and displays stored data.
 - Analyzer 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.
- Displays the factory and user characterizations data stored in the ECal module.
[Learn more.](#)

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.

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 Keysight.

See [Analyzer Accessories](#) for a list of Keysight verification kits.

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 contains a set of physical devices called standards. Each standard has a precisely known or predictable magnitude and phase response as a function of frequency. All Keysight Cal Kits and their standard definitions are stored in the analyzer. For a list of Keysight 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 the definitions ("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 four basic types that define the form or structure of the model to be used with that standard. The following are the four basic standard types:

Standard	Terminal Impedance
SHORT	zero ohms
OPEN	infinite ohms
LOAD	system impedance, Z_0
THRU/LINE	no terminal impedance

Learn about other Calibration Standards:

- [Data-Based Standard](#)
- [Sliding Load](#)
- [Offset Load](#)
- [Arbitrary Impedance 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:

$$\text{Loss} \left(\frac{\Omega}{s} \right) = \frac{\text{loss (dB)} \times Z_0(\Omega)}{4.3429(\text{dB}) \times \text{delay(s)}}$$

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 network analyzer 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, **SOLT** or **TRL** tab.

The different classes used in the analyzer

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 network analyzer 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 Keysight Vector Network Analyzers \(Application Note 1287-11\)](#)

Modify Calibration Kits

The following topics discuss Modifying Calibration Kits:

In this Topic

- [How to Modify Cal Kits](#)
- [Manage Cal Kits dialog](#)
- [Cal Kits and Firmware Upgrades](#)
- [Import Kit dialog](#)

Using VNA CalKit Editor

- [Connectors Tab](#)
- [Standards Tab](#)
- [SOLT Tab](#)
- [TRL Tab](#)

Concepts

- [Why Modify a Cal Kit](#)
- [VNA Cal Kit File Types](#)

Procedures

- [How to Create a New Cal Kit from an Existing Cal Kit](#)
- [Creating Custom Calibration Kits using a New Connector Family](#)
- [Noise Figure and TRL Cal \(separate topic\)](#)

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]**

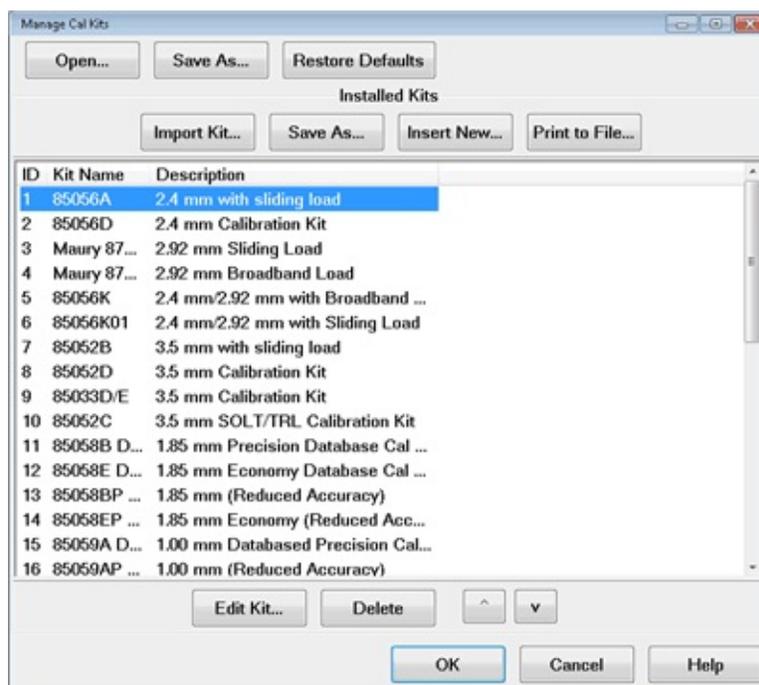
Using Menus

1. Click **Response**
2. then **Cal**
3. then **More**
4. then **Cal Kit**

Programming Commands

Manage Cal Kits

Manage Cal Kits dialog box help



The Manage Cal Kits dialog allows you to define the cal kits in the active workspace. These cal kits may be edited, created, or saved to a cal kit file.

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. [Learn how to import cal kit files.](#)
- All 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 analyzer 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 VNA to a *.wks file.

Restore Defaults Re-installs the default factory contents of all Keysight cal kits from the VNA hard drive. The factory Keysight cal kits are stored on the hard drive at c:\users\public\network analyzer\PnaCalKits\factory.

Installed Kits

Import Kit Starts the **Import Kit** dialog box.

Save As Saves the selected calibration kit and definitions (using **.ckt** file type).

Insert New Starts a blank **Edit Kit dialog box** to create new calibration kit definitions.

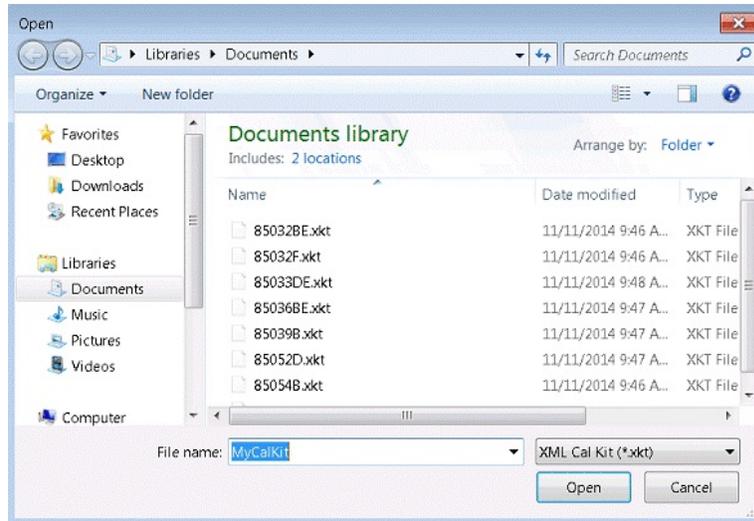
Print to File Prints the contents of the selected cal kit to a **.prn** file.

Edit Kit... Starts the **Connectors tab** of the Edit Kit dialog box to modify selected calibration kit definitions.

Delete Deletes selected calibration kit file.

^ Selects previous / next calibration kit in list.

Import Cal Kit dialog box help



Note: There is no limit to the number of cal kits that can be imported. However, during an **Unguided cal**, you can access **ONLY** mechanical cal kits #1 through 95.

Imports a cal kit file into the active cal kit workspace.

Files of type Select the file type of your Cal Kit. Learn more about [VNA Cal Kit File Types](#)

File name Navigate and select your cal kit file name.

Open Loads the selected file into the active cal kit workspace.

Note: See [Cal Kits and Firmware Upgrades](#)

Importing Cal Kits from "legacy" network analyzers

Cal kit files from "legacy" network analyzers (such as the 8510 or 8753) may not contain information that this VNA requires. Therefore, this VNA may modify the cal kit name, description, standards, and class assignments. You may need to correct these modifications after importing your legacy cal kit to meet your specific requirements.

- "Legacy" cal kit files are referenced to the VNA test port gender while modern cal kit files are referenced to the Device Under Test (DUT) connector gender. Therefore, when a legacy cal kit is imported, the genders of the standards in the legacy cal kit will be automatically reversed in the new cal kit.
- Legacy cal kits do not contain connector definitions. If a coaxial legacy kit is imported, then male and female coax connector definitions will be added to the kit. If a waveguide legacy kit is imported, then a genderless waveguide connector definition will be added to the kit.

Why Modify a Cal Kit

For most applications, the default calibration kit models provide sufficient accuracy for your calibration. However, several situations may exist that would 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. For example: A 7 mm LOAD is determined to be 50.4 Ω instead of 50.0 Ω .
- Modifying the THRU definition when performing a calibration for a non-insertable device.
- Performing a TRL calibration.

Last modified:

2-Apr-2013 New file

Creating 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 (50Ω)

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 **Connectors Tab**, click **Add** to name the new connector family.
2. Enter the Kit Description for the custom cal kit.
3. Click **Add** in the Connectors section of the dialog box.
4. Enter a Connector Family name.
5. Enter a Description of the connector.
6. Select the Gender of one of the connectors.
7. Enter the minimum and maximum Frequency Range.
8. Enter the Impedance.
9. Click the down-arrow to select the Media.
10. Enter the cut-off frequency
11. Click **Apply**.
12. Click **OK**.
13. If you need to add another connector gender, in the **Connectors Tab**, click **Add** in the Connectors section again for the next connector gender.
14. 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.

Enter Standards

Now that the connector family is added to the custom cal kit, you are ready to add new calibration standards.

1. In the **Standards Tab**, under the list of standards, click **Add**.
2. Select the type of standard (OPEN, SHORT, LOAD, or THRU), then click **OK**.
3. 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.
4. Repeat steps 2 - 3, as necessary, to add all standards and definitions to the new custom cal kit.
5. Assign each of the standards to a calibration class. This is done through the **TRL Tab** or **SOLT Tab**
6. Save the Cal Kit.

Last modified:

2-Apr-2013 New file

How to Create a New Cal Kit from an Existing Cal Kit

You can create a new custom Cal Kit using an existing Cal Kit as a starting point.

Here is how:

1. From the [Edit Cal Kits](#) dialog, load the Cal Kit you want to use as a starting point.
2. Immediately click **Save As** and change the file name. Select either ***.xkt** or ***.ckt** file type. [Learn more about these file types.](#)
3. Make modifications to your new custom Cal Kit as required.
4. Routinely save your work by clicking **Save**.

See Also

[About PNA Cal Kits and Firmware Upgrades](#)

Last modified:

2-Apr-2013 New file

VNA Cal Kit File Types

The VNA Cal Kit editor can open the following types of Cal Kit files:

VNA Families	File Type
Current PNA Series Cal Kits	*.xkt (default with A.10.00 and above) *.ckt *.ckx
Old PNA Series Cal Kit (before A.07.50)	*.ck1
8510 Cal Kit	.CK_*
8753, 8752, 8719, 8720, or 8722 Cal Kit	*.ck
Current FieldFox Cal Kits	*.xkt
Previous FieldFox Cal Kits	*.xml

The current revision of Cal Kit files can be downloaded at <http://na.support.keysight.com/pna/caldefs/stddefs.html>

File Save (As)

The VNA Cal Kit Editor can save Cal Kits in three formats:

- ***.xkt** - An XML-based format, supported by multiple VNA families (the default format).
- ***.ckt** - VNA binary format - can be read by VNA and PNA Rev. A.07.50 and higher. This format is provided for backwards compatibility with older PNA firmware revisions (going back to A.07.50) and may not support future new cal kit capabilities, which is expected of the *.xkt format.
- ***.prn** - Cal kit print files. This is a text file format which can be read into spreadsheets, but the Cal Kit Editor does not read-in these files. These files are only produced as a form of documentation.

About Opening Legacy VNA Kits

Cal kit files from Keysight "legacy" network analyzers (listed above) may not contain information that the VNA requires. When loaded into the VNA Cal Kit Editor, the cal kit name and description, the cal standards, and the cal class assignments will be modified 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. Modern VNA 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.
-

Connectors Tab

Cal Kit Name Allows you to change the Name of the selected calibration kit.

Cal Kit Description Allows you to change the description of the selected calibration kit.

Connector Family .Click the down arrow to select the connector family associated with the Cal Kit.

Add Starts the **Add Connector** dialog box which allows you to add new connector type to the calibration kit.

Delete Deletes - **WITHOUT WARNING** - the selected connector family.

Note: To modify a connector family or name, Add a new connector, then delete the old connector.

The following is the list of Factory-defined connector type strings:

APC 3.5 female	Type N (50) female	7-16 female	X-band waveguide
APC 3.5 male	Type N (50) male	7-16 male	P-band waveguide
APC 2.4 female	Type N (75) female	2.92 mm female	K-band waveguide
APC 2.4 male	Type N (75) male	2.92 mm male	Q-band waveguide
APC 7	Type F (75) female	1.85 mm female	R-band waveguide
	Type F (75) male	1.85 mm male	U-band waveguide
	Type A (50) female	1.0 mm female	V-band waveguide
	Type A (50) male	1.0 mm male	W-band waveguide
	Type B		

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

Gendered - The connector family contains both Male and Female connectors.

Genderless - The connector family does NOT contain Male and Female connectors. APC7 connectors are an example of this connector type.

Impedance

Specify the impedance of the standard.

Media

The medium (or 'geometry') of the connector (COAX or WAVEGUIDE).

Cutoff Frequency If Media is Waveguide, type the low-end cutoff frequency.

Height/Width Ratio Used to calculate waveguide loss. This value is usually on the data sheet for waveguide devices.

About 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**

For waveguide, choose TRL (Thru-Reflect-Line) calibration type . These calibration types are more accurate and take fewer steps than SOLT.

Add Connector Family



Enter a name for the new connector family. Then click **OK**.

Available at the bottom of every tab

Save As - Allows you to save the cal kit to a new file name and type.

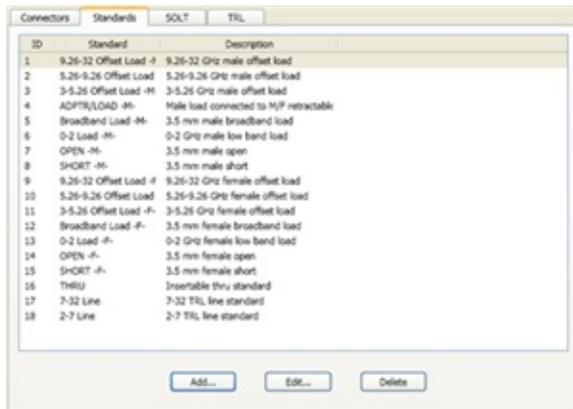
Save - Saves the cal kit to the same file name and type.

Close - Closes the cal kit editing session. The file is NOT saved automatically.

Last modified:

2-Apr-2013 New file

Standards Tab



Allows you to Add, Edit or Delete cal standards in a cal kit.

Add Standard (Open, Short, Load, Thru, or Data-based)

Add Standard dialog box help



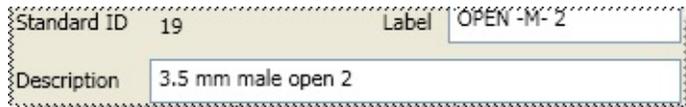
Allows you to add standards to the calibration kit file.

Choose from:

- OPEN
- SHORT
- LOAD
- THRU
- DATA-BASED
- ISOLATION

Standards dialog box help

The following fields apply to **ALL** standard types:



A screenshot of a dialog box with a light beige background and a dashed border. It contains three fields: 'Standard ID' with the value '19', 'Label' with the value 'OPEN-M-2', and 'Description' with the value '3.5 mm male open 2'.

The other areas of the dialog change depending on the type of standard selected.

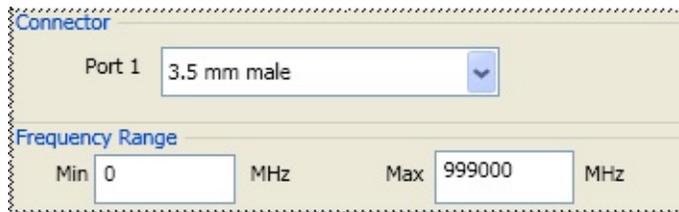
Identification

Standard ID Number in list of standards

Label Type of standard. This usually appears in prompts for standards.

Description Description of standard.

The following fields apply to ALL standard types EXCEPT the Isolation type:



A screenshot of a dialog box with a light beige background and a dashed border. It is divided into two sections. The top section, titled 'Connector', has a 'Port 1' dropdown menu set to '3.5 mm male'. The bottom section, titled 'Frequency Range', has 'Min' set to '0' MHz and 'Max' set to '999000' MHz.

Connector

Indicates the type and gender (Male, Female, None) of the standard.

Thru and Isolation standards have two connectors.

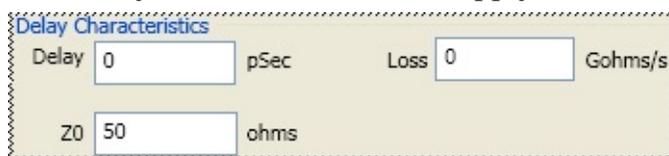
Data-Based standards MAY have two connectors.

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.

The Delay Characteristics fields apply to MOST standard types:



A screenshot of a dialog box with a light beige background and a dashed border. It is titled 'Delay Characteristics' and contains three fields: 'Delay' set to '0' pSec, 'Loss' set to '0' Gohms/s, and 'Z0' set to '50' ohms.

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.

Other fields are unique to standard type

Choose from:

- OPEN
- SHORT
- LOAD
- THRU
- DATA-BASED
- ISOLATION

Open Standard

C0, C1, C2, C3 Specifies the fringing capacitance.

Open Characteristics

C0	<input type="text" value="0"/>	F(e-15)	C2	<input type="text" value="0"/>	F(e-36)/Hz ²
C1	<input type="text" value="0"/>	F(e-27)/Hz	C3	<input type="text" value="0"/>	F(e-45)/Hz ³

These are the unique fields of the dialog. See the areas that are common to all standards.

Short Standard

L0, L1, L2, L3 Specifies the residual inductance.

Short Characteristics

L0	<input type="text" value="0"/>	H(e-12)	L2	<input type="text" value="0"/>	H(e-33)/Hz ²
L1	<input type="text" value="0"/>	H(e-24)/Hz	L3	<input type="text" value="0"/>	H(e-42)/Hz ³

Load Standard

Choose from the following



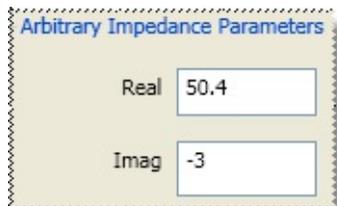
Fixed Load Specifies the load type as Fixed. The fixed load is assumed to be a perfect termination without reflection.

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 that has an impedance value different from system Z_0 .

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

Offset Load Parameters

First Offset THRU

Second Offset 7-32 Line

Load ADPTR/LOAD -M-

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. 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

The screenshot shows a configuration dialog box for a Thru Standard. It is divided into several sections:
1. **Connectors**: Port 1 is set to '3.5 mm female' and Port 2 is set to '3.5 mm male'.
2. **Frequency Range**: Min is '0' MHz and Max is '38800' MHz.
3. **Delay Characteristics**: Delay is '0' pSec, Loss is '0' Gohms/s, and Z0 is '50' ohms.
4. A checkbox for **Virtual Device** is checked.

Connectors - Defines connector type at both ends of the Thru standard.

Virtual Device

Most cal kits have only one Thru standard definition for SOLT calibrations. For these cases, use the default selections (checked for zero-length Thrus and cleared for non-zero-length Thrus).

This checkbox is used to make forward and reverse measurements of your Thru standard for the same pair of ports in two separate steps. This is NOT common for zero-length (Flush) Thru standards.

When **checked**, calibration prompts involving that Thru will **omit** the Description. For example “Connect port 1 to port 2”. This is the common prompt for Flush-Thru standards.

When **cleared**, calibration prompts for that Thru will **include** the Description. For example “Connect <standard description> between ports 1 and 2”.

To make forward and reverse measurements of your Thru standard for the same pair of ports as two separate steps, do the following:

1. Create separate definitions of the Thru standard(s) using the same settings, except for the Label and Description. **Clear** this checkbox for BOTH definitions.
2. For one Thru definition, in the label and description include the word 'FORWARD' to prompt the operator to use this standard for the forward measurement. Assign this standard to the SOLT “FWD TRANS” and “FWD MATCH” classes of the cal kit.
3. For the Thru definition, in the label and description include the word 'REVERSE' to prompt the operator to use this standard for the reverse measurement. Assign this standard to the SOLT “REV TRANS” and “REV MATCH” classes of the cal kit.
4. When you perform SOLT calibrations using this cal kit, the forward measurements of the Thru will be measured in one connection step, and the reverse measurements in another.

Data-Based Standard

Response Data Summary

Number of Response Data Variables = 4
index = 0: Number of Data Variable Values = 900
Data Variable Name = S11
index = 1: Number of Data Variable Values = 900
Data Variable Name = S21

Load Data File...

Accuracy Data Summary

Standard contains no accuracy data

Load Data File...

Virtual Device

Note: To learn how to modify data-based standard files, visit

<http://na.support.keysight.com/pna/dbcal.html>

Learn about the relative accuracy of **Databased versus Polynomial Cal Kits**.

The modified file can then be uploaded into the VNA.

Upload Data From File

Click **Load Data File**, then navigate to the *.dat file which is provided with the data-based Cal Kit. Both Response data and Accuracy (Uncertainty) data is provided in a single *.dat file.

For Advanced Users

Response data can be loaded from a *.s2p or *.cti file.

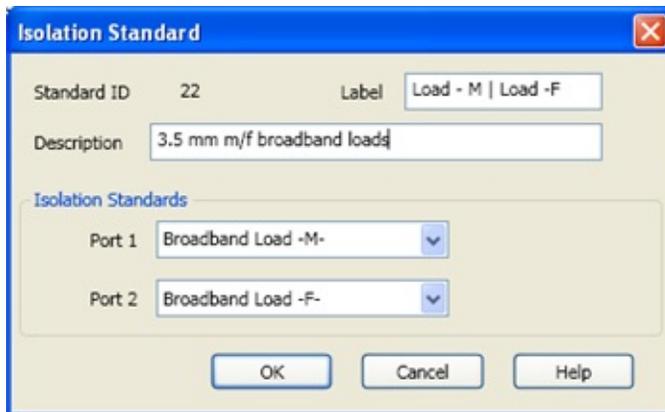
Accuracy data can be loaded from a *.cti file. Starting with Option 015, you can also load data from a *.dsd (S-parameter Data Standard Definition) file. The *.dsd file contains both Response data and Accuracy (Uncertainty) data where the accuracy data is in covariant form. Learn more about Dynamic Uncertainty.

Virtual Device

This checkbox is displayed for a Data-Based cal standard when the standard has been defined to have 2 ports.

- When Cleared (default) calibration prompts for that standard will include it's Description. For example "Connect <standard description> between ports 1 and 2".
- When Checked, calibration prompts for that standard will NOT include its Description, so the prompt will be worded as if the data-based standard is a zero-length Thru connection. For example "Connect port 1 to port 2".

Isolation Standard



The pair of loads are considered one standard.

Both loads are connected to the VNA and measured with the same prompt.

Available at the bottom of every tab

Save As - Allows you to save the cal kit to a new file name and type.

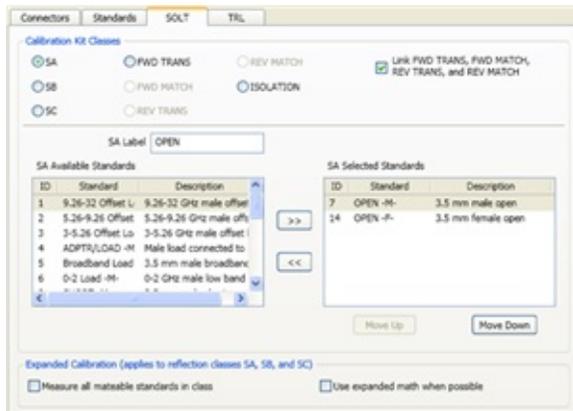
Save - Saves the cal kit to the same file name and type.

Close - Closes the cal kit editing session. The file is NOT saved automatically.

Last modified:

2-Apr-2013 New file

SOLT Tab



Note: This dialog looks similar to the dialog that appears after the **Cal Wizard View / Modify dialog**. However, setting changes in that dialog affect **ONLY** the calibration that is in progress. These settings, accessed through **Modify Cal Kit**, changes the cal kit for all future calibrations that use this cal kit.

Allows you to assign single or multiple standards to **SOLT** Calibration Classes. Click the **TRL tab** to assign standards to TRL Calibration Classes.

1. For each Cal Kit Class, select **Available Standards** from the left list, then click **>>** to copy the standard to the Cal Kit.
2. Use **MOVE UP** and **MOVE DOWN** to change the **ORDER** of the standards. The order is used in guided calibrations to determine which standards in that class will be used in calibrations that involve the frequency ranges over which the standards are defined. Guided calcs will include standards in the order in which they appear in this class list, and in the case where standards in the class list have frequency ranges that overlap, the order also determines which standard is used for frequencies in the overlap range. For example, let's assume that you define a broadband Short from Min Freq.= 0 Hz and Max Freq.= 999 GHz, and that standard is listed first in the SB or TRL REFLECT class. If you then list a frequency-banded Short with the same connector below the broadband short in those same classes, then guided calibrations would not use the frequency-banded Shorts because the broadband Shorts would always be given priority.

SOLT <cal class> Label

The cal standard category label that appears in the VNA's user interface during **unguided** SOLT calibrations.

Calibration Kit Classes

For each calibration class, select **Available Standards**, then click >> to move to the **Selected Standards** list.

- **SA** - OPEN Standards (standards in the SA class are not always Opens)
- **SB** - SHORT Standards
- **SC** - LOAD Standards
- **FWD / REV Trans and Match** - THRU Standards. Most Cal Kits do NOT include a physical THRU standard, but assume that an Insertable Thru will be used.
- ISOLATION - Isolation standard. For PNA analyzers, ISOLATION calibration is not usually recommended. It could be beneficial in some situations where custom user-supplied test set hardware is being used.

Note: In older VNA models, the SA, SB, and SC classes were labeled S11A, S11B, S11C and S22A, S22B, S22C. Separate classes for ports 1 and 2 is now considered unnecessary, especially in the context of VNA models with more than 2 ports. When saving a cal kit in the PNA *.ckt format with this editor and viewing the kit in PNA firmware with the older user interface, the standard selections for those three classes are duplicated in both the S11 and S22 selections for those classes.

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 VNA 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.

Available at the bottom of every tab

Save As - Allows you to save the cal kit to a new file name and type.

Save - Saves the cal kit to the same file name and type.

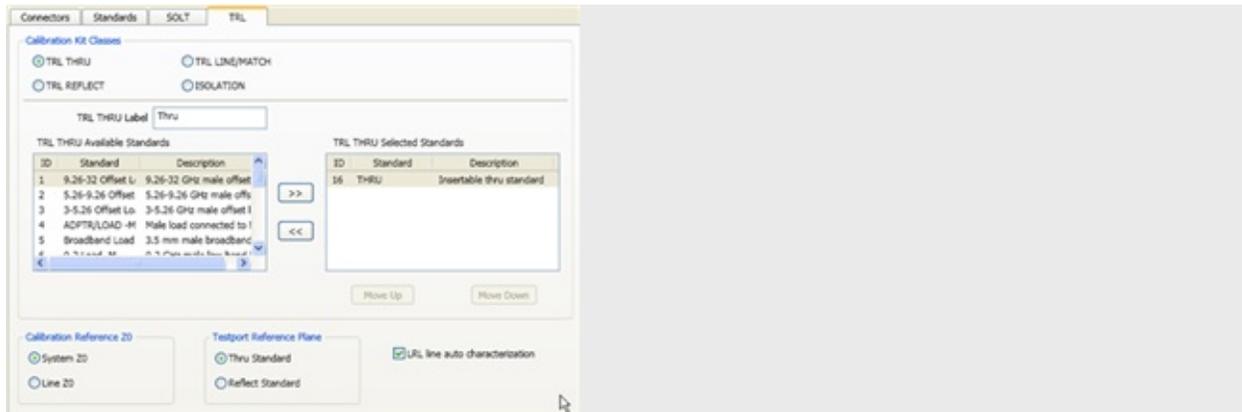
Close - Closes the cal kit editing session. The file is NOT saved automatically.

Last modified:

2-Apr-2013 New file

TRL Tab

On the image below, click a setting area to learn more.



Note: This dialog will look different if you clicked Help at the Cal Wizard View / Modify dialog. The functionality is the same.

Allows you to assign single or multiple standards to Calibration Classes.

1. For each Cal Kit Class, select **Available Standards** from the left list, then click >> to copy the standard to the Cal Kit.
2. Use **MOVE UP** and **MOVE DOWN** to change the **ORDER** of the standards. The order is used in guided calibrations to determine which standards in that class will be used in calibrations that involve the frequency ranges over which the standards are defined. Guided calcs will include standards in the order in which they appear in this class list, and in the case where standards in the class list have frequency ranges that overlap, the order also determines which standard is used for frequencies in the overlap range. For example, let's assume that you define a broadband Short from Min Freq.= 0 Hz and Max Freq.= 999 GHz, and that standard is listed first in the SB or TRL REFLECT class. If you then list a frequency-banded Short with the same connector below the broadband short in those same classes, then guided calibrations would not use the frequency-banded Shorts because the broadband Shorts would always be given priority.

Note: The TRL LINE/MATCH class has a slight exception to these prioritization behaviors. In general, Line standards are given a higher priority than Match

standards. So if a Line standard and a Match standard are defined to have the same frequency range and the Match standard is listed above the Line standard in the class order, a guided TRL cal will still prefer to use the Line standard rather than the Match standard.

TRL <cal class> Label

The cal standard category label that appears in the VNA's user interface during **unguided** TRL calibrations.

Cal Kit Classes

- For PNA analyzers, ISOLATION calibration is not usually recommended. It could be beneficial in some situations where custom user-supplied test set hardware is being used.

TRL 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.
- The THRU standard and LINE standard have the same characteristic impedance and are perfectly matched. They define the reference impedance of the calibration.
- If a THRU standard with the correct connectors is NOT available, an adapter removal cal can be performed.

TRL REFLECT

- The REFLECT standard can be anything with a high reflection, as long as it is the same when connected to one or more VNA ports.
- The REFLECT standard on each port is identical.
- 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.

TRL LINE

The LINE and THRU standards establish the reference impedance for the measurement after the calibration is completed. 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 $f_1 \times f_2$).

Note: The TRL LINE standard must have a delay that is greater than 0 (zero) ps. Otherwise, calibration correction calculations will contain unpredictable results.

TRL 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.

Calibration Reference Z0 (TRL only)

System Z0 The system impedance is used as the reference impedance. Choose when the desired test port impedance differs from the impedance of the LINE standard. Also, choose 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.

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 VNA 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.

Available at the bottom of every tab

Save As - Allows you to save the cal kit to a new file name and type.

Save - Saves the cal kit to the same file name and type.

Close - Closes the cal kit editing session. The file is NOT saved automatically.

Markers

Markers provide a numerical readout of measured data, a search capability for specific values, and can change stimulus settings. This topic discusses all aspects of markers.

- [Number of General Purpose and Reference Markers](#)
- [Creating and Moving Markers](#)
- [Marker Dialog](#)
- [Searching with Markers](#)
- [Marker Functions](#) (Change Instrument Settings)
- [Marker Display](#)
- [Marker Table](#)

Note: Marker Readout can be turned ON / OFF and customized from the **View/Display** menu. [Learn more.](#)

Other Analyze Data topics

How to Create Markers	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press MARKER2. then [Marker n]	<ol style="list-style-type: none">1. Click Marker/Analysis2. then Marker3. select a marker number

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 ∇ . 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

Programming Commands



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 (and Reference) Markers 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 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**. **See Also: Number of General Purpose and Reference Markers.**

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.

(Marker) Format Displays the marker data in a format that you choose. The Trace Default setting has the same marker and grid formats. Choose from the following:

Log/Phase	Log Mag	Real
Linear/Phase	Linear Mag	Imaginary
Real/Imag	Phase	Kelvin
R+jX (complex impedance)	SWR	Fahrenheit
G+jB (complex admittance)	Delay	Celsius

Noise Marker Format - Available ONLY in IMSpectrum and SA measurement classes. For comparison purposes, electronic noise measurements are often displayed as though the measurement was made in a 1 Hz Res BW. However, making an actual measurement at a 1 Hz Res BW is impossible, and at 10 Hz, extremely slow.

A Noise Marker mathematically calculates the noise measurement **at that single data point** as though it were made using a 1 Hz bandwidth.

To accurately measure noise, the Noise Marker should NOT be placed on, or too close to, a signal. The distance from a signal depends on several factors. To know if an accurate reading is being made, move the Noise Marker until consistent measurements are displayed in adjacent data points.

All Off Switches OFF all markers on the active trace.

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. 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.

Coupling Method - Determines the scope of coupling. Choose from the following:

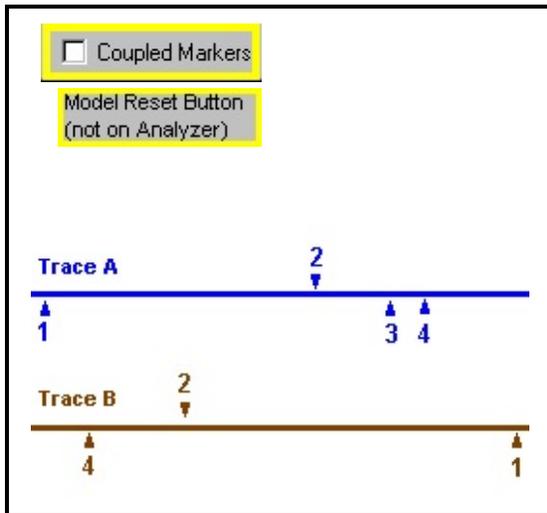
- **Channel** – Traces that are associated with the same channel are eligible to have their markers coupled. For example, all traces on Channel 1 have their markers coupled. Changing a marker stimulus or ON | OFF state on a trace associated

with Channel 1 has NO effect on markers on Channel 2 traces.

- **All** – All traces on all channels will be eligible to have their markers coupled.

Coupled Markers Model

This model simulates the use of coupled markers in the VNA:



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.

Searching with Markers

You can use markers to search and return data for the following trace criteria:

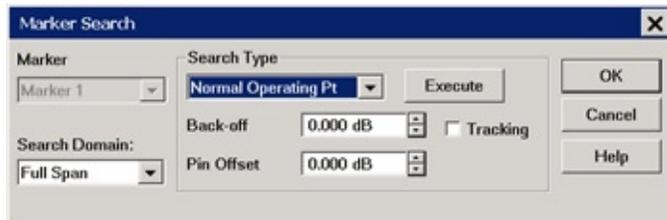
- **Max and Min**: find the highest or lowest points on the trace
- **Peak**, then move to other peaks (left, right, next highest)
- **Target Value**: find a specific Y-axis value
- **Bandwidth** (Filters)

- **Compression Point** (Amplifiers)
- **About PSAT and PNOP Markers**
 - **Power Saturation** (Amplifiers)
 - **Power Normal Operating Point** (Amplifiers)
- **What is a Peak?**
- **Search Domain**
- **Search Within**

How to Search with Markers	
Using front-panel HARDKEY [softkey] buttons	Using Menus
1. Press SEARCH	1. Click Marker/Analysis 2. then Marker Search

Marker Search dialog box help

← Programming Commands →



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. If there is no valid data match for the search type, the marker will not move from its current position.

- **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

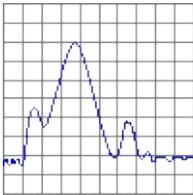


Four markers are automatically deployed to find the first negative or positive bandpass in the selected search domain.

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

◀ Programming Commands ▶

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

← Programming Commands →

Uses Markers 1, 2, and 3 to quickly identify output power saturation parameters of an amplifier.

Back-off is a point at which the output power is sufficiently lower than the saturated output power so that the device under test behaves in a more linear fashion.

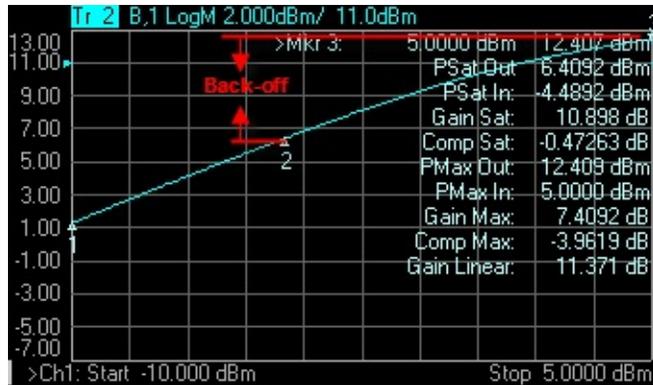
Note: Valid ONLY for Power IN vs Power OUT measurements. These markers can also be used on a CompOut trace with Compression Analysis mode in the Gain Compression Application.

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:

Param	Description	Calculated from...
>Mkr 3	Marker 3 X-axis and Y-axis value	Marker 3 X-axis and Y-axis value
PSat Out	Output power at the saturation point.	Marker 2 Y-axis value
PSat In	Input power at the saturation point.	Marker 2 X-axis value
Gain Sat	Gain at the saturation point.	Psat Out - Psat In
Comp Sat	Compression at the saturation point.	Gain Sat - Gain Linear
PMax Out	Maximum output power.	Marker 3 Y-axis value
PMax In	Input power at the maximum output power.	Marker 3 X-axis value
Gain Max	Gain at the maximum output power.	PMax Out - PMax In
Comp Max	Compression at the maximum output power.	Gain Max - Gain Linear

Gain Linear	Linear gain at the first data point.	Marker 1 - Y-axis value MINUS X-axis value
--------------------	--------------------------------------	--

- **Comp. Not Found** is displayed when the requested Back-off point is not found.
- When **Discrete** marker is NOT selected (the default setting), the three markers find an interpolated value between the two closest data points.
- When **Discrete** marker is selected (NOT interpolated), the three markers reside on the closest data points.

Power Normal Operating Point Marker

◀ Programming Commands ▶

Uses Markers 1, 2, 3, and 4 to quickly identify Normal Operating Point parameters of an amplifier.

Back-off is a point at which the output power is sufficiently lower than the saturated output power so that the device under test behaves in a more linear fashion.

The power normal operating point is the output power where the input is offset from the back-off input power by the Pin Offset.

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:

Param	Description	Calculated from...
>Mkr 4	Marker 4 - X-axis and Y-axis values	Marker 4 - X-axis and Y-axis values
Pnop Out	Output power at the power normal operating point.	Marker 4 Y-axis value
Pnop In	Input power at the power normal operating point.	Marker 4 X-axis value
Pnop Gain	Gain at the power normal operating point.	Pnop Out - Pnop In
Pnop Comp	Compression at the power normal operating point.	Pnop Gain - Linear Gain*
PMax Out	Maximum output power.	Marker 3 Y-axis value
PMax In	Input power at the maximum output power.	Marker 3 X-axis value
Gain	Gain at the maximum output power.	PMax Out - PMax In

Max		
Comp Max	Compression at the maximum output power.	Gain Max - Linear Gain*
PBO Out	Output power at the back-off point.	Marker 2 Y-axis
PBO In	Input power at the back-off point.	Marker 2 X-axis
PBO Gain	Gain at the back-off point.	PBO Out - PBO In

***Linear Gain (not shown): Marker 1 - Y-axis value MINUS X-axis value**

- **PNOP Not Found** is 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.

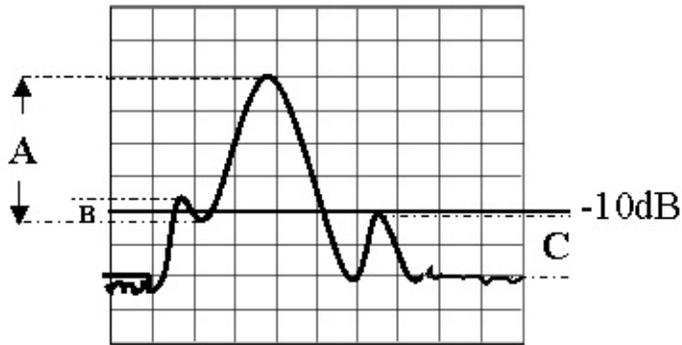
Example:

Threshold Setting: -10dB

Excursion Setting: 1dB

Scale = 1 dB / Division

Mouse over the graphic to find a valid peak.



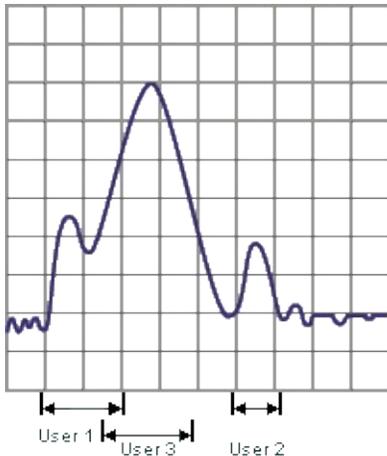
- **Peak A** = Valid Peak (Above Threshold and Excursion Settings)
- **Peak B** = Invalid Peak (Below Excursion Setting)
- **Peak C** = Invalid Peak (Below Threshold Setting)

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.



Search Within

Using the **Zoom feature**, you can search for the Maximum, Minimum, or a Target value within a 'zoomed' frequency range. The zoomed frequency range becomes the **User 16** Search Domain span.

A marker is created if not already present on the trace. If markers are already present on the trace, the lowest marker is moved to the found value.

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.
3. Select **Search Within**.
4. Then choose from the following:
 - **Max** - A marker moves to the HIGHEST value within the zoomed range.
 - **Min** - A marker moves to the LOWEST value within the zoomed range.
 - **Target** - A marker moves to the first value within the zoomed range that is currently set in the **Marker Search 'Target' setting**. The same Discrete Marker rules apply as those for the standard Target Marker Search.

Marker Functions - Change Instrument Settings

The following settings change the relevant PNA settings to the position of the active maker.



How to change instrument settings using markers

Using front-panel HARDKEY [softkey] buttons

1. Press **MARKER**
2. then [**Marker Function**]

Using Menus

1. Click **Marker/Analysis**
2. then **Marker Function**

Marker Function dialog box help

Programming Commands



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.**

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:

Using front-panel HARDKEY [softkey] buttons

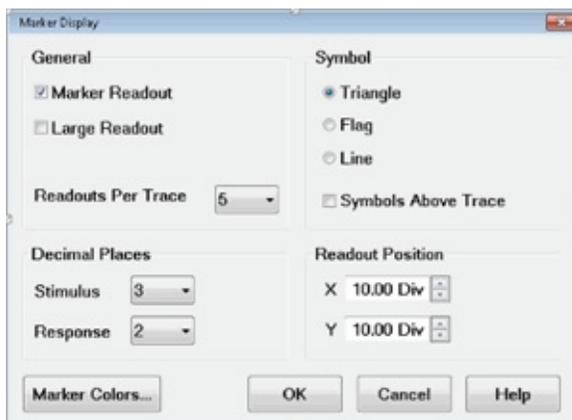
1. Press **MARKER**
2. then **[Properties]**
3. then **[Marker Display]**

Using Menus

1. Click **Marker/Analysis**
2. then **Marker**
3. then **Marker Display**

Marker Display dialog box help

Programming Commands



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.

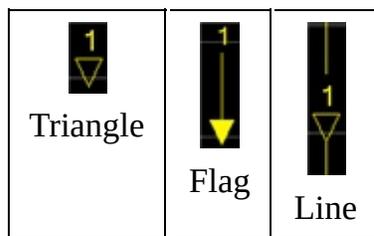
Cleared - Shows the marker readout in normal font size.

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.



Line symbols are NOT used on Smith or Polar **display formats**.

Symbols can be set independently for each window.

Symbols Above Trace

Cleared - ONLY the active marker is displayed above the trace. Inactive markers are displayed below the trace.

Checked - ALL marker symbols are displayed above the trace. The active marker is always filled solid.

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	Using Menus
<ol style="list-style-type: none">1. Press DISPLAY2. then [More]3. then [Tables]4. then [Marker Table]	<ol style="list-style-type: none">1. Click Response2. then Display3. then Tables4. then Marker Table

 **Programming Commands** 

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

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

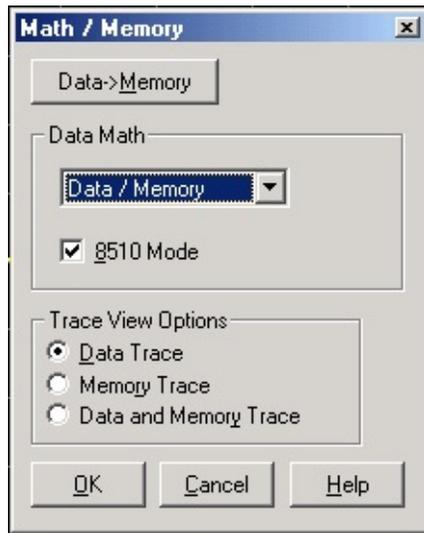
1. Press **MEMORY**

Using Menus

1. Click **Marker/Analysis**
2. then **Memory**
3. then **Memory**

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.support.keysight.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](#).

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.

S21 values to compare	Data/Mem	Data-Mem
0.5 dB and 0.6 dB	0.1 dB	-39 dB
0.5 dB and 0.7 dB	0.2 dB	-33 dB

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.

S11 values to compare	Data/Mem	Data-Mem
-10 dB and -12 dB	2 dB	-24 dB
-50 dB and -52 dB	2 dB	-64 dB

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.

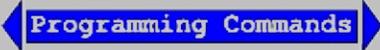
The user ranges for a channel can overlap each other.

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	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press ANALYSIS 2. then [Statistics] 3. then [Trace Statistics] 	<ol style="list-style-type: none"> 1. Click Marker/Analysis 2. then Analysis 3. then Trace Statistics
	

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

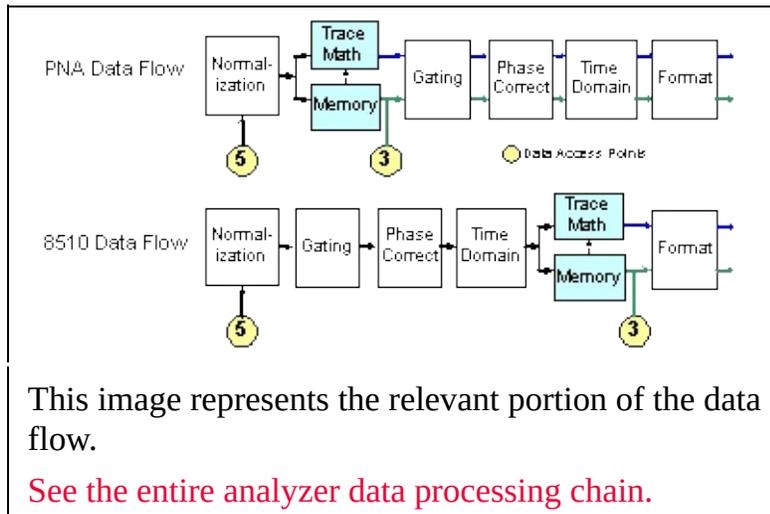
3-Sep-2008	Removed legacy content
28-Aug-2008	Added Memory trace utility note
22-Apr-2008	Added 8510 preference link
27-Aug-2007	Edited trace display settings
2-Feb-2007	MX added UI

8510 Mode

On the **Trace Math** dialog, check 8510 Mode to simulate the Keysight 8510 data processing chain as it pertains to Trace Math and Memory. This setting applies to all channels. When the box is checked or cleared, the analyzer performs an **Instrument Preset** and retains its setting through subsequent Instrument Presets.

This setting can be saved as part of an **instrument state**. However, when recalled, this setting is assumed only temporarily. When a subsequent analyzer Preset is performed, the analyzer reverts to the setting that was in effect before the state was recalled.

You can **set a preference** to always use 8510 mode.



A settings change in any of the operations that occur after the Memory operation on the above analyzer **Data Flow** diagram changes both the Data trace and the Memory trace. For example, after storing a data trace to memory, when you change the format for the Data Trace, the format for the Memory Trace is also changed to the same setting.

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.

Equation Editor is available with M937x Option 102. [Learn more.](#)

- [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

[Using Noise Power Traces in Equation Editor](#)

[Equation Editor and MATLAB](#)

[Equation Editor Import Functions](#)

[External DC Meter Data Conversion](#)

- [BestFit.dll](#)
- [EqnErrorTerms.dll](#)
- [Expansion.dll](#)

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 analyzer. 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 analyzer to make a directivity measurement of your 3-port DUT. This is not a native 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:

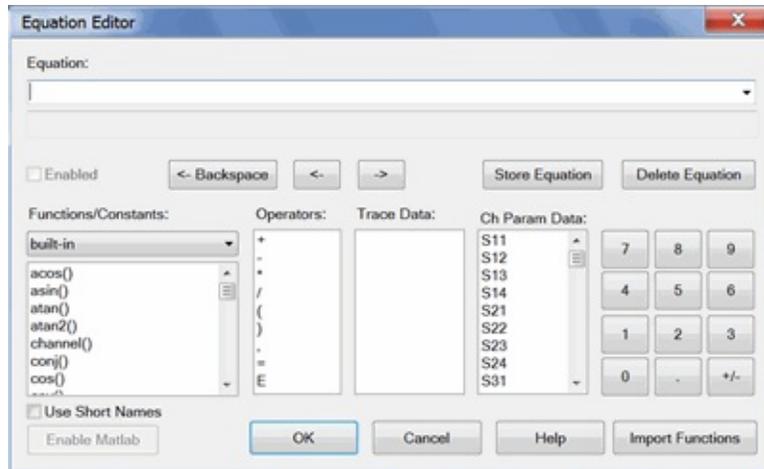
$$\text{DIR} = S12 * S23 / S13$$

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	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press ANALYSIS 2. then [Equation Editor] 	<ol style="list-style-type: none"> 1. Click Marker/Analysis 2. then Analysis 3. then Equation Editor
	

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). See [Data that is used in Equations](#).

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.

Keypad: Provided to allow navigation of the entire dialog with a mouse.

Import Functions Click to launch the **Import Functions** Dialog box.

Use Short Names Some functions have shortened names that are entered automatically when checked. Both long and short names can be used interchangeably.

Enable Matlab Available when a full MATLAB version is installed by you on the analyzer. [Learn more.](#)

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 $(S_{21})/(1-S_{11})$ or an equation $(DIR = S_{12} * S_{23} / S_{13})$. 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 screen.
- **Data** - analyzer measurements that are acquired but not displayed. When an equation trace refers to data that is not displayed, the analyzer 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 correction is applied to the channel, equation editor traces in that channel will attempt to use corrected parameter data regardless if correction is on/off for the measurement. If there is no corrected data available, then raw data will be used. TrX notation always ignores the correction state.

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 correction is applied to the channel, equation editor traces in that channel will attempt to use corrected parameter data regardless if correction is on/off for the measurement. If there is no corrected data available, then raw data will be used. TrX notation always ignores the correction state.

Referring to Traces in a different channel

When the equation trace refers to a trace on a different channel:

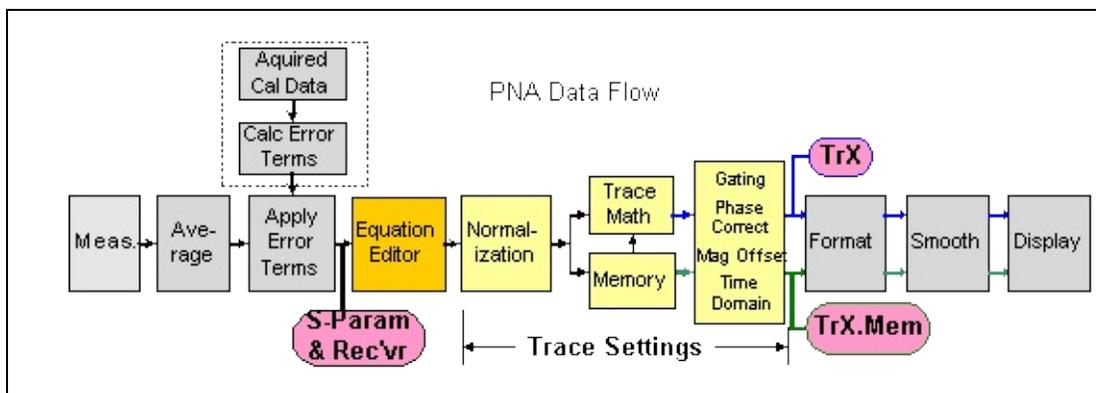
- The trace must already be displayed.
- Must refer to the trace using **TrX** notation.
- The Equation trace and the referred trace **MUST** have the same number of data points or the Enable checkbox will not be available.
- The Equation trace is updated when the last referred data in the same channel 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 analyzer 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.
- If correction is applied to the channel, equation editor traces in that channel will attempt to use corrected parameter data regardless if correction is on/off for the measurement. If there is no corrected data available, then raw data will be used. TrX notation always ignores the correction state.
- 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.
When using a mouse with the analyzer, hover over a function in the dialog to learn how it is used.

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.

Function/Constant	Description
acos(scalar a)	returns the arc cosine of a in radians
asin(scalar a)	returns the arc sine of a in radians
atan(scalar a)	returns the arc tangent of a in radians
atan2	returns the phase of complex a = (re,im) in radians has the following two argument sets: <ul style="list-style-type: none"> • atan2(complex a) - returns the phase in radians • atan2(scalar a, scalar b)
conj(complex a)	takes a and returns the complex conjugate
cos(complex a)	takes a in radians and returns the cosine
cpx(scalar a, scalar b)	returns a complex value (a+ib) from two scalar values
e	returns the constant $\approx 2.71828\dots$
exp(complex a)	returns the exponential of a
getNumPoints()	returns the number of points for the current sweep
im(complex a)	returns the imag part of a as the scalar part of the result (zeroes the imag part)

<p>kfac(complex a, complex b, complex c, complex d) when entered in EE: kfac(S11,S21,S12,S22)</p>	<p>k-factor: $k = (1 - a ^2 - d ^2 + a*d-b*c ^2) / (2 * b*c)$ returns a scalar result - the imaginary part of the complex result is always 0</p>
<p>ln(complex a)</p>	<p>returns the natural logarithm of a</p>
<p>log10(complex a)</p>	<p>returns the base 10 logarithm of a</p>
<p>mag(complex a)</p>	<p>returns sqrt(a.re*a.re+a.im*a.im)</p>
<p>max(complex a, complex b, ...)</p>	<p>returns the complex value that has the largest magnitude of a list of values.</p>
<p>max_hold(complex a)</p>	<p>holds the current maximums of the sweep. Disable the equation to reset. See example</p>
<p>median(complex a, complex b,...)</p>	<p>returns the median of a list of complex values</p> <ul style="list-style-type: none"> • The median is determined by sorting the values by magnitude, and returning the middle one. • If an even number of values is passed, then the smaller of the two middle values is returned.
<p>min(complex a, complex b, ...)</p>	<p>returns the complex value that has the smallest magnitude of a list of values.</p>
<p>min_hold(complex a)</p>	<p>holds the current minimums of the sweep. Disable the equation to reset. See example</p>
<p>mrkx(a,b)</p>	<p>returns the x-axis value of marker number b on trace number a.</p>
<p>mrky(a,b)</p>	<p>returns the y-axis value of marker number b on trace number a.</p>
<p>mu1(complex a, complex b, complex c, complex d) when entered in EE: mu1(S11,S21,S12,S22)</p>	<p> $\mu_1 = (1 - a ^2) / (d - \text{conj}(\mathbf{a}) * (a*d-b*c) + b*c)$ </p>
<p>mu2(complex a,</p>	<p> $\mu_2 = (1 - \mathbf{d} ^2) / (a - \text{conj}(\mathbf{d}) * (a*d-b*c) + b*c)$ </p>

complex b, complex c, complex d) when entered in EE: mu1(S11,S21,S12,S22)	
for both mu1 and mu2 (Usually written with the Greek character μ)	<ul style="list-style-type: none"> • conj is the complex conjugate. For scalars a and b, conj(a+ib) = (a-ib) • returns a scalar result - the imaginary part of the complex result is always 0
phase(complex a)	returns atan2(a) in degrees
PI	returns the numeric constant pi (3.141592), which is the ratio of the circumference of a circle to its diameter
pow(complex a,complex b)	returns a to the power b
re(complex a)	returns the scalar part of a (zeroes the imag part)
sin(complex a)	takes a in radians and returns the sine
sqrt(complex a)	returns the square root of a , with phase angle in the half- open interval (-pi/2, pi/2]
tan(complex a)	takes a in radians and returns the tangent
traceDataArray(complex a)	returns the entire set of points from a sweep. Function is intended to be used as an argument in an custom function to allow access for data array processing.
xAxisArray() or xAxisArray(integer a)	returns the current value of the x-axis for this channel or from a specified channel.
xAxisIndex()	returns the current index in the sweep.
xAxisValue() or xAxisValue(integer a)	returns the current value of the x-axis index for this channel or from a specified channel.

Operators used in Equation Editor

Operator	Description
----------	-------------

+	Addition
-	Subtraction
*	Multiplication
/	Division
(Open parenthesis
)	Close parenthesis
,	Comma - separator for arguments (as in S11, S22)
=	Equal (optional)
E	Exponent (as in 23.45E6)

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**.

$$20\log(a) + 2 = 20\log(x)$$

$$\log(a) + 2/20 = \log(x) \text{ // divide all by 20.}$$

$$x = 10^{(\log(a) + 2/20)} \text{ // swap sides and take 10 to the power of both sides}$$

$$x = 10^{\log(a)} * 10^{(2/20)}$$

$$x = a * 10^{(2/20)}$$

The equation is entered into Tr2 as:

$$\text{Offset} = \text{Tr1} * \text{pow}(10, 2/20)$$

To offset by 5 dB

$$\text{Offset} = \text{Tr1} * \text{pow}(10, 5/20).$$

Balanced Match using a 2-port analyzer

$$\text{SDD11} = (\text{S11} - \text{S21} - \text{S12} + \text{S22})/2$$

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)

$$\text{Tr1}*\text{sqrt}(\text{Tr1}/\text{Tr3})$$

- Tr1 = input signal power
- Tr3 = intermodulation power (both traces measured with single receivers)

Harmonics in dBc

$$B_1/\text{Tr2}$$

- B_1 is tuned to a harmonic frequency
- Tr2 = power at fundamental frequency, measured with B_1 receiver

PAE (Power Added Efficiency)

$$P_{\text{out}} - P_{\text{in}} / P_{\text{dc}}$$

Type the following equation into a new trace with an unratioed measurement, such as AI1. The data format is REAL:

$$\text{PAE} = 100 * (.001*\text{pow}(\text{mag}(\text{Tr1}),2) - (.001*\text{pow}(\text{mag}(\text{Tr1}),2)/\text{pow}(\text{mag}(\text{Tr2}),2)))/(\text{Tr3}*\text{Tr4})$$

Where:

- Tr1 - a trace that measures unratioed B receiver.
- Tr2 - a corrected S21 trace (amplifier gain)
- Tr3 - a trace that measures voltage (AI1) across a sensing resistor.
- Tr4 = an equation trace containing $I_{\text{supp}} = (\text{Tr3} / \text{value of sensing resistor})$.

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 analyzer, 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 analyzer 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 analyzer.
- **CSV** - read by Spreadsheet software. Can NOT be recalled by the analyzer.
- **MDIF** - compatible with Keysight ADS (Advanced Design System). Can NOT be recalled by the analyzer.
- **Print to File** (bmp, jpg, png) - saves an image of the 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.

Import Functions

Several additional functions are provided with the PNA. In addition, you can create custom functions which are compiled into a DLL. Import these functions for use in the Equation Editor.

- [How to Import Functions](#)
- [Supplied User Functions](#)
 - [BestFit.dll](#)
 - [EqnErrorTerms.dll](#)
 - [Expansion.dll](#)

See Also

[Custom Equations In PNA.pdf](#) Detailed directions. (This link requires an internet connection.)

[Create custom functions for Equation Editor](#) Template. (This link requires an internet connection.)

[Equation Editor](#) Main topic.

How to Import Functions

From the main [Equation Editor](#) dialog, click [Import Functions](#)



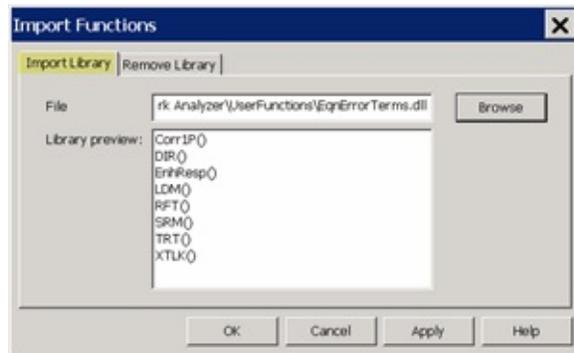
Import Functions dialog box help

Imports and removes libraries that are used with Equation Editor. A library is a *.DLL file that contains one or more functions.

Although not all functions are applicable to all channels or data sets, they will still appear in the "Function/Constants" list.

Once imported, each library is automatically loaded when the PNA application starts. If a function is not found or if an error occurs while loading, the PNA will not attempt to reload the library when starting.

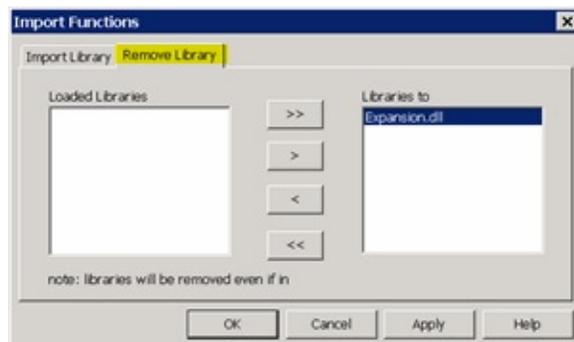
Import Library tab



Browse Click to navigate to the .DLL file on the PNA. The recommended location for the custom equation DLLs is the “C:\Program Files (x86)\Keysight\Network Analyzer\UserFunctions” directory on the PNA.

Library Preview Lists the functions that are contained in the library. Click **OK** or **Apply** to load the library.

Remove Library tab



Left pane Lists the imported libraries. These also appear in the Equation Editor main dialog and remain until removed from the PNA.

Arrows Click the relevant arrows to move some (>) or all (>>) libraries from the PNA.

Right Pane Lists the libraries to remove.

Click **OK** or **Apply** to remove the library.

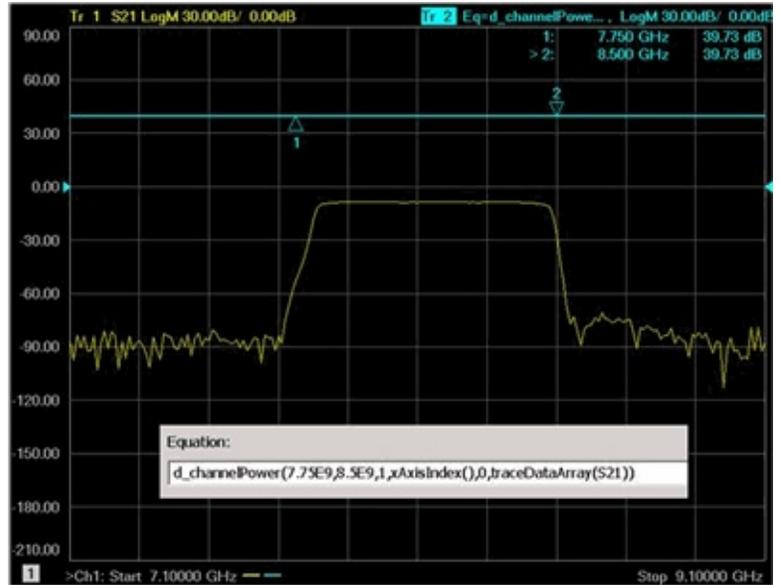
Supplied User Functions

The following functions are supplied with the PNA, but must be imported into Equation Editor. They are available on the PNA at: 'C:\Program Files (x86)\Keysight\Network Analyzer\UserFunctions'.

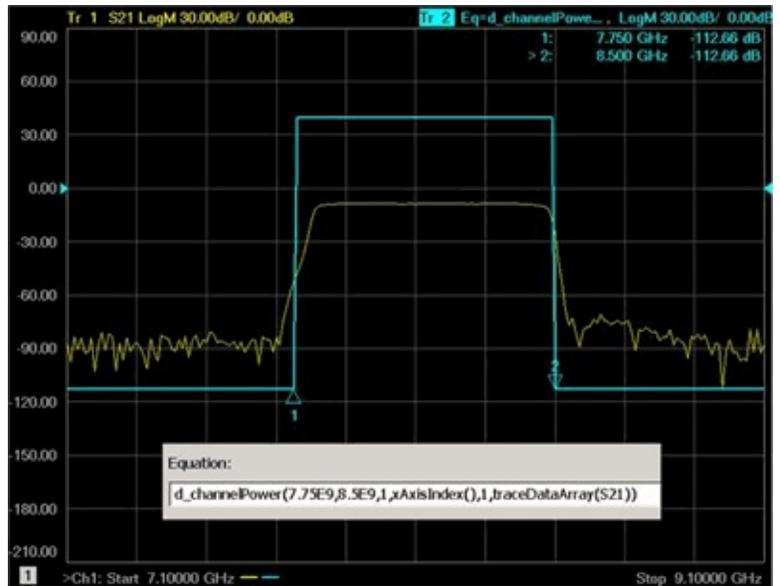
- BestFit.dll
- EqnErrorTerms.dll
- Expansion.dll

BestFit.dll

d_best_fit_dB()	<i>d_best_fit_dB(getNumPoints(),xAxisIndex(),xAxisArray)</i> Draws the best fit linear regression line to data specified (sum-square) of the trace data in log-magnitude format. T
d_channelPower()	<i>d_channelPower(FA,FB,CHANNELNUM,xAxisIndex(</i> Computes the channel power for the specified measurement frequencies. <ul style="list-style-type: none">• FA and FB specify the frequency start/stop values in• CHANNELNUM is the 1-based channel number to• PARAM indicates the measurement to compute cha• TRACETYPE indicates how to display the computed<ul style="list-style-type: none">• If TRACETYPE= 0, the display is a flat line v• if TRACETYPE= 1 (default), the display is se from FA to FB. See examples below.• If CHANNELNUM is hosting an IM Spectrum mea formula: channelPower = 10Log10((area under PARAM t• For all other measurement types, the formula is: channelPower = 10Log10((area under PARAM t <p>TraceType Example 1: FA=7.75 GHz, FB=8.5 GHz, CH power is displayed over the entire frequency range.</p>



TraceType Example 2: As above, but TRACETYPE is 1 outside of specified range.



`d_DFLP()`

d_DFLP(FreqStart, FreqStop, xAxisIndex(), getNumP

Computes the deviation from linear phase data of the spectrum displayed in Phase format, the displayed data will be resampled if the slope has been removed. The FreqStart and FreqStop arguments specify the span for this computation. To cover the channel's entire bandwidth, use a large number such as 1E100.

`d_flatness_dB()`

d_flatness_dB(getNumPoints(),xAxisIndex(),xAxisArra

Computes the magnitude flatness of the trace data in PAF format, normalizing the results to 0 dB.

d_mean()	<p><i>d_mean(traceNum)</i></p> <p>Computes the mean of the specified trace and creates a re point. The traceNum argument should be replaced by an of the trace for which you want to compute the mean. Th S11) or trace names (such as Tr1). The result of this func analysis function.</p>
d_min_max_dev()	<p><i>d_min_max_dev(getNumPoints(),xAxisIndex(),xAxisAr</i></p> <p>Computes the slope and intercept of the line which minim and the data specified by PARAM. This is done in two pl magnitude, and a minimum-deviation line is fitted to it. S another minimum-deviation line is fitted to that. The mag complex-valued trace. The displayed traces represent the best-fit data. This function is typically used when you wi</p>
d_min_max_dev_d2()	<p><i>d_min_max_dev_d2(getNumPoints(),xAxisIndex(),xAxis</i></p> <p>Computes the parameters of the parabola which minimiz data specified by PARAM. This is done in two phases. Fi magnitude, and a minimum-deviation quadratic is fitted t another minimum-deviation quadratic is fitted to that. Th a complex-valued trace. The displayed traces represent th best-fit data. This function is typically used when you wi behavior.</p>
d_min_max_dev_range()	<p><i>d_min_max_dev_range(FSTART,FSTOP,xAxisIndex(),</i></p> <p>Identical to the function d_min_max_dev(), but only ope FSTOP.</p>
d_min_sum_dev()	<p><i>d_min_sum_dev(getNumPoints(),xAxisIndex(),xAxisAr</i></p> <p>Computes the slope and intercept of the line which minim and the data specified by PARAM. This is done in two pl magnitude, and a minimum-deviation line is fitted to it. S another minimum-deviation line is fitted to that. The mag complex-valued trace. The displayed trace represents the outliers than d_min_max_dev().</p>
d_tilt_dB()	<p><i>d_tilt_dB(getNumPoints(),xAxisIndex(),xAxisArray(),tr</i></p> <p>function when displayed in LogMag format is the total d parameter.</p>
d_unwrap()	<p><i>d_unwrap(getNumPoints(),xAxisIndex(),traceDataArra</i></p> <p>The result of the unwrap() function when displayed in Re</p>

specified trace or Parameter in degrees.

EqnErrorTerms.dll

d_Corr1P() ***d_Corr1P(chan, xAxisIndex(),rcvr, src, RAWDATA)***
Computes and displays 1 port corrected data for the trace data supplied in the RAWDATA placeholder.
chan - the channel of interest.
xAxisIndex() - the bucket (data point) number.
rcvr - the port number of the receiver used to acquire the data.
src - the port being driven. The rcvr and src arguments are needed to select the appropriate error terms used in the correction process.
RAWDATA - Select the data to be corrected by substituting in a trace number or parameter name.

d_DIR() ***d_DIR(chanNum, xAxisIndex(), rcvr, src)***
Displays the directivity term from the cal set used by the channel <chanNum>.
Set chanNum to the desired channel.
Set rcvr and src to the port number for the desired directivity term.

d_EnhResp() ***d_EnhResp(chan,xAxisIndex(),rcvr, src,RAWMATCH,RAWGAIN)***
Computes the corrected gain using enhanced response correction techniques. This technique is useful in cases where you want to ignore the output match of a device or the output match cannot be accurately measured. In this the raw input match and gain are supplied to equation (via RAWMATCH and RAWGAIN placeholders). To use this equation select a trace (TR n) or parameter to use in place of the raw match and gain terms.
chan - the channel number
xAxisIndex() - the bucket number (do not modify)
rcv - The port where the data is acquired.
Sr - The port being driven. The src and rcvr ports are required so that the appropriate error terms are used to calculate the result.

d_LDM() ***d_LDM(chanNum, xAxisIndex(), rcvr, src)***
Displays the loadmatch term from the calset used by the channel <chanNum>.

Set chanNum to the desired channel.

Set rcvr to the load port, and src to the source port for the desired load match term.

LDM(ch, xAxisIndex(), 2,1) gives you the match presented by port 2 while driving port 1.

d_RFT()	<p><i>d_RFT(chanNum, xAxisIndex(), rcvr, src)</i></p> <p>Displays the reflection tracking term from the calset used by the channel <chanNum>.</p> <p>Set chanNum to the desired channel.</p> <p>Set rcvr and src to the port number for the desired reflection tracking term.</p>
d_SRM()	<p><i>d_SRM(chanNum, xAxisIndex(), rcvr, src)</i></p> <p>Displays the sourcematch term from the calset used by the channel <chanNum>.</p> <p>Set chanNum to the desired channel.</p> <p>Set rcvr and src to the port number for the desired source match term.</p>
d_TRT()	<p><i>d_TRT(chanNum, xAxisIndex(), rcvr, src)</i></p> <p>Displays the transmission tracking term from the calset used by the channel <chanNum>.</p> <p>Set chanNum to the desired channel.</p> <p>Set rcvr to the receive port and src to the source port such that TRT(ch, xAxisIndex(), 2, 1) gives you the transmission tracking term for the port 2 input receiver driven by port 1, or in other words, the raw S21 tracking term.</p>
d_XTLK()	<p><i>d_XTLK(chanNum, xAxisIndex(), rcvr, src)</i></p> <p>Displays the isolation term from the calset used by the channel <chanNum>.</p> <p>Set chanNum to the desired channel.</p> <p>Set rcvr to the receive port and src to the source port such that XTLK(ch, xAxisIndex(), 2, 1) gives you the isolation term for the port 2 input receiver while port 1 is on.</p>

admittance(x)	Admittance(x) = 1/x Calculate the admittance
max_hold(x)	(KEY, getNumPoints(), xAxisIndex(), PARAM) Shows maximum value of each point
min_hold(x)	(KEY, getNumPoints(), xAxisIndex(), PARAM) Shows minimum value of each point.
PAE(B,S21,AI1,AI2,R,SCALE)	.001 * (B - (B/S21)) / (SCALE^2*AI1(AI1 - AI2)/R) Power Added Efficiency. <ul style="list-style-type: none"> • B - power out • S21 - corrected amplifier gain • AI1 - DC power supply • AI2 - DC power amp • R - resistance • SCALE - scale
reset(x)	x - a number. Resets the max_hold() or min_hold() function. To reset a given max_hold() or min_hold() function, call reset with the same key.
SDD11(S11,S21,S12,S22)	(S11 - S21 - S12 + S22)/2 Differential mode reflection
SDC11(S11,S21,S12,S22)	(S11 - S21 + S12 - S22)/2 C to D mode conversion reflection
SCD11(S11,S21,S12,S22)	(S11 + S21 - S12 - S22)/2 D to C mode conversion reflection
SCC11(S11,S21,S12,S22)	(S21 + S12 + S11 + S22)/2 Common mode reflection

Use the following two equations to display impedance versus frequency.

Replace 'LOAD' with the value for Z0 (usually 50).

Note: You can read out impedance versus time (not using this function) by creating a marker on a **Time Domain** trace, then changing the marker format to R+jX. [Learn how.](#)

$z\text{Reflect}(S_{11}, \text{LOAD})$	$\text{LOAD} * (1 + S_{11}) / (1 - S_{11})$
$z\text{Transfer}(S_{21}, \text{LOAD})$	$2 * \text{LOAD} * (1 - S_{21}) / S_{21}$

Last Modified:

- 13-Mar-2014 Updated folders again.
- 25-Nov-2013 Updated for Win7 folder
- 2-Apr-2013 Added time domain note
- 26-Feb-2013 Edits to user dlls
- 9-Dec-2010 Added equation help and link to DS document.
- 9-Nov-2010 MX New topic

External DC Meter Data Conversion

When creating equations using values from an external DC meter, it is important to understand how these values are stored in the PNA's data buffers and the conversion that occurs when used in an equation. For example, when a voltage is read from an external DC meter, the value is displayed on the PNA as you would expect. That is, if you are reading a voltage level of 2 V from the DC meter in a trace, the PNA will display a level of 2 V. However, the value stored in the PNA data buffers is not a voltage but is a unit-less value. Voltage, Amperes, dBm, and Watts values from an external DC meter are converted so that the format matches that of the data in the PNA internal receivers. In this way, all of the formats within the PNA are the same. This information is important when performing analysis using the Equation Editor because the trace data is the converted value.

See Also

[Equation Editor](#)

[Configure a DC Device](#)

The following table shows the formats (which are selected from the Type setting on the External DC Meter Properties dialog) and corresponding equations that convert between external DC meter readings and the PNA representation when using the trace data in an equation.

Note: Z0 is the characteristic impedance (typically 50 Ohms), dcMeter is the value from the external DC meter, and pnaVal is the value stored in the PNA data buffers. All data types are REAL.

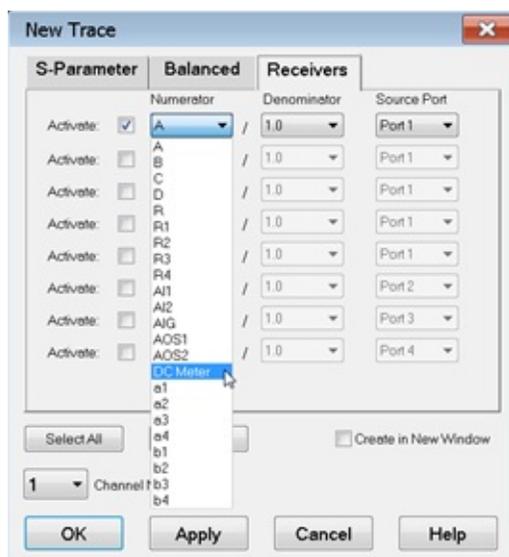
Formats	DC Meter to PNA Data Conversion	PNA to DC Meter Data Conversion
V (volts - default)	+/- $\sqrt{(\text{dcMeter} * \text{dcMeter} / Z0) * 1000}$	+/- $\sqrt{(\text{pnaVal} * \text{pnaVal} / 1000) * Z0}$
A (amperes)	+/- $\sqrt{(\text{dcMeter} * \text{dcMeter} * Z0) * 1000}$	+/- $\sqrt{(\text{pnaVal} * \text{pnaVal} / Z0) / 1000}$
dBm	$\text{pow}(10, \text{dcMeter} / 20)$	$20 * \log(\text{pnaVal})$
W (watts)	$\sqrt{\text{dcMeter} * 1000}$	$\text{pnaVal} * \text{pnaVal} / 1000$

K (kelvin)	N/A	N/A
F (degrees)	N/A	N/A
C (degrees)	N/A	N/A

External DC Meter Voltage Example

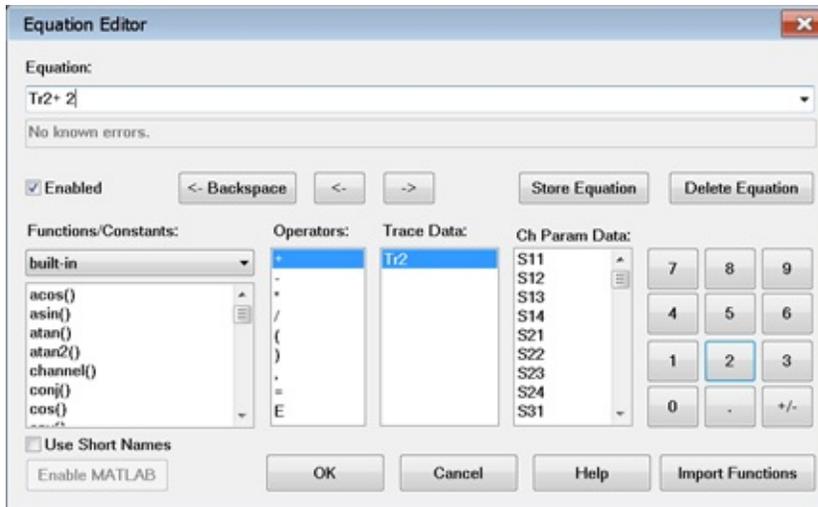
The following example shows how trace data is converted when used in an equation. In this example, a level of 2 V is read from an external DC meter.

1. Configure the external DC meter as described in Configure a DC Device.
2. In the **External DC Meter** dialog, ensure that **Type** is set to **V**.
3. Select **Trace/Chan** then **New Trace**.
4. In the **New Trace** dialog, check **Activate**, click on the corresponding down arrow in the **Numerator** column, select the external DC meter from the drop down list, then click **OK**.

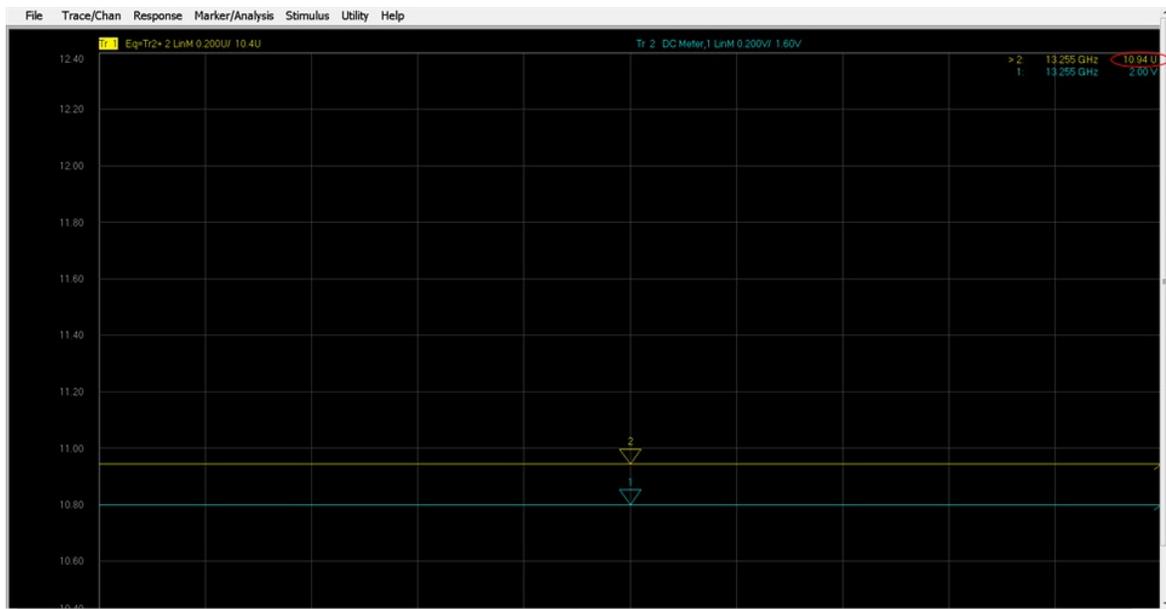


Note: If the external DC meter is not displayed in the list, ensure that **Active - Show in UI** is checked in the **External Device Configuration** dialog.

5. Trace 1 and Trace 2 should now be displayed on the PNA. Add markers to both traces. The Trace 2 marker should read 2.00 V from the external DC meter.
6. Select Trace 1, then select **Response, Format, Lin Mag**.

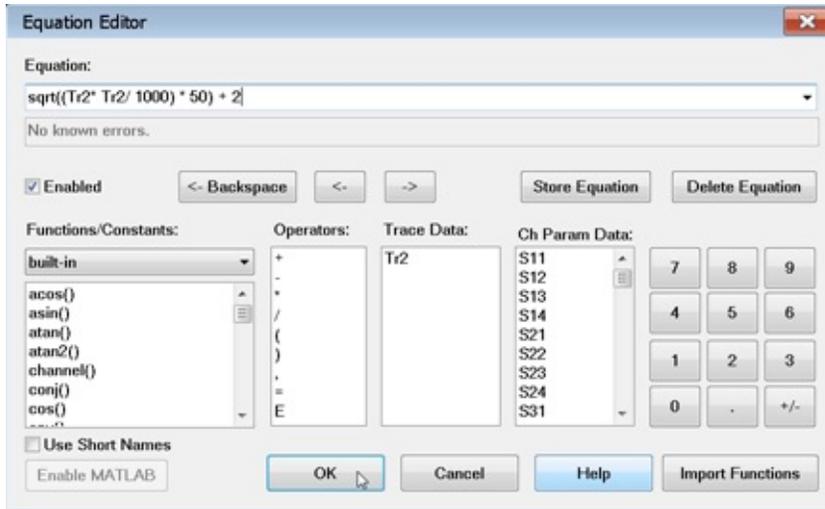


- Note that instead of a voltage level of 4.00 V, the Trace 1 marker reads 10.94 U (unit-less value).

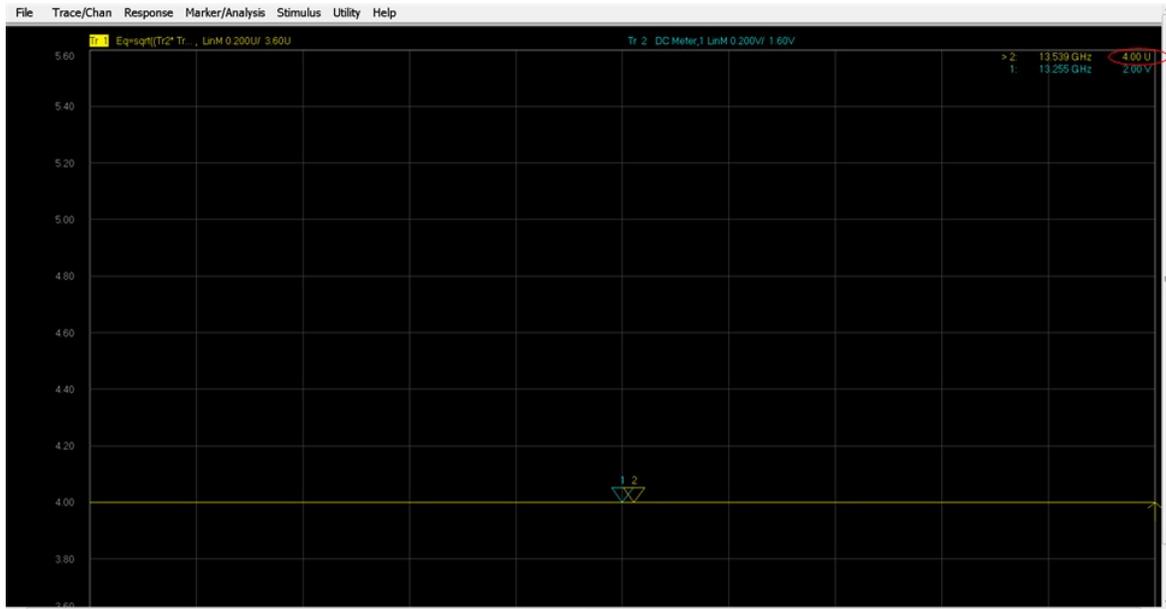


As shown in the table above, a voltage from an external DC meter is converted using $\sqrt{\text{dcMeter} * \text{dcMeter} / Z_0} * 1000$. Therefore, substituting 2 for dcMeter in the equation and using 50 as Z_0 results in a value of 8.94. Adding a value of 2 to the Trace 2 data, as defined in the Trace 1 equation, results in the displayed marker value of 10.94.

- To ensure that the displayed value is 4 instead of 10.94, which is not useful, use the equation from the **PNA to DC Meter Data Conversion** column of the table above as follows:



11. The Trace 1 marker now displays a value of 4.00 U.



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 VNA 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.

By default, limit lines are drawn in the same color as the trace on which they are created. However, all limit lines can be drawn in Red by setting a preference.

[Learn more.](#)

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:

Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none">1. Press ANALYSIS2. then [Limits]3. then [Limit Test]	<ol style="list-style-type: none">1. Click Marker/Analysis2. then Analysis3. then Limit Test



Limit Table

	TYPE	BEGIN STIMULUS	END STIMULUS	BEGIN RESPONSE	END RESPONSE
1	MIN	1.930000 GHz	1.990000 GHz	-5.000000 dB	-5.000000 dB
2	MAX	1.000000 GHz	1.500000 GHz	-60.000000 dB	-50.000000 dB
3	MAX	2.050000 GHz	3.000000 GHz	-50.000000 dB	-60.000000 dB
4	OFF	<input type="text" value="0.0Hz"/>	0.000000 Hz	0.000000 dB	0.000000 dB

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.

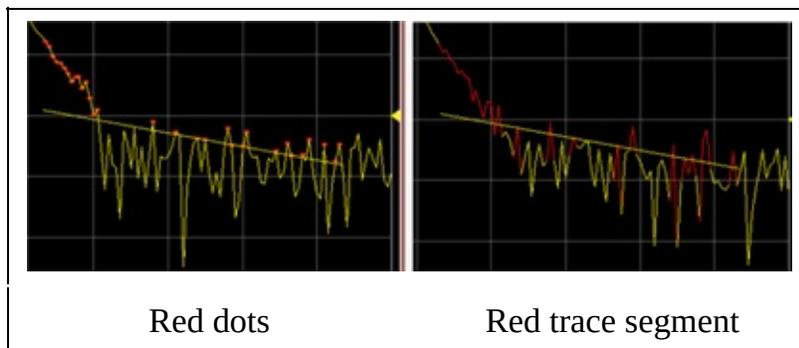
You can choose to provide a visual and / or audible PASS / FAIL indication.

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

You can display the data points that fail limit line testing as red dots or as a red trace segment. The default behavior (red trace) can be changed with a Preference setting. [Learn how.](#)

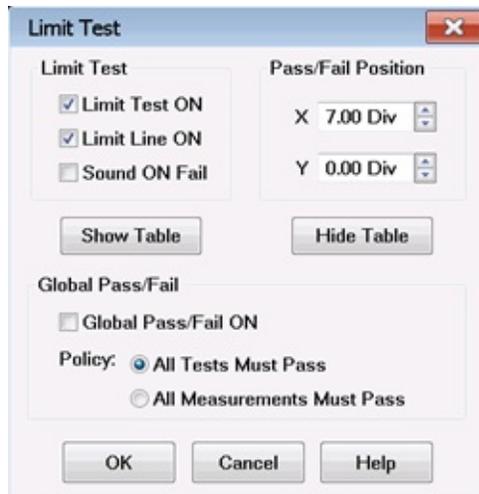


PASS is the default mode of Pass / Fail testing.

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



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.

Pass/Fail Position

Sets the position of the Limit Line Pass/Fail status indicator on the VNA screen.

X - X-axis position. 0 is far left; 10 is far right.

Y - X-axis position. 0 is bottom; 10 is top.

Show Table Shows the table that allows you to create and edit limits.

Hide Table Makes the limits table disappear from the screen.

Global Pass/Fail

The Pass/Fail indicator provides an easy way to monitor the status of ALL measurements.

Global Pass/Fail 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

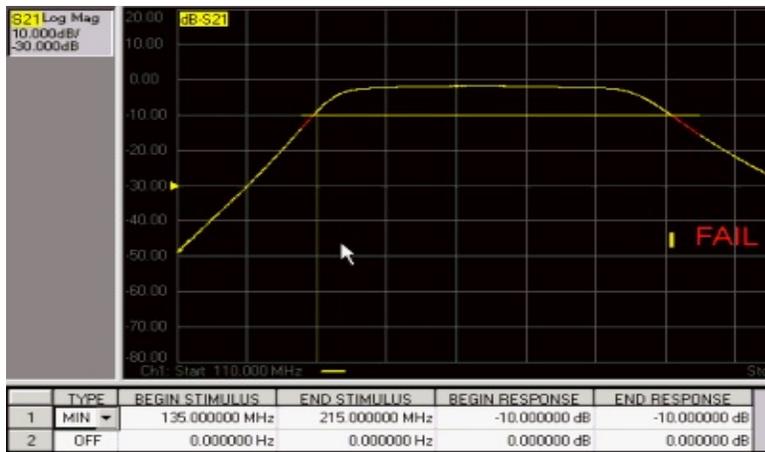
Limits are checked only at the actual measured data points. Therefore, It is possible for a device to be out of specification without a limit test failure indication if the data point density is insufficient.

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.



Last Modified:

- 14-Mar-2013 Edited default fail type
- 11-Aug-2009 Added failed dots
- 3-Sep-2008 Removed legacy content
- 2-Feb-2007 MX Added UI

Save and Recall a File

You can save and recall files to and from an internal or external storage device in a variety of file formats.

- [How to Save Instrument State](#)
- [How to Save 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)

Other Data Outputting topics

How to Save Instrument State and Calibration Files

Use one of the following methods:

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

1. Press **SAVE**
2. then **[Save], [Save As], or [Auto Save]**

Using Menus

1. Click **File**
2. then **Save** or **Save As**

[Learn all about Instrument State files.](#)

Save Immediately saves the instrument 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 Menu. Saves state and calibration data to the internal hard disk in the c:\users\public\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.

Note: Filenames (not including the path name) **MUST** be limited to 64 characters.

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 analyzer. These file types are only recognized by Keysight VNA 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 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 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 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.

Note: Filenames (not including the path name) **MUST** be limited to 64 characters.

Save as type Choose from: (click each to learn more about each file type): ***.prn**, ***.SNP**, ***.SNPX**, ***.cti (citifile)**, ***.csv**, ***.mdf**.

- To save the screen as .bmp, .jpg, or .png graphics file types, click **File / Print to File**. [Learn more](#).
- Save Uncertainty (Opt 015) data (*.u*p, *.dsd, *.sdatcv). [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.
- **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.

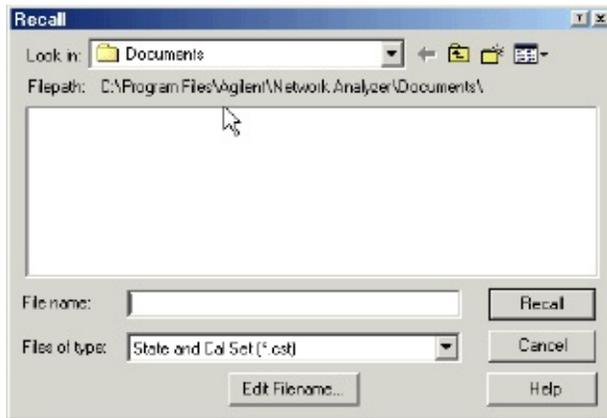
Save Saves the file to the specified file name and directory.

How to Recall (open) a file

Use one of the following methods:

Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press RECALL 2. then [Recall] 3. Select from a list of 12 files shown on softkeys. The list can be sorted by 'most recently used' or alphabetically depending on a preference. The preference setting appears at the bottom of the second page of softkeys listing files to be recalled, or on the Preference dialog. 4. or press Recall to start the following dialog. 	<ol style="list-style-type: none"> 1. Click File 2. then Recall 3. Select from the top 6 'most recently used' list. 4. or click Recall to start the following dialog.

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 analyzer: All **State files**, Citi files, SNP files.

Recalling instrument state files

When an Instrument State file is recalled, 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

Citi files and SNP files can be recalled and viewed in the analyzer.

1. Click **File** then **Recall**.
2. Select **Citifile Data** or **Snp**.
3. Select the file to recall
4. Click **Recall**.

Note: Filenames (not including the path name) that are longer than 64 characters will NOT be recalled.

Recalled data is ALWAYS displayed 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 analyzer**. 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 analyzer** 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**.

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 analyzer.

An **Instrument State** contains almost every analyzer setting. The following settings are NOT saved and recalled with Instrument State:

- **GPIB address**
- **RF power ON/OFF** (depends on current setting)

The following file types are used to save and recall instrument states and Cal Set information:

File Types		Information that is stored for each channel		
.csa	.cst	.sta	Instrument State Information	
			Channels/Traces	Averaging
			Windows	Markers
			Triggering	Math/memory
			Format	Limits
			Scale	More...

				Stimulus Information:	
				Frequency range	Alternate sweep
				Number of points	Port powers
				IF bandwidth	Source attenuators
				Sweep type	Receiver attenuators
				Sweep mode	Test Set port map
				Cal Set Information	
				GUID (Globally Unique Identifier) provides link to Cal Set	
				Name, Description, Modify date	
				Stimulus Information:	
				Frequency range	Alternate sweep
				Number of points	Port powers
				IF bandwidth	Source attenuators
				Sweep type	Receiver attenuators
				Sweep mode	Test Set port map
				Error Terms: Directivity, Crosstalk, Source match, Load match, Reflection tracking, Transmission tracking	

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 analyzer attempts 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](#)

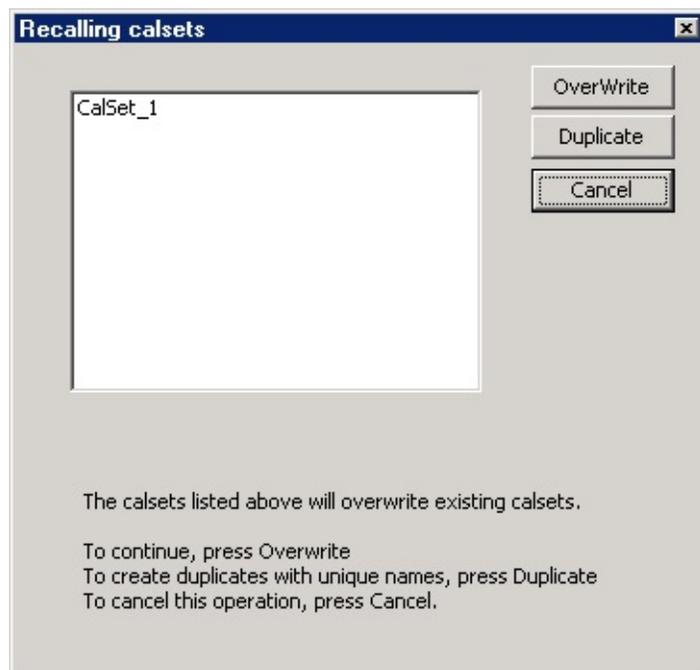
***.csa files**

- 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 used for storing cal sets. These files should never be accessed or copied by the user.

Recalling Cal Sets

Recalling Cal Sets dialog box help



Both **.cal** and **.csa** file types contain whole Cal Sets. Because all channels are saved, there could be more than one Cal Set in either of these file types.

When these file types are recalled, a check is made to see if the incoming Cal Set GUID matches an existing 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 analyzer. You are then offered the following choices:

- **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.

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 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 analyzer:

- *.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 analyzer.
- **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)

- *.SNP file format, also known as Touchstone format, is specified by IBIS. [See the Touchstone specification.](#)
- *.SNP file format is used by CAE programs such as Keysight's Microwave Design System (MDS) and Advanced Design System (ADS).
- *.SNP data is saved using the **File, Save Data As** dialog.

Before saving measurement data, always **trigger a single** measurement, and then allow the channel to go into Hold. This ensures that the entire measurement trace is saved.

*.SNP files and other analyzer settings

- .SNP data can be **recalled** and viewed on the analyzer, or read by the **embed/de-embed** functions.
- When **Fixturing** is enabled, all of the enabled data transforms (De-embedding, Port Z Conversion, and so forth) are applied to saved SNP files.
- When **Smoothing** is applied to a trace, the smoothing is NOT saved when the format is Real, Imaginary (RI). Select a different format to save the smoothed data.
- Balanced parameters can be saved to *.SNP files. See the **"Choose Ports "** dialog.
- **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.

What is Saved

*.SNP data is generally used to gather all S-parameters for a fully corrected measurement.

The analyzer saves the data that is available on the channel of the active

measurement.

File Type	# of Ports	# of S-parameters saved
*.s1p	1	1 S-parameter
*.s2p	2	4 S-parameters
*.s3p	3	9 S-parameters
*.s4p	4	16 S-parameters
...
*.SNP	N	N^2 S-parameters

- If correction for a **Full N-port cal** is applied, then valid data is returned for all corrected s-parameters. Response cals will save uncorrected data.
- If requesting **less** data than is available, the **Choose ports for SNP data** dialog appears.
- If correction is NOT applied, the analyzer 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.

.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**:

```
!Keysight Technologies,E8362B,US42340026,Q.03.54
!Keysight 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(S22) Imag(S22)

***.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) Real(S23) Imag(S23)
Real (S31) Imag(S31) Real(S32) Imag(S32) Real(S33) Imag(S33)

***.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)
Real (S21) Imag(S21) Real(S22) Imag(S22) Real(S23) Imag(S23) Real(S24) Imag(S24)
Real (S31) Imag(S31) Real(S32) Imag(S32) Real(S33) Imag(S33) Real(S34) Imag(S34)
Real (S41) Imag(S41) Real(S42) Imag(S42) Real(S43) Imag(S43) Real(S44) Imag(S44)

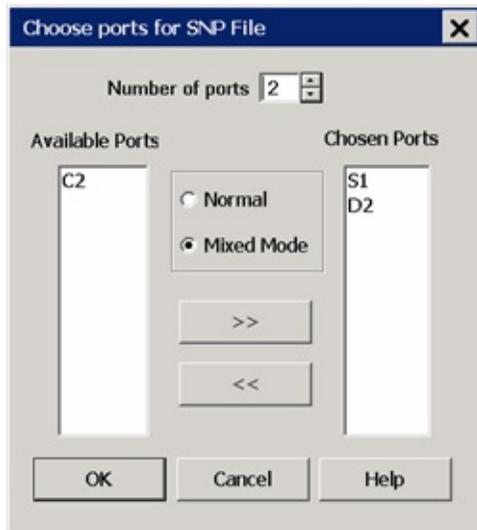
.S2PX Data Output

*.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 appears when any of the following conditions exist while attempting to save data to an *.snp file:

- 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 **ONLY** 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 or Remove ports from or to the following columns:

Available Ports All test set ports are listed. 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 Keysight 8510 Network Analyzer and Keysight'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)**

```
CITIFILE A.01.01
!Agilent Technologies, E83
!Agilent E8364B: A.07.50.
!Format: LogMag/Angle
!Date: Thursday, April 17
NAME CH1_DATA
VAR Freq MAG 3
DATA S11 DBANGLE
VAR_LIST_BEGIN
10000000
25005000000
50000000000
VAR_LIST_END
BEGIN
-14.837049,-54.52504
12.872375,105.69022
12.423248,-142.31508
END
CITIFILE A.01.01
!Agilent Technologies, E83
!Agilent E8364B: A.07.50.
!Format: LinMag/Angle
!Date: Thursday, April 17
NAME CH2_DATA
VAR Freq MAG 3
DATA SC21 MAGANGLE
VAR_LIST_BEGIN
10000000
25005000000
50000000000
VAR_LIST_END
BEGIN
0.00012980904,85.130768
0.0006259337,-56.60146
0.0062053087,-99.910851
END
```

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 analyzer. [Learn more.](#)

*.csv Files

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/Angle
!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 Keysight ADS (Advanced Design System).
[Learn more at the Keysight website.](#)

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

```
!MDF A.01.01
!Agilent Technologies, N5242A, US473000004, A.08.20.01
!Format: LogMag/Angle
!Date: Thursday, October 23, 2008 11:14:01

BEGIN CH1_DATA
%Freq (real) S[1,1] (complex) S[1,2] (complex) S[2,1] (complex) S[2,2] (complex)
1e+009 0.042540751 -0.024109257 0.56931657 0.042540751 -
0.024109257 0.56931657
...
...
...
END

BEGIN CH2_DATA
%Freq (Hz) S[1,1] (complex) S[1,2] (complex) S[2,1] (complex) S[2,2] (complex)
1e+009 0.042540751 -0.024109257 0.56931657 0.042540751 -
0.024109257 0.56931657
...
...
...
END
```

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

Using Menus

1. Press **SYSTEM**
2. then **[Configure]**
3. then **[More]**
4. then **[Preferences]**
5. then **Data Saves**

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **More**
5. then **Preferences**
6. then **Data Saves**

Programming Commands

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 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)

[Learn more about SnP files.](#)

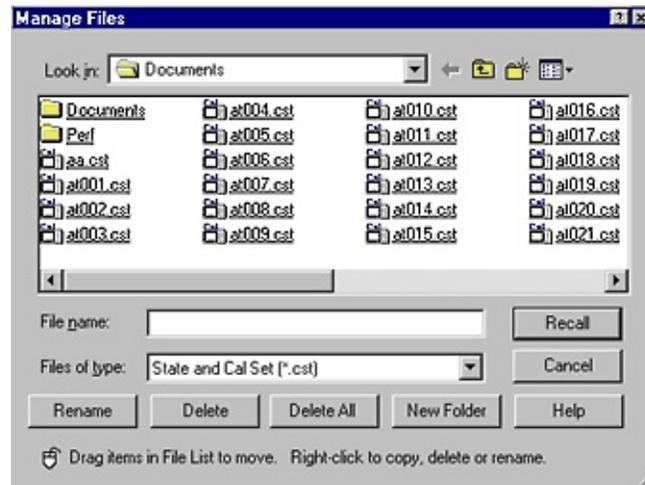
Auto - Data is saved in LogMag or LinMag if one of these is the currently selected format. If format is other than these, then data is saved in Real/Imag.

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.

Programming Commands

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.

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.

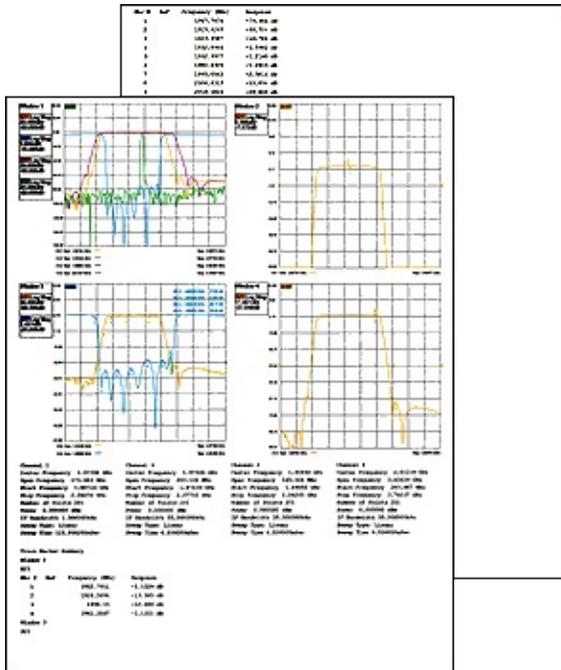
For more information, refer to Microsoft Windows Help or your printer documentation.

Printing

- [Print a Hardcopy](#)
- [Page Setup](#)
- [Print to File](#)

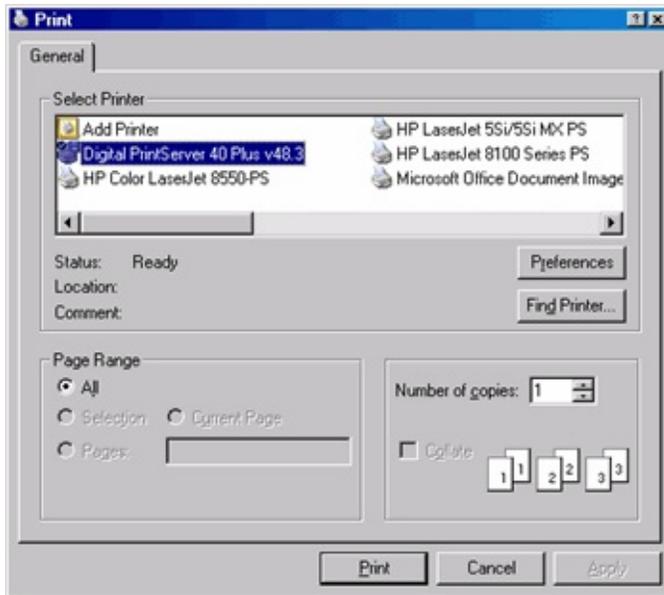
The measurement information on the screen can be printed to any local or networked printer that is connected to the PNA. The graphic below shows an

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

Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press PRINT 2. then [Print] 	<ol style="list-style-type: none"> 1. Click File 2. then Print 3. then Print
<p>No programming commands are available for this feature.</p>	

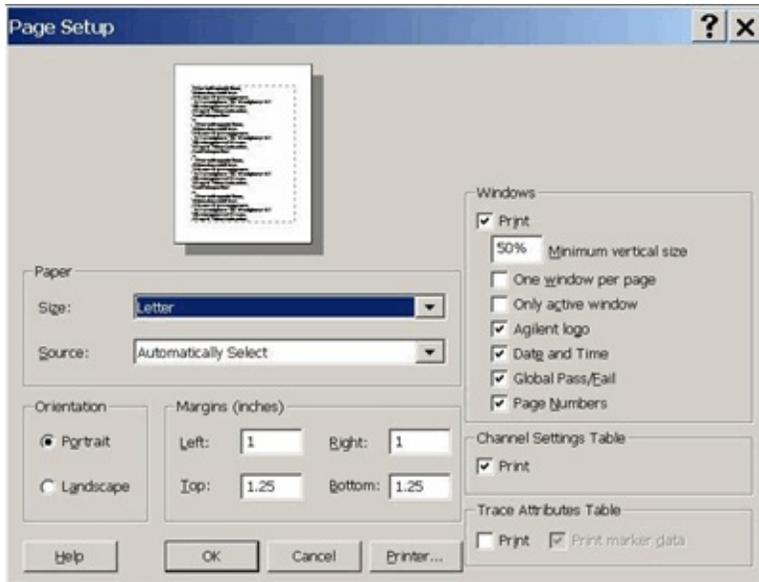


Note: For information on the choices in the Print dialog box, see Windows Help.

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	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press PRINT 2. then [Page Setup] 	<ol style="list-style-type: none"> 1. Click File 2. then Print 3. then Print Options



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.

Keysight logo Check to print the Keysight 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:\users\public\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	Using Menus
<ol style="list-style-type: none">1. Press PRINT2. then [Print to File]	<ol style="list-style-type: none">1. Click File2. then Print3. then Print to File

Last modified:

25-Nov-2013	Default folder for Win7
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

Programming Guide

Two ways to find programming commands:

1. From a simulated User Interface:

[File](#) [Trace/Chan](#) [Response](#) [Cal](#) [Marker/Analy](#) [Stimulus](#) [Utility](#)

	GPIB / SCPI
2.	Command Tree
See Also	Example Programs
	Learning about GPIB

See Also

- **Important** [Programming the M937xA](#)
- [New PXI Programming Commands](#)
- [Remotely Specifying a Source Port](#)
- [Your Programs on Windows 7](#)
- [Using Macros](#)
- [Data Access Map](#)

SCPI Command Tree

see Also

- [Example Programs](#)
- [Find commands using a simulated PNA UI](#)
- [See list of all SCPI Errors.](#)
- [See Calibrating the PNA Using SCPI](#)
- [Synchronizing the Analyzer 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
DATA	Sends and queries data.
EQUation	Equation Editor
FILTer	Time domain gating
FORMat	Display format
FSIMulator	Balanced measurements and Fixturing
FUNcTion	Trace Statistics
HOLD	Trace Hold
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:AXIS	X-Axis Domain
CALPod	Controls CalPod units
CONTRol	Interface control, ECal module state control, and Rear-panel connector control.
CSET	Work with a Cal Set without having to select it into that channel.
DISPlay	Display settings
FORMat	Format for data transfer
HCOPy	Hardcopy printing
INITiate	Continuous or manual triggering
LXI	LXI communications
MMEMory	Saves and recalls instrument states
OUTPut	Turns RF power ON and OFF
+ SENSE	Click to hide and show SENSE branches
AVERage	Sweep Averaging
BANDwidth	IF Bandwidth
CORRection	Calibration and other correction settings
CKIT	Manage Cal Kits and ECal modules
COLL:CKIT	Edit Cal Kit definitions
COLL:GUIDed	Perform Guided Cals
CSET	Manage Cal Sets
EXTension	Port Extensions
COUple	Chopped or Alternate sweep
FREQuency	Frequency sweep settings
ROSCillator	Returns the source of the reference oscillator
SEGment	Segment sweep settings
SWEep	Sweep types
TEMPerature	Returns the temperature on the receiver board
X:VALues	Returns X-axis values for channel
SOURce	Source power to the DUT

POWer:CORR	Source power calibration
STATus	Reads the PNA status registers
SYSTem	Misc PNA capabilities
CAL:All	Calibrate All Channels
	PXI module capabilities
CAP:HARD:MODule	
COMMunicate	Controls and queries PNA settings
DATA:MEMory	Shared memory commands
PREFerences	VNA Preferences
SERV:PVER	Instrument Cal dates
TRIGger	Trigger measurements

IEEE 488.2 Common Commands

- ***CLS** - Clear Status
- ***ESE** - Event Status Enable
- ***ESE?** - Event Status Enable Query
- ***ESR?** - Event Status Enable Register
- ***IDN?** - Identify
- ***OPC** - Operation complete command
- ***OPC?** - Operation complete query
- ***OPT?** - Identify Options Query
- ***RST** - Reset
- ***SRE** - Service Request Enable
- ***SRE?** - Service Request Enable Query
- ***STB?** - Status Byte Query
- ***TST?** - Result of Self-test Query
- ***WAI** - Wait

see Also

- [Example Programs](#)
- [Synchronizing the Analyzer and Controller](#)
- [SCPI Command Tree](#)

***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](#).

***IDN? - Identify**

Returns a string that uniquely identifies the analyzer. The string is of the form "Keysight 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 comma-separated string identifying the analyzer option configuration. [See a list of VNA options](#).

***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 hardware 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 exist.

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
0	1	Phase Unlock	The source has lost phaselock. This could be caused by a reference channel open or a hardware failure.
1	2	Unleveled	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.
2	4	Not used	
3	8	EE Write Failed	An attempted write to the EEPROM has failed. This is possibly caused by a hardware failure.
4	16	YIG Cal Failed	The analyzer was unable to calibrate the YIG. Either the phaselock has been lost or there has been a hardware failure.
5	32	Ramp Cal Failed	The analyzer was unable to calibrate the analog ramp generator due to a possible hardware failure.
6	64	Not used	

***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 **INITtiate: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 Analyzer and Controller](#)

Examples `ABOR`
`abort`

Query Syntax Not applicable

Default Not applicable

Calculate:Correction Commands

Controls error correction functions.

CALCulate:CORRection

EDELay

| **DISTance**

| **TIME**

| **MEDium**

| **UNIT**

| **WGCutoff**

[STATe]

| **INDicator?**

TYPE

OFFSet

| **[MAGNitude]**

| **PHASe**

Click on a keyword to view the command details.

Blue keywords are superseded.

See Also

- [Example Programs](#)
- [Calibrating with SCPI](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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: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, **WAVEguide** 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<cnum>:CORRection:EDELay:WGCutoff <num>

(**Read-Write**) Sets the waveguide cutoff frequency used when the electrical delay media is set to WAVEguide. (See [CALCulate:CORRection:EDELay:MEDIum <char>](#).)

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1.

<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<cnum>:CORRection:EDELay:WGCutoff?

Return Type Numeric

Default 45 MHz

CALCulate<cnum>:CORRection[:STATe] <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

<cnum> Any existing channel number. If unspecified, value is set to 1

<bool> Correction state. Choose from:

0 - Correction OFF

1 - Correction ON

Examples

```
CALC:CORR ON
```

```
calculate:correction:state off
```

Query Syntax CALCulate<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

```
CALC:CORR:IND?
```

```
calculate2:correction:state:indicator?
```

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 Unguided 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 (NO Power Cal included):

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 4 Port(1,2,3,4)"
```

For Full Calibrations (including Power Cal):

After the Full <n> port, include the string, "with power"

For example:

```
"Full 4 Port with power(1,2,3,4)"
```

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 Enhanced Response Calibrations:

"EnhancedResp(sourcePort, recPort)

Where:

- sourcePort = stimulus port number
- recPort = receiver port number

For FCA Calibrations:

Learn more about this setting.

- **"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 unratioed 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?

Return Type 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:Data Commands

Controls writing and reading PNA measurement data.

```
CALCulate:DATA
  | CUSTom
    | CATalog?
  | SNP?
    | PORTs?
      | SAVE
```

Click on a **red** keyword to view the command details.

Red is a superseded command.

See Also

- [Example Programs](#)
- [Data Access Map](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

(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, RDATA, 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
- Use **FORMat:DATA** to change the data type (<REAL,32>, <REAL,64> or <ASCIi,0>).
- Use **FORMat:BORDER** to change the byte order. Use "NORMal" when transferring a binary block from LabView or Vee. For other programming languages, you may need to "SWAP" the byte order.

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.
- RDATA** 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 (Same behavior as SDATA).
- Reads** data from **Data Access Map** location **Raw Measurement** (access point 0).
- Returns TWO numbers per data point.
 - Returned numbers are uncorrected (regardless of correction state)
- 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...
	SCORR1	Forward Directivity
	SCORR2	Forward Source Match
	SCORR3	Forward Reflection Tracking
	SCORR4	Forward Isolation
	SCORR5	Forward Load Match
	SCORR6	Forward Transmission Tracking
	SCORR7	Reverse Directivity
	SCORR8	Reverse Source Match
	SCORR9	Reverse Reflection Tracking
	SCORR10	Reverse Isolation
	SCORR11	Reverse Load Match
	SCORR12	Reverse Transmission Tracking

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**

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <name> Name of the buffer to be read or written
- <data> Data to be read or written to the custom buffer. Format as one number per data point.

Examples

```
CALC:DATA:CUST 'VectorResult0',0,1,2,3,4,5 'Write  
CALC:DATA:CUST? 'VectorResult0' 'Read
```

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](#)

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:DATA:CUST:CAT?  
calculate:data:custom:catalog?
```

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.](#)

This command is valid **ONLY** with standard S-parameter measurements.

Notes

- 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.
- To avoid frequency rounding errors, specify **FORM:DATA** <Real,64> or <ASCIi, 0>

[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 1-Port data for the active measurement if the active measurement is a reflection parameter such as S11 or S22. The behavior is UNDEFINED if the active measurement is a transmission parameter such as an S21.

2 (S2P) returns data for the four 2 port parameters associated with the current measurement. Default. Data that is not available is zero-filled.

3 (S3P) returns data for the nine 3 port parameters associated with the current measurement. Data that is not available is zero-filled.

4 (S4P) returns data for the sixteen 4 port parameters associated with the current measurement. Data that is not available is zero-filled.

SnP data can be output using several data formatting options. See [MMEM:STORe:TRACe:FORMat:SNP](#).

See also [MMEM:STOR <file>.<snp>](#)

Examples `CALC:PAR:DEF MyMeasurement, S11`
`CALC:PAR:SEL MyMeasurement`
`CALC:DATA:SNP? 1`

Return Type Depends on [FORMat:DATA](#).

Default Not Applicable

CALCulate<cnum>:DATA:SNP:PORTs? <"x,y,z">

Note: This command replaces **CALC:DATA:SNP?**. 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.](#)

This command is valid **ONLY** with standard S-parameter measurements.

Notes

- 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.
- To avoid frequency rounding errors, specify **FORM:DATA** <Real,64> or <ASCIi, 0>
- Data that is not available is zero-filled.
- For sweeps with a large number of data points, always follow this command with *OPC? [Learn more.](#)

[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.

<"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 [MMEM:STORe:TRACe:FORMat:SNP](#).

Examples **CALC:DATA:SNP:PORTs? "1,2,4,5,7" 'read data for these ports**

Return Type Depends on **FORMat:DATA**

Default Not Applicable

CALCulate<cnum>:DATA:SNP:PORTs:SAVE <"x,y,z">, <filename>

Note: This command replaces **MMEM:STOR 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.](#)

- The Normal vs Mixed Mode selection is NOT used as it is in the **Choose Ports dialog**. Instead, data is returned as it is displayed on the trace. If the selected measurement is Mixed Mode (balanced), then balanced data is returned. If the selected measurement is an S-parameter, then S-parameter data is returned.
- This command is valid **ONLY** with the Standard measurement class (NOT applications).
- Data that is not available is zero-filled.
- For sweeps with a large number of data points, always follow this command with *OPC? [Learn more.](#)

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.
- <"x,y,z"> **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, enclosed in quotes. 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:\users\public\network analyzer\documents\MyData.s3p';*OPC?`

Return Type Depends on [FORMat:DATA](#)

Default Not Applicable

Calculate:Equation Commands

Controls Equation Editor capabilities.

CALCulate:EQUation:

LIBRary

| **FUNCTIONS**

| **IMPort?**

| **REMove**

STATe

TEXT

VALid?

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Equation Editor](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate:EQUation:LIBRary:FUNCTIONS <string>

(Read-only) Returns the functions in the specified DLL.

Parameters

<string> Full path and filename of the *.dll to be read.

Examples

```
functions = CALC:EQU:LIBR:FUNC "C:/Program  
Files/Keysight/Network Analyzer/UserFunctions/Expansion.dll"
```

Query Syntax CALCulate:EQUation:LIBRary:FUNctions?

Return Type Comma delimited string of function names.

Default Not Applicable

CALCulate:EQUation:LIBRary:IMPort <string>

(Read-Write) Imports the functions in the specified DLL and returns whether the functions have been imported into the PNA.

Parameters

<string> Full path and filename of the *.dll.

Examples

'Write - Imports functions

```
CALC:EQU:LIBR:IMPort "C:/Program  
Files/Keysight/Network Analyzer/UserFunctions/Expansion.dll"
```

'Read if Imported

```
functions = CALC:EQU:LIBR:IMPort "C:/Program  
Files/Keysight/Network Analyzer/UserFunctions/Expansion.dll"
```

Query Syntax CALCulate:EQUation:LIBRary:IMPort?

Returns the following:

1 - Imported

0 - NOT imported

Return Type Boolean

Default Not Applicable

CALCulate:EQUation:LIBRary:REMOve <string>

(Write-only) Removes an imported an Equation Editor DLL from the PNA.

Parameters

<string> Full path and filename of the *.dll.

Examples `CALC:EQU:LIBR:REM "C:/Program Files/Keysight/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>

(Read-Write) Specifies an equation or expression to be used on the selected measurement for the specified channel.

See [Critical Note](#)

Parameters

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> 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<num>:EQUation:TEXT?

Return Type String

Default Not Applicable

CALCulate<num>:EQUation:VALid?

(Read-Only) 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

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> 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

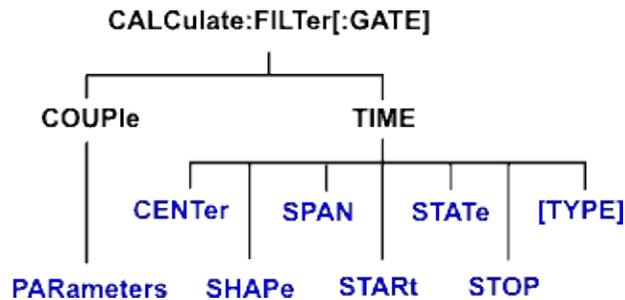
Last Modified:

30-Jan-2014 Several edits to cnums

10-Jan-2011 New topic

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.



Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Gating](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>:FILTer[:GATE]:COUPle:PARAmeters <num>

(Read-Write) 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)

CALCulate<cnum>:FILTer[:GATE]:TIME:CENTer <num>

(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:
 $\pm (\text{number of points}-1) / \text{frequency span}$

Note: This command will accept **MIN** or **MAX** instead of a numeric parameter. See [SCPI Syntax](#) for more information.

Examples

```
CALC:FILT:GATE:TIME:CENT -5 ns  
calculate2:filter:time:center maximum
```

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:SHAP MAX  
calculate2:filter:time:shape normal
```

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:
 $\pm (\text{number of points}-1) / \text{frequency span}$

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:
 $\pm (\text{number of points}-1) / \text{frequency span}$

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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

See Also

- [Example](#) using this command.
 - [Learn About Data Format](#)
 - [Synchronizing the Analyzer 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
- KELVin
- FAHRenheit
- CELSius

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 from:

- **MLOG** - Log magnitude
- **MLIN** - Linear magnitude

<units> For unratiod MLOG 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
- **DBMA** Units are displayed in dBmA. 0 dBmA = 0.001 Ampere

For unratiod MLIN measurements, choose from:

- **W** -Units are displayed in Watts
- **V** -Units are displayed in Volts
- **A** -Units are displayed in Amperes

Examples

```
CALC:FORM MLOG, DBM  
calculate2:format mlog,dbmv
```

Query Syntax CALCulate<cnum>:FORMat:UNIT? <dataFormat>

Return Type Character

Default MLOG, DBM

Last Modified:

28-July-2014 Added MLIN

25-Aug-2008 Added Units

1-Oct-2007 Added temperature formats

Calculate:FSimulator Commands

Specifies settings and fixturing for Balanced Measurements.

CALCulate:FSIMulator

BALun [More commands](#)

EMBed [More commands](#)

GLOop [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

```
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

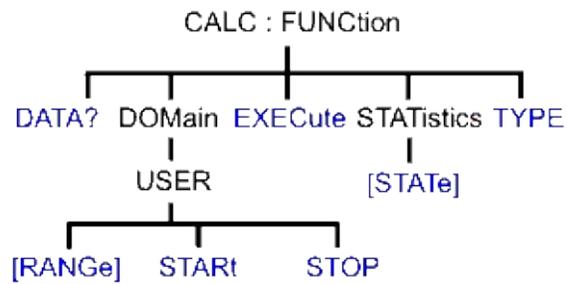
Default OFF

Last Modified:

4-Nov-2013 Fixed link

16-Nov-2010 Added extrapolate

Calculate:Function Commands



Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Trace Statistics](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>: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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

Return Type Depends on **FORM:DATA**

Example

```
CALCulate2:FUNCtion:DATA?
```

Default Not applicable

CALCulate<cnum>: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

```
CALC:FUNC:DOM:USER 4  
calculate2:function:domain:user:range 0
```

Query Syntax CALCulate<cnum>:FUNCTion:DOMain:USER[:RANGe]?

Return Type Numeric

Default 0 - Full Span

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

```
CALC:FUNC:DOM:USER:STAR 1,1e9  
calculate2:function:domain:user:start 2,2e9
```

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:function: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 `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.

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:

PTPeak - the difference between the max and min data points on the trace.

STDEV - standard deviation of all data points on the trace

MEAN - mean (average) of all data points on the trace

MIN - lowest data point on the trace

MAX - highest data point on the trace

Examples `CALC:FUNc:TYPE PTP`
`calculate2:function:type stdev`

Query Syntax `CALCulate<cnum>:FUNCtion:TYPE?`

Return Type Character

Default PTPeak

Group Delay Aperture Commands

Controls the Aperture setting used to make Group Delay measurements.

CALCulate:GDElay FREQuency PERCent POINts
--

Click on a keyword to view the command details.

see Also

- [Learn about Group Delay Aperture](#)
- [Example Programs](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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<cnum>: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<cnum>: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<cnum>:GDElay:POINts?

Return Type Numeric

Default 11 points. This can be changed to two points with a [preference setting](#).

Last Modified:

23-Feb-2010 MX New topic

Calc:Limit Commands

Controls the limit segments used for pass / fail testing.

CALCulate:LIMit:

DATA

| **DELeTe**

DISPlay

| **[STATe]**

FAIL?

SEGMENT

| **AMPLitude**

| **START**

| **STOP**

| **COUNT?**

| **STIMulus**

| **START**

| **STOP**

| **TYPE**

SOUND

| **[STATe]**

[STATe]

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Limit Lines](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more.](#)

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

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

Examples CALC2:LIM:DATA:DEL

Query Syntax Not Applicable

Default Not Applicable

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

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 the display of limit segments ON.
OFF (or 0) - turns the display of limit segments OFF.

Examples

```
CALC:LIM:DISP:STAT ON  
calculate2:limit:display:state off
```

Query Syntax CALCulate<cnum>:LIMit:DISPlay[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> 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<cnum>:LIMit:SEGMent<snum>:AMPLitude:STARt <num>

(Read-Write) Sets the start (beginning) 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:SEGM1:AMPL:STAR 10  
calculate2:limit:segment2:amplitude:start 10
```

Query Syntax CALCulate<cnum>:LIMit:SEGMent<snum>AMPLitude:STARt?

Return Type Numeric

Default 0

CALCulate<cnum>:LIMit:SEGMent<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:SEGM1:AMPL:STOP 10  
calculate2:limit:segment2:amplitude:stop 10
```

Query Syntax CALCulate<cnum>:LIMit:SEGMent<snum>AMPLitude:STOP?

Return Type Numeric

Default 0

CALCulate<cnum>:LIMit:SEGMent:COUNt?

(Read-only) Returns the number of segments used in a limit test. All segments are counted, whether they are on or not.

Parameters Not Applicable

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

Examples CALC:LIM:SEGM:COUN?

Return Type Numeric

Default Not Applicable

CALCulate<cnum>:LIMit:SEGMent<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:SEGM1:STIM:STAR 10  
calculate2:limit:segment2:stimulus:start 10
```

Query Syntax CALCulate<cnum>:LIMit:SEGMent<snum>STIMulus:STARt?

Return Type Numeric

Default 0

CALCulate<cnum>:LIMit:SEGMent<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  
calculate2: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  
calculate2:limit:segment3:type lmax
```

Query Syntax CALCulate<cnum>:LIMit:SEGMENT<snum>:TYPE?

Return Type Character

Default OFF

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.

CALCulate:MARKer:

AOFF

BUCKet

BWIDth

COMPression

| **LEVel**

| **PIN?**

| **POUT?**

COUPling

| **METHod**

| **[STATE]**

DELTA

DISCcrete

DISTance

FORMat

FUNCCtion

| **APEak**

| **EXCursion**

| **THReshold**

| **DOMain**

| **USER**

| **STARt**

| **STOP**

| **EXECute**

| **[SElect]**

| **TRACking**

PNOP **more commands**

PSATuration **more**

commands

REFerence

| [\[STATe\]](#)

| [X](#)

| [Y?](#)

SET

[\[STATe\]](#)

[TARGet](#)

[TYPE](#)

[X](#)

[Y?](#)

Click on a keyword to view the command details.

See Also

- [Marker example program](#)
- Marker Readout [number](#) and [size](#) commands.
- [Learn about Markers](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

Important: Learn about [programming the reference marker](#).

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<n>:BUCKet <num>

(Read-Write) Sets and reads the data point (bucket) number of the trace on which the marker resides. When the markers are **interpolated (non-discrete)**, the returned value is the nearest marker bucket position.

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> Marker number to move or query. The marker must already exist. If unspecified, <n> is set to 1.
- <num> Data point (bucket) number. Choose any data point between: 0 and the number of data points minus 1.

Examples

```
CALC:MARK:BUCK 5  
calculate2:marker2:bucket 200
```

Query Syntax CALCulate<cnum>:MARKer<n>:BUCKet?

Return Type 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.

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:bwid -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.

Default -3

CALCulate<cnum>:MARKer<mkr>:COMPression:LEVel <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<cnum>:MARKer:COMPression:LEVel?

Return Type Numeric

Default +1

CALCulate<cnum>:MARKer<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<cnum>:MARKer: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

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <mr> Any existing marker number from 1 to 10; if unspecified, value is set to 1.

Examples

```
CALC:MARK:COMP:POUT?  
calculate2:marker:compression:pout?
```

Return Type Numeric

Default Not applicable

CALCulate<cnum>:MARKer:COUPling:METHod <char>

(Read-Write) Sets and reads the scope of Coupled Markers. This is a global setting that affects all markers. [Learn more](#).

Note: This command will not take effect until Coupled Markers is turned on using [CALC:MARK:COUP:STATe ON](#).

Note: The preset behavior of Coupled Markers depends on the setting of [SYSTem:PREFerences:ITEM:MCControl](#), [SYSTem:PREFerences:ITEM:MCMethod](#), and [SYSTem:PREFerences:ITEM:MCPRest](#).

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <char> **CHANnel** - Coupling is limited to traces in the same channel.
ALL - Coupling occurs across all channels.

Examples

```
CALC:MARK:COUP:METH CHAN  
calculate:marker:coupling all
```

Query Syntax CALCulate:MARKer:COUPling:METHod?

Return Type Character

Default ALL

CALCulate:MARKer:COUPling[:STATe]<ON|OFF>

(Read-Write) Sets and reads the state of Coupled Markers (ON and OFF). The scope of coupled markers can be changed with **CALC:MARK:COUP:METH.**

Parameters

<ON|OFF> **OFF (0)** - Turns Coupled Markers OFF
ON (1) - Turns Coupled Markers ON

Examples

```
CALC:MARK:COUP ON  
calculate:marker:coupling off
```

Query Syntax CALCulate:MARKer:COUPling:[STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default 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`.

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 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>:DISCcrete <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>:DISCcrete?

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 15; 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
NOISe - Noise (available ONLY in IM Spectrum measurement classes).

Examples

```
CALC:MARK:FORMAt MLIN  
calculate2:marker8:format Character
```

Query Syntax CALCulate<cnum>:MARKer<mkr>:FORMAt?

Return Type Character

Default DEFault

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

- <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> Excursion value. Choose any number between **-500** and **500**.

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  
calculate2:marker8:function:apeak:excursion maximum
```

Query Syntax CALCulate<cnum>:MARKer<mkr>:FUNCtion:APeak:EXCursion?

Return Type Numeric

Default 3

CALCulate<cnum>:MARKer<mkr>:FUNCtion:APeak:THResho<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

- <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 15; if unspecified, value is set to 1
- <num> Threshold value. Choose any number between **-500** and **500**.

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>

(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 15; if unspecified, value is set to 1
- 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

Use **CALC:MARK:FUNC:DOM:USER:STOP** 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:START**

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 15; if unspecified, value is set to 1
- <start> The analyzer's **Minimum** x-axis value

Examples

```
CALC:MARK:FUNC:DOM:USER:START 500E6  
calculate2:marker8:function:domain:user:start 1e12
```

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

- <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 15; if unspecified, value is set to 1.
- <stop> Stop value of x-axis span; Choose any number between the analyzer's **MINimum** and **MAXimum** x-axis value.

Examples **CALC:MARK:FUNC:DOM:USER:STOP 500e6**
calculate2:marker8:function:domain1:user:stop 1e12

Query Syntax CALCulate<cnum>:MARKer<mkr>:FUNCtion:DOMain:USER:STOP?

Return Type Numeric

Default The analyzer's **MAXimum** x-axis value.

CALCulate<cnum>:MARKer<mkr>:FUNCtion:EXECute <func>

(Write-only) Immediately executes (performs) the specified search function.

[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 15; if unspecified, value is set to 1.
- <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 15; 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 ltargt
```

Query Syntax CALCulate<cnum>:MARKer<mkr>:FUNCtion[:SElect]?

Return Type Character

Default MAX

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 15; 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 15; 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 ON or OFF. When turned OFF, existing Delta markers revert to general-purpose markers.

Important: Learn about [programming 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.

<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.

Important: Learn about [programming the reference marker](#).

See [Critical Note](#)

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <num> X-axis value. Choose any number within the operating domain of the reference marker.

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.

Important: Learn about [programming 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.

Examples `CALC:MARK:REF:Y?`

`calculate2:marker:reference:y?`

Return Type Character

Default Not applicable

CALCulate<num>:MARKer<mrk>: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

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.
- <mrk> Any marker number from 1 to 15; 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
 - **RELevel** - 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 **CALC:MARK:SET CENT**
calculate2:marker8:set span

Query Syntax Not Applicable

Default Not Applicable

CALCulate<cnum>:MARKer<mkr>[:STATe] <ON|OFF>

(Read-Write) Turns the specified marker ON or OFF. 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 15; 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 15; 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 15; 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 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.

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 15; 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

Calculate:Math Commands

Controls math operations on the currently selected measurement and memory.

CALCulate:MATH:

FUNCtion

MEMorize

Click on a keyword to view the command details.

See Also

- [Example Programs](#)
- [Learn about Math Operations](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>: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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<char> The math operation to be applied. Choose from the following:

NORMal Trace data only

ADD Data + Memory

SUBTract Data - Memory

MULTiPLY Data * Memory

DIVide Data / Memory

Examples

```
CALC:MATH:FUNC NORM  
calculate2:math:function subtract
```

Query Syntax CALCulate<cnum>:MATH:FUNCtion?

Return Type Character

Default NORMal

CALCulate<cnum>:MATH:MEMorize

(Write-only) Puts the currently selected measurement trace into memory. (Data->Memory).

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:MATH:MEM  
calculate2:math:memorize
```

Query Syntax Not applicable

Default Not applicable

Calculate:Mixer Command

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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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:

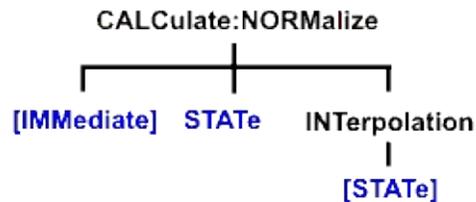
9-Sep-2010 Edit for GCX

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.



Click on a 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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

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 unratiod 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)** - normalization is applied to the measurement.
OFF (or 0) – normalization is NOT applied to the measurement.

Examples

```
CALC:NORM:STAT ON  
calculate2:normalize:state off
```

Query Syntax CALCulate<cnum>:NORMAlize:STATe?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF

CALCulate<cnum>:NORMAlize:INTerpolate[:STATe] <ON | OFF> **Superseded**

Note: This command is replaced with **SENS:CORR:INT[:STATe] ON|OFF**

(Read-Write) Turns normalization interpolation ON or OFF. Normalization is enabled only for measurements of unratiod 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
```

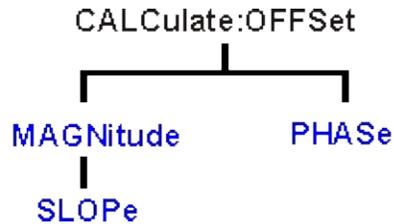
Query Syntax CALCulate<cnum>:NORMAlize:INTerpolate[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

Calculate:Offset Commands

Allows the data trace magnitude and phase to be offset.



Click on a keyword to view the command details.

See Also

- [Example Programs](#)
- [Learn about Magnitude Offset](#)
- [Learn about Phase Offset](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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

- <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: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.

For application measurements, use Calc:Custom commands.

CALCulate:PARAmeter:
CATalog
EXTended
DEFine
EXTended
DELete
ALL
MNUMber
[SELEct]
MODify
EXTended
SELEct
TNUMber?
WNUMber?

Click on a keyword to view the command details.

Blue commands are superseded.

See Also

- [Example Programs](#)
- [Learn about Measurement Parameters](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par:Select](#). [Learn more](#).

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.

The returned string of this command is easily parsed and used to create measurements using the **CALC:PAR:EXT** command.

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 Application Measurements see CALC:CUST:DEF

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> Name of the measurement. Any non-empty, unique string, enclosed in quotes.

<param> Parameter to be measured. Quotes are optional.

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.

Note: For Application Measurements see **CALC:CUST:DEF**

You must select the measurement using **CALC:PAR:SELection** 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.](#)

Note: For Application Measurements see [CALC:CUST:DEF](#)

Examples	<pre>CALC4:PAR:EXT 'ch4_S33', 'S33' Defines an S33 measurement calculate2:parameter:define:extended 'ch1_a', 'b9, 1' logical 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</pre>
----------	--

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[:SELEct] <n>[,fast]

(Read-Write) Sets and returns the selected measurement for the channel using the **Tr#**. 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.

Parameters

- <cnum> Channel number of the measurement to be selected. If unspecified, <cnum> is set to 1.
- <n> Numeric - Measurement number. These are the same numbers you see in the “Tr1”, “Tr2” annotation next to the parameter name on the PNA screen.
- [fast] Optional. The PNA display is NOT updated. Therefore, do not use this argument when an operator is using the PNA display. Otherwise, sending this argument results in much faster sweep speeds. There is NO other reason to NOT send this argument.

Examples

```
CALC:PAR:MNUM 2  
calculate2:parameter:mnumber:select 3,fast
```

Query Syntax

CALCulate<cnum>:PARAmeter:MNUMber[:SELEct]?

There is NO query available to determine if the FAST argument has been set.

Return Type

Numeric

Default

1 (Trace number when factory preset is performed)

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'
```

Query Syntax Not Applicable

Default Not Applicable

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 Application 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>[,fast]

(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.

- Use **CALC:PAR:MNUM** to select a measurement by **Tr#** number. [Learn more](#).
- To obtain a list of currently named measurements, use **CALC:PAR:CAT?**

Parameters

<cnum> Channel number of the measurement to be selected. If unspecified, <cnum> is set to 1.

<Mname> String - Name of the measurement. CASE-SENSITIVE. Do NOT include the parameter name that is returned with Calc:Par:Cat?

[fast] Optional. The PNA display is NOT updated. Therefore, do not use this argument when an operator is using the PNA display. Otherwise, sending this argument results in much faster sweep speeds. There is NO other reason to NOT send this argument.

Examples

```
CALC:PAR:SEL 'TEST'  
calculate2:parameter:select 'test',fast
```

Query Syntax

CALCulate:PARAmeter:SELEct?

There is NO query available to determine if the FAST argument has been set.

Return Type

String

Default "CH1_S11_1" (Trace name when factory preset is performed)

CALCulate<cnum>:PARAmeter:TNUMber?

(Read-only) Returns the trace number of the selected trace. Select a trace using **Calc:Par:Select**.

Parameters

<cnum> Channel number of the trace. If unspecified, <cnum> is set to 1.

Examples

```
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

<cnum> Channel number of the selected trace. If unspecified, <cnum> is set to 1.

Examples CALC:PAR:WNUM?
calculate2:parameter:wnumber?

Return Type Numeric

Default Not Applicable

Last modified:

2-Sept-2014 Calc:Par:Def:Ext..Optional quotes
23-Dec-2013 Added fast to select and mnum commands
1-Aug-2013 Edited Calc:Par:Mod to all app meas
10-Jul-2013 Added note about using EXT commands together.
7-Mar-2012 Added 'for apps, use...'
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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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.



Click on a keyword to view the command details.

See Also

- [Example Programs](#)
- [Learn about Smoothing](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

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

Default 1.5

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

```
CALC:SMO:POIN 50  
calculate2:smoothing:points 21
```

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

```
CALC:SMO ON
```

```
calculate2:smoothing:state off
```

Query Syntax CALCulate<cnum>:SMOothing[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF

CALC:HOLD Commands

Controls the Trace Hold settings.

CALCulate:HOLD TYPE CLear
--

Click on a keyword to view the command details.

see Also

- [Learn about Trace Hold](#)
- [Example Programs](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>:HOLD:TYPE <char>

(Read-Write) Sets the type of trace hold to perform.

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> Trace Hold type. Choose from:
OFF - Disables the Trace Hold feature.
MINimum - Sets Trace Hold to store the lowest measured data points.
MAXimum - Sets Trace Hold to store the highest measured data points.

Examples

```
CALC:HOLD:TYPE MAX  
calculate2:hold:type minimum
```

Query Syntax CALCulate<ch>:HOLD:TYPE?

Return Type Character

Default OFF

CALCulate<cnum>:HOLD:CLEar

(Write-only) Resets the currently-stored data points to the live data trace and restarts the currently-selected Trace Hold type.

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:HOLD:CLE  
calculate2:hold:clear
```

Query Syntax Not Applicable

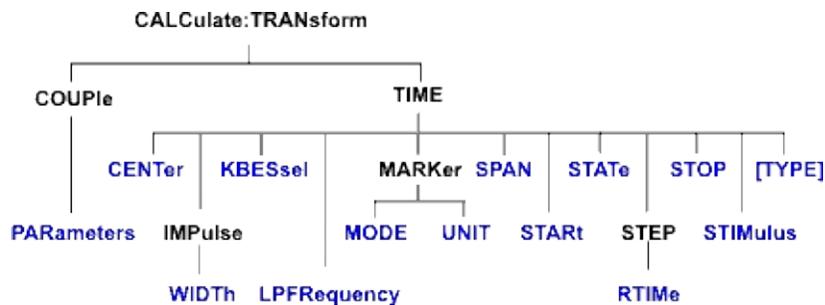
Default Not Applicable

Last Modified:

30-Mar-2015 New topic

Calculate:Transform Commands

Specifies the settings for time domain transform.



Click on a keyword to view the command details.

See Also

- [Example Programs](#)
- [Learn about Time Domain](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [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

<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; any number between:
 $\pm (\text{number of points}-1) / \text{frequency span}$

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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<num> Impulse width in seconds; Choose any number between:
.6 / frequency span and **1.39 / frequency span**

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

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <num> Window width for Kaiser Bessel in seconds; Choose any number between:
0.0 and **13.0**

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

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

Examples

```
CALC:TRAN:TIME:LPFR  
calculate2:transform:time:lpfrequency
```

Query Syntax Not applicable

Default Not applicable

CALCulate<cnum>: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

<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:

AUTO If the active measurement is an S-Parameter, automatically chooses reflection or transmission. If non S-Parameter measurements, reflection is chosen.

REFlection 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:MARK:MODE REFL  
calculate2:transform:time:marker:mode auto
```

Query Syntax CALCulate<cnum>:TRANSform:TIME:MARKer:MODE?

Return Type Character

Default Auto

CALCulate<cnum>:TRANSform:TIME:MARKer:UNIT <char>

(Read-Write) Specifies the unit of measure for the display of marker distance values. This settings 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

```
CALC:TRAN:TIME:SPAN 1e-8  
calculate2:transform:time:span maximum
```

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:
 $\pm (\text{number of points}-1) / \text{frequency span}$

Note: This command will accept **MIN** or **MAX** instead of a numeric parameter. See [SCPI Syntax](#) for more information.

Examples

```
CALC:TRAN:TIME:STAR 1e-8  
calculate2:transform:time:start minimum
```

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:
 $\pm (\text{number of points}-1) / \text{frequency span}$

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

<cnum> 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:
.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

<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:

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 LPASs (Lowpass). (STEP **cannot** be used with TYPE = BPASs.)

:STIM STEP will set :TYPE to **LPASs**

:TYPE BPASs 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

Default IMPulse

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

CALCulate:X (Axis) Commands

Controls the display of X-axis for various measurements.

CALCulate:X: AXIS :DOMain [:VALues]

Click on a keyword to view the command details.

See Also

- [Example Programs](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<ch>:X:AXIS <string>

(Write-Read) Sets the X-axis of the selected measurement to a DC Source. This command does not change the default setting for new traces.

Parameters

- <ch> Channel number of the selected measurement. If unspecified, value is set to 1.
- <string> String - (Not case-sensitive) For all channels EXCEPT DIQ, choose from the following:
- "Default" - The default X-axis setting for the selected measurement. For Application measurements, the X-Axis domain is set with specific commands.
 - "AO1" - Internal DC source #1
 - "AO2" - Internal DC source #2

Note: For DIQ channels, see [CALC:X:AXIS:DOMain](#)

Examples `CALC:X:AXIS 'Default'`
`calculate:x:axis "AO1"`

Query Syntax `CALCulate<ch>:X:AXIS?`

Return Type String

Default "Default"

CALCulate<ch>:X:AXIS:DOMain <string>

(Write-Read) Sets and returns the X-Axis domain of the selected DIQ measurement.

Parameters

<ch> The Differential IQ channel number. If unspecified, value is set to 1.

Choose one of these:	Then set X-Axis Source (CALC:X:AXIS) using one of these as the argument.
"Frequency"	"F1", "F2", etc.
"Power"	Source port: "Port 1", "Port 2", etc.
"Phase"	Source port: "Port 1", "Port 2", etc.
"DC"	DC Source: "AO1", "AO2"
"Points"	"Points"

Example

1. CALC:X:AXIS:DOM "Power"
2. CALC:X:AXIS "Port 1"

Query Syntax CALCulate<ch>:X:AXIS:DOMain?

Return Type String

Default CALC:X:AXIS:DOMain: "Frequency"
CALC:X:AXIS: "F1"

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:MNUM** or **Calc:Par:Select**. [Learn more](#).

This command can be used for all Measurement Classes.

Note: To avoid frequency rounding errors, specify **FORM:DATA** <Real,64> or <ASCIi,0>

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

CalPod Commands

The following commands are sent as a string argument from:

CONTRol:CALPod:COMMand <string>

CALPod

DISable

ENABle

HIDE

INITialize

| **ACTive**

| **ALL**

LAUNCh

RECorrect

| **ACTive**

| **ALL**

SHOW

STATe

TEMP?

Click on a [blue](#) keyword to view the command details.

In addition to the above Calpod commands, the following IEE 488 Common Commands can also be sent as a string argument:

- ***CLS** - Clears all errors and event data from the error/event queue.
- ***IDN?** - Returns the instrument identification information.
- ***OPC?** - Operation complete query. This query immediately returns a value, independent of whether or not the operation is complete. A return value of 0 indicates the operation is not complete. A value of +1 indicates the operation is complete. Typically this command is used in a loop with a 0.25 second delay when waiting for an operation to complete.
- ***TST?** - Performs a communication test on all the currently enabled Calpods. 0 = Test failed on one or more enabled Calpods. 1 = All enabled Calpods working.

- **SYSTEM:ERROR?** - Queries the Event/Error queue and returns the most recent error element.

Important Notes

- ALL commands on this page are sent as a string argument from:
CONTrol:CALPod:COMMand <string>
- Use single quotes ONLY (NOT double quotes) for the CONT:CALP:COMM string arguments.
- Sending queries requires TWO question marks. See following note as example.
- To read errors with the commands on this page, use the Calpod query:
CONT:CALP:COMM? 'SYSTEM:ERROR?'
- ALL queries return strings.

see Also

- [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

CALPod:DISable <port>

(Write-only) Unassign Calpod serial number from the specified PNA port.

See important notes.

Parameters

<port> PNA port number to un-assign.

Examples `CONT:CALP:COMM 'CALP:DIS 2'`

Query Syntax Not Applicable

Default Not Applicable

CALPod:ENABle <port>,<sn>

(Write-read) Assign or return the Calpod serial number for the specified PNA port. If a Calpod module is already assigned to the specified PNA port, this assignment will replace the existing assignment.

See [important notes](#).

Parameters

<port> PNA port number to be assigned the Calpod serial number.

<sn> Calpod serial number.

Examples `CONT:CALP:COMM 'CALP:ENAB 2, 0001234' 'WRITE
CONT:CALP:COMM? 'CALP:ENAB? 2' 'READ`

Query Syntax `CONTrol:CALPod:COMMand? 'CALPod:ENABle? <port>'`

Return Type String

Default Not Applicable

CALPod:HIDE

(Write-only) Hides the Calpod setup dialog.

See important notes.

Parameters None

Examples CONT:CALP:COMM 'CALP:HIDE'

Query Syntax Not Applicable

Default Not Applicable

CALPod:INITialize:ACTive

(Write-only) Performs the initialize process for the active (selected) channel. Select a channel using **Calc:Par:Select**.

See important notes.

Parameters None

Examples `CONT:CALP:COMM 'CALP:INIT:ACT'`

Query Syntax Not Applicable

Default Not Applicable

CALPod:INITialize:ALL

(Write-only) Performs the initialize ALL channels process.

See important notes.

Parameters None

Examples `CONT:CALP:COMM 'CALP:INIT:ALL'`

Query Syntax Not Applicable

Default Not Applicable

CALPod:LAUNch

(Write-only) Starts the Calpod software. The Calpod software can be started using this (Launch) command or by activating the Calpod user interface. Once the Calpod software is started it remains active until the PNA application is terminated.

Send this command first in your program, then wait a couple seconds while the software starts before sending the next command.

See [important notes](#).

Parameters None

Examples `CONT:CALP:COMM 'CALP:LAUN'`
`wait 3`

Query Syntax Not Applicable

Default Not Applicable

CALPod:RECorrect:ACTive

(Write-only) Performs the recorrect process for the active (selected) channel. Select a channel using **Calc:Par>Select**.

See important notes.

Parameters None

Examples `CONT:CALP:COMM 'CALP:REC:ACT'`

Query Syntax Not Applicable

Default Not Applicable

CALPod:RECorrect:ALL

(Write-only) Performs the recorrect process for ALL channels.

See important notes.

Parameters None

Examples CONT:CALP:COMM 'CALP:REC:ALL'

Query Syntax Not Applicable

Default Not Applicable

CALPod:SHOW

(Write-only) Shows the Calpod setup dialog.

See important notes.

Parameters None

Examples `CONT:CALP:COMM 'CALP:SHOW'`

Query Syntax Not Applicable

Default Not Applicable

CALPod:STATE <sn>,<state>

(Write-only) Sets the specified Calpod module to specified impedance state.

See important notes.

Parameters

<sn> Serial number of the Calpod module. When set to **1**, all modules are set to the specified state.

<state> Impedance state. Not case sensitive. Choose from:
Short, Open, Load, or Thru

Examples `CONT:CALP:COMM 'CALPod:STATE 0001234,thru'`

Query Syntax Not Applicable

Default Thru

CALPod:TEMP? <sn>

(Read-only) Returns the temperature of the specified Calpod module in degrees Celsius.

See important notes.

Parameters

<sn> Serial number of the Calpod module.

Examples CONT:CALP:COMM? 'CALPod:TEMP? 0001234'

Query Syntax Not Applicable

Return Type String

Default Not Applicable

Last Modified:

11-Apr-2013	Fixed typo (BH) and removed note for launch
27-Sep-2012	Updated the Launch command (LH)
14-Sep-2012	Removed links from IEEE commands (BH)
25-Jun-2012	Edit OPC? (LH)
4-Jan-2011	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

CALPod:COMMand

ECAL:MODule:

| **PATH:**

| **COUNT?**

| **STATe**

| **STATe**

Click on a [keyword](#) to view the command details.

Blue command is superseded.

CONTrol:CALPod:COMMand <string>

(Write-Read) Sends commands that control a Calpod module. Reads query versions Calpod commands.

See [ALL Calpod commands](#).

[Learn more about Calpod](#).

Parameters

<string> Calpod command. See [ALL Calpod commands](#) that can be used in this string.

Write Example

```
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.

Read Example

```
CONT:CALP:COMM? '*OPC?'
```

'returns 0 if the calpod software is currently processing an operation'

'returns 1 if operations are complete'

Return Type

String

Default

Not Applicable

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

CONTRol: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.

<stateNum>	N4432A and N4433A States	N4431A States	N469x States**	8509x States
One-Port Reflection States				
1	Open	Open	Impedance 1	Open
2	Short	Short	Impedance 2	Short
3	Impedance 1	Impedance 1	Impedance 3	Impedance 1
4	Impedance 2	Impedance 2	Impedance 4	Impedance 2
5			Impedance 5	
6			Impedance 6	
7			Impedance 7	
Two-Port Transmission States				
1	Thru	Thru	Thru	Thru
2	Confidence	Confidence	Confidence	Confidence

** The following modules have only FOUR Impedance states (1, 2, 3, 4):
 N4600B, N4601B, N4602A, N4606B

IN4090D ,IN4091D ,IN4092A ,IN4090D.

Examples **CONT:ECAL:MOD:PATH:STATE A,5**
control:ecal:module2:state BC,1
See example program

Query Not Applicable
Syntax

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 Keysight ECAL modules.

8509x Modules		
State	Port A	Port B
Open	0	0
Short	43	43
Load	33	33
Mismatch	4	16
Thru	84	
Confidence	20	

N469x Modules		
State	Port A	Port B
Open	36	33
Short	39	45
Load	37	37

Mismatch (Offset short)	53	53
Impedance 5 (Offset open)	5	5
Impedance 6 (Offset short)	21	21
Impedance 7 (Offset short)	38	41
Thru	42	
Confidence	40	

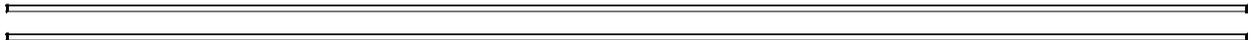
N4431A Modules				
State	Port A	Port B	Port C	Port D
Open	-1398	-1384	-2774	-2654
Short	-1350	-1381	-2582	-2642
Load	26985	-26986	-26986	26985
Mismatch	-26986	26985	26985	-26986
Path	Thru		Confidence	
AB Path	-2590		-598	
AC Path	-4011		85	
AD Path	-2517		16042	
BC Path	-1650		-598	
BD Path	-4011		85	
CD Path	-1352		16042	

N4432A and N4433A Modules				
State	Port A	Port B	Port C	Port D
Open	-6971	-11835	-14895	-14876
Short	-14395	-12859	-14899	-14905
Load	-14907	-14907	-14907	-14907
Offset Short	-9787	-6459	-14874	-14887
Path	Thru		Confidence	
AB Path	13765		30069	
AC Path	-10519		-2327	
AD Path	-10538		-2346	
BC Path	-5655		-1559	
BD Path	-5674		-1578	
CD Path	-15051		30069	

Examples `CONT:ECAL:MOD:STAT 36`
`control:ecal:module2:state 38`

Query Syntax Not Applicable

Default Not Applicable



CONTRol:HANDler:EXT:INDEX:LOGic <char>

(Read-Write) Sets the logic of the index line ("Trig Out" port) on the last PXIe module. There is no soft front-panel element for this feature.

Parameters

<char> Choose from:

POSitive - Causes the Index line to have positive logic (Low signal indicates that the measurement is complete)

NEGative - Causes the Index lines to have negative logic (High signal indicates that the measurement is complete).

Examples

```
CONT:HAND:EXT:INDEX:LOG POS
control:handler:ext:index:logic negative
```

Query Syntax CONTRol:HANDler:EXT:INDEX:LOGic?

Return Type Character

Default POSitive

CSET:Fixture Commands

Manages several aspects of Cal Sets.

CSET:

| [CATalog?](#)

| [COPY](#)

| [DALL](#)

| [DATE?](#)

| [DELete](#)

| [EXISts?](#)

| **ETERm:**

 | [CATalog?](#)

 | [\[:DATA\]](#)

| **FIXTure:**

 | [CASCade](#)

 | [CHARacterize](#)

 | [DEEMbed](#)

 | [EMBed](#)

| [TIME?](#)

Click on a keyword to view the command details.

Note: There is no user-interface equivalent for some of these commands.

See Also

- [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

CSET:CATalog?

This command replaces **SENS:CORR:CSET:CAT?**

(Read-only) Returns the names of Cal Sets stored on the PNA.

Parameters None

Examples CSET:CAT?

Returns:

```
"CalSet_0913,CalSet_1,CalSet_2,CalSet_3,CalSet_4,CH1_CALREG,CH2_CALREG,CH3_CALREG,CH4_CALREG,MyCalAll_SMC_002,MyCalAll_STD_001"
```

Return Type Comma-separated string of names

Default Not Applicable

CSET:COPY <string>,<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

<string>, The first string is the name of the current Cal Set. The second string
<string> is the name of the new Cal Set copy.

Examples `CSET:COPY 'My2Port','My2PortCopy'`

Query Syntax Not Applicable

Default Not Applicable

CSET:DALL

(Write-only) Deletes ALL Cal Sets from the PNA, including phase reference and Global Delta Match Cal Sets.

Parameters None

Examples CSET:DALL

Query Syntax Not Applicable

Default Not Applicable

CSET:DATE? <string>

(Read-only) Returns the (year, month, day) that the specified Cal Set was last saved.

See Also

MMEM:DATE?

MMEM:TIME?

CSET:TIME?

Parameters

<string> Cal Set name.

Examples CSET:DATE? "CalSet_11"

'Returns:

+2013,+5,+1

Return Type Comma-separated integers.

Default Not Applicable

CSET:DEL <string>

This command replaces **SENS:CORR:CSET:DELeTe**

(Write-only) Deletes the specified Cal Set from the PNA.

- If the Cal Set is currently being used by a channel, the Cal Set is deleted and correction for the channel is turned off.
- If the Cal Set is not found, no error is returned.

Parameters

<string> Name of the Cal Set to delete. Not case-sensitive.

Examples CSET:DEL "MyCalSet"

Query Syntax Not Applicable

Default Not Applicable

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.
The GUID must also be enclosed in curly brackets.

Examples

```
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:ETERm:CATalog? <CSET Name>[,<errorTermFilter>]

(Read-only) Returns a list of error term names for the given Cal Set.

Parameters

<CSET Name> (String) Name of Cal Set to query.

<errorTermFilter> (Optional argument) CSET:ETER:CAT? <CSETName>, “<errorTermFilter>” will return only the error term names with the filter string in them. For example, if it is a full 2-port cal, then CSET:ETER:CAT? <CSETName>, “cross” would return all “Crosstalk(n,n)” error terms. (Note that the filter is not case sensitive.)

Entering CSET:ETER:CAT? <CSETName> "" or CSET:ETER:CAT? <CSETName> will return all error terms for the given Cal Set.

Examples

```
CSET:ETER:CAT? "CalSet_1"  
CSET:ETER:CAT? "CalSet_1", "trans"
```

Return Type Variant

Default Not Applicable

CSET:ETERm[:DATA] <CSET Name>,<ETerm Name>,<data>

(Read-Write) Sets and returns the error term data (real, imaginary pairs) for the given Cal Set and error term name.

Parameters

- <CSET Name> (String) Name of Cal Set to manipulate.
- <ETerm Name> (String) Name used to identify an error term in the Cal Set.
- <data> (Block) Error term data - a real/imaginary data pair for each data point.

Examples `CSET:ETER "CalSet_1","Directivity(1,1)", 0.237,-1.422, 0.513, 0.895`
`CSET:ETER? "CalSet_1","Directivity(1,1)" 'read`

Query Syntax CSET:ETERm:DATA? <CSET Name>,<ETerm Name>

Return Type Block data

Default Not Applicable

CSET:FIXTure:CASCade <s2p1>,<s2p2>,<s2pResult>,<char>

(Write-only) Combines the losses and phase shift of two S2P files into a single S2P file.
[Learn more.](#)

Parameters

<s2p1> (String) Path and filename of one of the S2P files to be combined.

<s2p2> (String) Path and filename of the other S2P file to be combined.

<s2pResult> (String) Path and filename of the combined S2P file.

<char> **(Character)** Format. Choose from:

- REIM - Real, imaginary data pairs
- LOG - Log magnitude, phase
- LINear - Linear magnitude, phase

Examples

```
CSET:FIXT:CASC "C:\Users\Public\Documents\Network Analyzer\CPM\a.s2p","C:\Users\Public\Documents\Network Analyzer\CPM\b.s2p","C:\Users\Public\Documents\Network Analyzer\CPM\c.s2p",LOG
```

Query Syntax Not Applicable

Default Not Applicable

CSET:FIXTure:CHARacterize <cs1>,<cs2>,<port>,<s2p>,<char> [<pivot>]

(Write-only) Characterizes a fixture based on two Cal Sets. The stimulus settings of the two Cal Sets do NOT have to be identical, but they MUST have a common frequency range for interpolation. A new S2P file is created. [Learn more about Cal Plane Manager.](#)

Parameters

<cs1> (String) Name of an existing Cal Set 1 which describes the cal closest to the PNA. The Cal Set must reside on the PNA.

<cs2> (String) Name of an existing Cal Set 2 which describes the cal closest to the DUT. The Cal Set must reside on the PNA.

<port> (Numeric) Port number described in the Cal Sets.

<s2p> (String) Name of the S2P file containing the adapter/fixture characterization.

<char> **(Character)** Format. Choose from:

- REIM - Real, imaginary data pairs
- LOG - Log magnitude, phase
- LINear - Linear magnitude, phase

[<pivot>] (Numeric) Optional argument. Phase value for the specified port.

Examples `CSET:FIXT:CHAR "CalSet1","CalSet2",1,"Fixture.s2p"
cset:fixture:characterize
"CalSet1","CalSet2",2,"Fixture.s2p",90`

Query Syntax Not Applicable

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.
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 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

```
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

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:EMB

```
"MyCalSet", "MyNewCalSet", "Fixture.s2p", 1, 1  
cset:fixture:embed  
"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:TIME? <string>

(Read-only) Returns the (hour, minute, second) that the specified Cal Set was last saved. The time is returned in local time as setup in the PNA operating system.

See Also

CSET:DATE?

MMEM:DATE?

MMEM:TIME?

Parameters

<string> Cal Set name.

Examples CSET:TIME? "CalSet_11"

'Returns:

+13,+6,+1

Return Type Comma-separated integers.

Default Not Applicable

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

TOOLbar

| **ENTRy[:STATE]**

| **EXTension[:STATE]**

| **KEYS[:STATE]**

| **MARKer[:STATE]**

| **MEAS[:STATE]**

| **STIMulus[:STATE]**

| **SWEep[:STATE]**

|

TRANSform[:STATE]

VISible

WINDow

| **ANNotation**

| **LIMit**

| **XPOSITION**

| **YPOSITION**

| **MARKer**

| **NUMBER**

| **RESolution**

| **RESPonse**

| **STIMulus**

|
SINGle[:STATe]

| **SIZE**

| **STATe**

| **SYMBol**

|
ABOVe[:STATe]

| **XPOStion**

| **YPOStion**

| **TRACe[:STATe]**

| **CATalog?**

| **ENABle**

| **SIZE**

| **[STATe]**

| **TABLE**

| **TITLe**

| **DATA**

| **[STATe]**

| **TRACe**

| **DELeTe**

| **FEED**

|
GRATicule:GRID:LTYPe

| **MEMory[:STATe]**

| **MOVE**

| **SELeCt**

| **[STATe]**

| **TITLe**

| **DATA**

| **[:STATe]**

| **Y[:SCALe]**

| **AUTO**

| **COUPle**

| **METHod**

| **[STATe]**

| **PDIVision**

| **RLEVel**

| **RPOStion**

| **Y:AUTO** |

Click on a keyword to view the command details.

Blue keywords are superseded.

see Also

- [Referring to Traces Channels Windows and Meas Using SCPI](#)
 - See an [example](#) using some of these commands
 - [Synchronizing the Analyzer 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:ANNotation: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:ANNotation:MESSage:STATe?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON (1)

DISPlay:ANNotation[: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:ANNotation[: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 DISPlay: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 DISPlay: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 **DISP:TMAX ON**
display:tmax 0

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 **DISP**
 display:tile

Default Not Applicable

DISPlay:TOOLbar:ENTRy[:STATe] <bool>

(Read-Write) Specifies whether to show or hide the active entry toolbar. [See this toolbar.](#)

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples

```
DISP:TOOL:ENTR ON  
display:toolbar:entry:state off
```

Query Syntax DISPlay:TOOLbar:ENTRy[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:EXTensions[:STATe] <bool>

(Read-Write) Specifies whether to show or hide the port extensions toolbar. [See this toolbar.](#)

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples

```
DISP:TOOL:EXT ON  
display:toolbar:extensions:state off
```

Query Syntax DISPlay:TOOLbar:EXTensions[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:KEYS[:STATe] <bool>

(Read-Write) Specifies whether to show or hide the virtual hardkeys on the PNA display. These are primarily used when the PNA is accessed remotely using VNC or Windows Remote Desktop.

Parameters

<bool> **ON** (or 1) - Keys ON.
OFF (or 0) - Keys OFF.

Examples **DISP:TOOL:KEYS ON**
display:toolbar:keys:state off

Query Syntax DISPlay:TOOLbar:KEYS [:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:MARKer[:STATe] <bool>

(Read-Write) Specifies whether to show or hide the marker toolbar. [See this toolbar.](#)

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples **DISP:TOOL:MARK ON**
display:toolbar:marker:state off

Query Syntax DISPlay:TOOLbar:MARKer[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:MEASurement[:STATe] <bool> **OBSOLETE**

This toolbar was eliminated with A.10.00

(Read-Write) Specifies whether to show or hide the measurement toolbar.

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples

```
DISP:TOOL:MEAS ON  
display:toolbar:measurement:state off
```

Query Syntax DISPlay:TOOLbar:MEASurement[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:STIMulus[:STATe] <bool> **OBSOLETE**

This toolbar was eliminated with A.10.00

(Read-Write) Specifies whether to show or hide the stimulus toolbar.

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples

```
DISP:TOOL:STIM ON  
display:toolbar:stimulus:state off
```

Query Syntax DISPlay:TOOLbar:STIMulus[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:SWEep[:STATe] <bool> **OBSOLETE**

This toolbar was eliminated with A.10.00

(Read-Write) Specifies whether to show or hide the sweep control toolbar.

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples **DISP:TOOL:SWE ON**
display:toolbar:sweep:state off

Query Syntax DISPlay:TOOLbar:SWEep[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:TOOLbar:TRANSform[:STATe] <bool>

(Read-Write) Specifies whether to show or hide the Time Domain toolbar. [See this toolbar.](#)

Parameters

<bool> **ON** (or 1) - Toolbar ON.
OFF (or 0) - Toolbar OFF.

Examples

```
DISP:TOOL:TRAN ON  
display:toolbar:transform:state off
```

Query Syntax DISPlay:TOOLbar:TRANSform[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

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 **DISP:VIS ON**
display:visible off

Query Syntax DISPlay:VISible?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:WINDow<wnum>:ANNotation:LIMit:XPOSition <num>

(Read-Write) Sets and returns the X-axis position of the Limit Line Pass/Fail indicator on the VNA screen. The lower-left corner of the Pass/Fail indicator is the point of reference for positioning.

Parameters

- <wnum> Any existing window number. If unspecified, value is set to 1.
- <num> X-axis position. Choose a value between 0 (far left) and 10 (far right).

Examples

```
DISP:WIND:ANN:LIM:XPOS 1.5  
display:window:annotation:limit:xposition 5
```

Query Syntax DISPlay:WINDow:ANNotation:LIMit:XPOSition?

Return Type Numeric

Default 7

DISPlay:WINDow<wnum>:ANNotation:LIMit:YPOSition <num>

(Read-Write) Sets and returns the Y-axis position of the Limit Line Pass/Fail indicator on the VNA screen. The lower-left corner of the Pass/Fail indicator is the point of reference for positioning.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1.

<num> Y-axis position. Choose a value between 0 (bottom) and 10 (top).

Examples

```
DISP:WIND:ANN:LIM:YPOS 1.5
```

```
display:window:annotation:limit:yposition 5
```

Query Syntax DISPlay:WINDow:ANNotation:LIMit:YPOSition?

Return Type Numeric

Default 0

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

```
DISP:WIND:ANN:MARK:NUMB 7  
display:window:annotation:marker:number 2
```

Query Syntax DISPlay:WINDow:ANNotation:MARKer:NUMBER?

Return Type Numeric

Default 5

DISPlay:WINDow<wnum>:ANNotation:MARKer:RESolution:STIM<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**.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1.

<num> Number of digits to display. Choose a value between 2 and 6.

Examples

```
DISP:WIND: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:RI<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**.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1.

<num> Number of digits to display. Choose a value between 1 and 4.

Examples

```
DISP:WIND: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[:STAT<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 [Marker](#) 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

```
DISP:WIND:ANN:MARK:SING ON  
display>window:annotation:marker:single off
```

Query Syntax DISPlay:WINDow:ANNotation:MARKer: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 **Marker** 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

```
DISP:WIND:ANN:MARK:SIZE LARG  
display>window:annotation:marker:size normal
```

Query Syntax DISPlay:WINDow:ANNotation:MARKer:SIZE?

Return Type Character

Default NORMAl

DISPlay:WINDow<wnum>:ANNotation:MARKer[:STATe] <ON | OFF>

(Read-Write) 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.
OFF (or 0) - turns marker readout OFF.

Examples

```
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

[See pictures of each](#)

Examples

```
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:SYMBol:ABOV<ON | OFF>

(Read-Write) Specifies whether or not to force marker symbols to be displayed above the trace. When ON, all marker symbols will be displayed above the trace and the active marker will be filled solid. See other SCPI **Marker** commands.

Parameters

<wnum> Any existing window number. If unspecified, value is set to 1.

<ON | OFF> **ON (or 1)** - ALL marker symbols are displayed above the trace. Only the active marker is filled solid.

OFF (or 0) - ONLY the active marker is displayed above the trace. The active marker is not filled solid.

Examples

```
DISP:WIND:ANN:MARK:SYMB:ABOV ON  
display>window:annotation:marker:symbol:above:state off
```

Query Syntax DISPlay:WINDow:ANNotation:MARKer:SYMBol:ABOVe[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF - ON in IM Spectrum and SA measurement classes

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

```
DISP:WIND:ANN ON  
display:window:annotation:trace:state off
```

Query Syntax DISPlay:WINDow: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

```
DISP:WIND:ENABle ON  
display:window1:enable off
```

Query Syntax DISPlay: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

```
DISP:WIND:SIZE MAX  
display:window:size norm
```

Query Syntax DISPlay: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 SEGMENT**
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

```
DISP:WIND:TITL:DATA 'hello'  
display:window2:title:data 'hello'
```

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

```
DISP:WIND:TITL ON  
Display>window1:title:state off
```

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 **DISP:WIND:TRAC:DEL**
display>window2:trace2:delete

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 <parameter>, but different <name>, 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 **DISP:WIND:TRAC:FEED 'test'**
display>window2:trace2:feed 'test'

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<fromWin>:TRACe<tnum>:MOVE <toWin>

(Write-only) Moves a trace from one window to another window.

Parameters

<fromWin> Window number to move the trace **from**. If unspecified, value is set to 1.

Use **Disp:Cat?** to read the existing window numbers.

<tnum> Trace number to be moved. If unspecified, value is set to 1.

Use **Disp:Wind:Cat?** to read the trace numbers in an existing 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.

<toWin> Number of the window to move the trace **to**. If the window does not exist, it will be created.

Examples **DISP:WIND:TRAC2:MOVE 2**
display>window2:trace2:move 1

Query Syntax Not applicable

Default Not applicable

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.
Use **Disp:Cat?** to read the existing window numbers.

<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.
Use **Disp:Cat?** to read the existing window numbers.

<tnum> Any existing trace number; if unspecified, value is set to 1.
Use **Disp:Wind:Cat?** to read the trace numbers in an existing 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.

<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 about Trace Titles.](#)

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.
Use **Disp:Cat?** to read the existing window numbers.

<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
Use **Disp:Cat?** to read the existing window numbers.
- <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 **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.
Use [Disp:Cat?](#) to read the existing window numbers.

<tnum> Any existing trace number; if unspecified, value is set to 1.
Use [Disp:Wind:Cat?](#) to read the trace numbers in an existing 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.

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

```
DISP:WIND:TRAC:Y:COUP:METh ALL  
Display>window2:trace:y:scale:method window
```

Query Syntax DISPlay:WINDow:TRACe:Y[:SCALe]:COUPle:METhod?

Return Type Character

Default OFF

DISPlay:WINDow<wnum>:TRACe:Y[:SCALe]:COUPle[:STATe] <bool>

(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
Use **Disp:Cat?** to read the existing window numbers.
- <bool> **ON** (or 1) - Scale coupling enabled for specified window.
OFF (or 0) - Scale coupling disabled for specified window.

Examples

```
DISP:WIND:TRAC:Y:COUP ON  
Display>window2:trace:y:scale:couple:state off
```

Query Syntax DISPlay:WINDow<wnum>:TRACe:Y[:SCALe]:COUPle[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

DISPlay:WINDow<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.
Use **Disp:Cat?** to read the existing window numbers.

<tnum> Any existing trace number; if unspecified, value is set to 1.
Use **Disp:Wind:Cat?** to read the trace numbers in an existing 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.

<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.
Use **Disp:Cat?** to read the existing window numbers.

<tnum> Any existing trace number; if unspecified, value is set to 1.
Use **Disp:Wind:Cat?** to read the trace numbers in an existing 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.

<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]:RPOSitio<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.
Use **Disp:Cat?** to read the existing window numbers.

<tnum> Any existing trace number; if unspecified, value is set to 1.
Use **Disp:Wind:Cat?** to read the trace numbers in an existing 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.

<num> Reference position on the screen measured in horizontal graticules from the bottom. Choose a value between 0 and 10.

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]:RPOSitio?

Return Type Numeric

Default 5

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.
Use **Disp:Cat?** to read the existing window numbers.

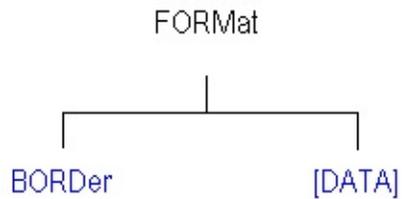
Examples **DISP:WIND:Y:AUTO**
display>window2:y:auto

Query Syntax Not applicable

Default Not applicable

Format Commands

Specifies the way that data will be transferred when moving large amounts of data.



Click on a keyword to view the command details.

see Also

- [Example Programs](#)
 - [Synchronizing the Analyzer 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 transferring measurement data and frequency data.

- 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**
- To transfer FIFO buffer data, use **SYST:FIFO:DATA?**

The following commands transfer frequency data. Use <**REAL, 64**>

- **CALC:DATA:SNP?**
- **CALC:DATA:SNP:PORTS?**
- **CALC:X?**
- **SENS:X?**

Use **FORMat:BORDER** to change the byte order. Use “NORMAL” when transferring a binary block from LabView or Vee. For other programming languages, you may need to SWAP the byte order.

Parameters

<char> In the PNA, measurement data is stored as 32 bit and frequencies stored as 64 bit. Therefore, use REAL,32 when getting data and REAL,64 when getting frequencies. That way you are guaranteed to avoid losing any precision as well as getting the maximum speed on the data transfer.

Choose from:

- **REAL,32** - (default value for REAL) Best for transferring large amounts of measurement data. Can cause rounding errors in frequency data.
- **REAL,64** - Slower but has more significant digits than

REAL,32. REQUIRED to accurately represent frequency data. See above list for commands which transfer frequency information.

- **AScii,0** - The easiest to implement, but very slow. Use when you have small amounts of data to transfer.

Note 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:

10-Sep-2012	Added 64 for frequency
6-Sep-2012	Added Form:BOrder link and note (JE)
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 keyword to view the command details.

Blue commands are superseded or obsolete.

see Also

- [Learn more about PNA Printing](#)
 - [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
-

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	HCOP hcopy:immediate
-----------------	---------------------------------------

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

<bool> Active window state. Chose from:
OFF or (0) - Print ALL windows.
ON or (1) - Print Active window only.

Examples HCOP:ITEM:AWIN 1
hcopy:item:awindow:state off

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  
hcopy:item:gpfail:state off
```

Query Syntax HCOPy:ITEM:GPFail[:STATE]?

Return Type Boolean

Default OFF (0)

HCOPy:ITEM:LOGO[:STATe] <bool>

(Read-Write) When ON, prints the Keysight Technologies logo in the page header.

This setting survives instrument preset and PNA application restart.

Parameters

<bool> Keysight logo print state. Chose from:
OFF or (0) - Prints the Keysight logo.
ON or (1) - Does NOT print the Keysight logo.

Examples

```
HCOP:ITEM:LOGO 1  
hcopy: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**
hcopy: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

hcopy: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:SEGD 1**
hcopy: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  
hcopy:item:swindow:state off
```

Query Syntax HCOPY:ITEM:SWINDOW[: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`
`hcopy: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

```
hcopy: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
hcopy: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:AWINDow** to specify all windows or only the active window.

This setting survives instrument preset and PNA application restart.

Parameters

<bool> Windows print state. Chose 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:DIMensions: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

<int> Choose from:1

Integer	Description
1	Letter 8 1/2 x 11 in
2	Letter Small 8 1/2 x 11 in
3	Tabloid 11 x 17 in
4	Ledger 17 x 11 in
5	Legal 8 1/2 x 14 in
6	Statement 5 1/2 x 8 1/2 in
7	Executive 7 1/4 x 10 1/2 in
8	A3 297 x 420 mm
9	A4 210 x 297 mm
10	A4 Small 210 x 297 mm
11	A5 148 x 210 mm
12	B4 (JIS) 250 x 354
13	B5 (JIS) 182 x 257 mm

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

hcopy:page:size 5

Query Syntax HCOPY:PAGE:SIZE?

Return Type Integer

Default 1

HCOPY:SDUMP:DATA?

(Read-only) Returns the display image in a definite-length arbitrary binary block. The format of the data is PNG by default. Use **HCOPY: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.

Examples **HCOPY:SDUM?**
hcopy:sdump?

Return Type A definite-length arbitrary binary block

Default Not Applicable

HCOPY:SDUMp:DATA:FORMAt <char>

(Read-Write) Sets the graphic format for **HCOPY:SDUMp:DATA?**

Parameters

<char> Choose from: **JPG** | **BMP** | **PNG**

Examples **HCOP:SDUMp:DATA:FORMAt BMP**

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.

Examples HCOP:PRIN?
hcopy:printers?

Query Syntax HCOPy:PRINters?

Return Type String

Default Not Applicable

Last modified:

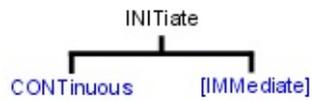
13-Apr-2011 Obsolete seg data

15-Mar-2010 Added SDUMP commands (A.09.20)

Nov. 1, 2006 Added new commands

Initiate Commands

Controls triggering signals



Click on a **red** keyword to view the command details.

see Also

- **Example** [Triggering the PNA](#)
 - [Learn about Triggering](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

INITiate:CONTinuous <boolean>

(Read-Write) Specifies whether the PNA trigger source is set to Internal (continuous) or Manual.

- For SIMPLE, single-triggering of a single channel, use **Sens:Sweep:Mode SINGLE** which sets the number of trigger signals each channel will ACCEPT (Continuous, Groups, **Single**, or HOLD - none.)
- This command is a subset of **TRIG:SEQ:SOURce**, which can also set the trigger source to External.
- 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).

- This command requires **Trigger:Source** to be set to Manual. This causes ONE trigger signal to be SENT each time INIT:IMM is issued.
- For SIMPLE, single-triggering of a single channel, use **Sens:Sweep:Mode SINGLE** which sets the number of trigger signals each channel will ACCEPT (Continuous, Groups, **Single**, or HOLD - none.)

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.

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

<cnum> Any existing channel number. If unspecified, value is set to 1

Examples INIT
initiate2:immediate

Query Syntax Not applicable

Default Not applicable

Last modified:

8-Mar-2013 Updated again with Single note

April 23, 2007 Updated Init:Imm

Memory Commands

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](#).

MMEMory:

CATalog?

CDIRectory

COPY

DATE?

DELeTe

LOAD

MDIRectory

MOVE

RDIRectory

STORe

| **CITI**

| **DATA**

| **FORMat**

| **DATA**

| **ENR**

| **TRACe**

| **CONTents**

| **CITIfile**

| **FORMat**

| **CITIfile**

| **SNP**

TIME?

TRANsfer

Click on a keyword to view the command details.

Blue commands are superseded.

See Also

- [Example Programs](#)

- [Learn about Save / Recall and File Types](#)
- [Synchronizing the Analyzer and Controller](#)
- [SCPI Command Tree](#)

Specifying Path Names

The MMEM commands use the following rules to specify path names:

- The default folder for Windows 7 is "c:\users\public\documents\Network analyzer" [Learn more](#).
 - 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:\users\public\documents\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 the default path, then no punctuation is required.

MMEM:CDIR Service

If the new folder is at a different level than the default, 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 See [Specifying Path Names](#)

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:DATE? <fileName>

(Read-only) Returns the (year, month, day) that the specified file was last saved.

To query the last date and time a cal set was modified, use [CSET DATE?](#) and [CSET:TIME?](#)

See Also

[MMEM:TIME?](#)

Parameters

<fileName> String - File name. See [Specifying Path Names](#)

Example `MMEM:DATE? "myFile.txt"`

'Returns

+2013,+4,+12

`mmemory:date? "c:\users\public\documents\Network analyzer\UserCalSets\Calset_18.pcs"`

'Returns

+2013,+4,+12

Return Type Comma-separated integers

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>

(Write-only) Stores the specified file (.sta, .cal, .cst, .csa, .snp, s2px).

Learn about [saving SNP files on the PNA](#).

Learn about [saving S2Px files on the PNA](#).

To save other 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)

No <char> is specified for s1p, s2p, s2px and so forth.

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'  
MMEM:STOR 'MyData.S2P'
```

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 path that already exists with filename.
If the same level as the default, then no path is required.

```
MMEM:STOR:CIT:DATA 'MYFile.cti'
```

Of you can specify an absolute path and filename:

```
mmemory:store:citfile:data  
"c:\users\public\documents\Network analyzer\myFile.cti'
```

Query Syntax Not Applicable

Default See [Specifying Path Names](#)

MMEMory:STORe:CITifile:FORMat <filename> - **Superseded**

This command is replaced with **MMEMory:STORe:DATA**.

(Write only) Saves FORMATTED trace data to .cti file. [Learn more](#).

Parameters

<filename> Any path that already exists with filename.
If the same level as "c:\users\public\network analyzer\documents", then no path is required

```
MMEM:STOR:CIT:FORM 'MYFile.cti'
```

Of you can specify an absolute path and filename:

```
mmemory:store:citfile:format  
"c:\users\public\documents\Network analyzer\myFile.cti'
```

Query Syntax Not Applicable

Default See [Specifying Path Names](#)

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**

NOTE: Not all choices are valid with other arguments. See **Valid parameter combinations** below.

Parameters

<filename> (String) Name and extension of the file to which data will be saved. If the extension does not agree with the file type, an error is NOT returned but the data may NOT be what you expect.

See rules for specifying a filename.

<type> (String) File type to save. Choose from:

"PRN Trace Data" - *.prn data. **Learn more.**

"Citifile Data Data" - unformatted *.cti data. **Learn more.**

"Citifile Formatted Data" - formatted *.cti data.

"CSV Formatted Data" - formatted *.csv data. **Learn more.**

"MDIF Data" - *.mdf data. **Learn more.**

"GCA Sweep Data" - Gain compression data. **Learn more.**

"IMD Sweep Data" - Swept IMD data. **Learn more.**

<scope> (String) How much data to save. Choose from:

"Trace" - only the specified measurement number is saved.

"Displayed" - all displayed measurements are saved.

"**Channel**" - all measurements that are in the channel in which the selected measurement reside are saved.

"**Auto**"

For all Standard Meas Class (S-parameter) channels:

- When correction is OFF, the specified trace is saved.
- When correction is ON, all corrected parameters associated with the calibrated ports in the Cal Set are saved.

For all other channels:

- When correction is OFF or ON, the specified trace is saved.

<format> The format in which data is saved. Choose from:

"**Displayed**" - the format is the same as that in which it is displayed on the PNA screen.

"**RI**" - Real / Imaginary

"**MA**" - Magnitude / Angle

"**DB**" - LogMag / Degrees

<selector> (Integer) Choose from:

-1 Use when <scope> = "Displayed" (does NOT require a selected trace).

<measurement number> Use for all other <scope> selections.
Use **Calc:Par:MNUM?** to read the measurement number of the selected trace.

The following are **valid parameter combinations** for ALL measurement classes:

Parameters			
<type> (String)	<scope> (String)	<format> (String)	<selector> (Numeric)
"PRN Trace Data"	"Trace"	"Displayed"	Measurement number

Example: MMEMoRY:STORe:DATA "myData.prn", "PRN Trace

Data", "Trace", "Displayed", 2

"Citifile Data Data"	"Trace" or "Auto" or "Channel"	"RI"	Measurement number
	"Displayed"	"RI"	-1

Example: MMEMoRY:STORe:DATA "myData.cti", "Citifile Data Data", "AUTO", "RI", 3

"Citifile Formatted Data"	"Trace" or "Auto"	"RI" or "MA" or "DB"	Measurement number
	"Channel"	"RI" or "MA" or "DB" or "Displayed"	Measurement number
	"Displayed"	"RI" or "MA" or "DB" or "Displayed"	-1

Example: MMEMoRY:STORe:DATA "myData.cti", "Citifile Formatted Data", "AUTO", "MA", 3

"CSV Formatted Data"	"Trace" or "Auto" or "Channel"	"RI" or "MA" or "DB" or "Displayed"	Measurement number
	"Displayed"	"RI" or "MA" or "DB"	-1

Example: MMEMoRY:STORe:DATA "myData.csv", "CSV Formatted Data", "displayed", "RI", -1

"MDIF Data"	"Trace" or "Auto"	"RI" or "Displayed" or "Channel"	Measurement number
	"Displayed"	"RI" or "Displayed"	-1

Example: MMEMoRY:STORe:DATA "myData.mdf", "MDIF Data", "displayed", "displayed", -1

The following parameter combinations save *.csv files in specific formats for GCA and Swept IMD classes:

Parameters			
<type> (String)	<scope> (String)	<format> (String)	<selector> (Numeric)

"GCA Sweep Data"	"Auto"	"DB"	GCA channel number
------------------	--------	------	--------------------

Example: MMEemory:STORe:DATA "myData", "gca sweep data", "displayed", "displayed", -1

"IMD Sweep Data"	"Auto"	"DB"	Swept IMD channel number
------------------	--------	------	--------------------------

Example: MMEemory:STORe:DATA "myData.mdf", "MDIF Data", "displayed", "displayed", -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)

To set and read ENR data, use [SENS:CORR:ENR:CAL:TABLE:DATA](#).

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

Default Auto

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:TIME? <fileName>

(Read-only) Returns the (hour, minute, second) that the specified file was last saved. The time is returned in local time as setup in the PNA operating system.

To query the last date and time a cal set was modified, use **CSET DATE?** and **CSET:TIME?**

See Also

MMEM:DATE?

Parameters

<fileName> String - File name. See [Specifying Path Names](#)

Example `MMEM:TIME? "myFile.txt"`

'returns

+12,+34,+12

`mmemory:time? "c:\users\public\documents\Network analyzer\UserCalSets\Calset_18.pcs"`

'returns

+12,+34,+12

Return Type Comma-separated integers

Default Not applicable

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:

```
#210ABCDE+WXYZ<nl><end>
```

Where:

- always sent before definite block data

2 - specifies that the byte count is two digits (2)

10 - specifies the number of data bytes that will follow, not counting <NL><END>

ABCDE+WXYZ - 10 digits of data

<NL><END> - always sent at the end of block data

Example **See example program**

Query Syntax MMEMory:TRANsfer? <fileName>
Reads block data from the specified file location.

Default Not applicable

Output Commands

Controls two output functions: RF power and Noise Source.

OUTPut:

| [MANual:NOISe\[:STATe\]](#)

| [\[:STATe\]](#)

Click on a [red](#) keyword to view the command details.

See Also

- [Example Programs](#)
- [Synchronizing the Analyzer 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

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.

SENSe:AVERage

| [CLEar](#)

| [COUNt](#)

| [MODE](#)

| [\[STATe\]](#)

Click on a keyword to view the command details.

See Also

- [Example](#) using some of these commands.
 - [Learn about Averaging](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SENSe<cnum>:AVERage:CLEar

(Write-only) Clears and restarts averaging of the measurement data. Does NOT apply to point averaging.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.

Examples

```
SENS:AVER:CLE  
sense2:average:clear
```

Query Syntax Not applicable

Default Not applicable

SENSe<cnum>:AVERAge:COUNT <num>

(Read-Write) Sets the number of measurements to combine for an average. Must also set **SENS:AVER[:STATe] ON**

Parameters

<cnum> 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:COUNT 999**
sense2:average:count 73

Query Syntax SENSe<cnum>:AVERAge:COUNT?

Return Type Numeric

Default 1

SENSe<cnum>:AVERAge:MODE <char>

(Read-Write) Sets the type of averaging to perform: Point or Sweep.

Parameters

<cnum> 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.

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 SENSe<cnum>:AVERAge:MODE?

Return Type Character

Default Sweep

SENSe<cnum>: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

```
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 Analyzer 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:Bwidth**)

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 Varies with 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:

7-May-2012 Removed Preset

15-Jan-2008 MIN and MAX added

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]**

| **METHod**

| **NOISe**

| **ENR:ADAP:DEEMbed[:STATe]**

| **LO:PCAL[:STATe]**

| **PSEN:ADAP:DEEMbed[:STATe]**

| **SAVE**

| **SESSion - More Commands**

| **SWEep:CHANnel**

| **AOFF**

| **[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**

| **OVERrange[:STATe]**

| **PMAP**

| **SIMCal**

| **TRIG:FREE**

RPOWER:OFFSet

| **[AMPLitude]**

RVELocity

| **COAX**

SEOWord

SCPI

| **[STATe]**

[STATe]

TCOLd:USER:VALue

TStandards

| **[STATe]**

TYPE

| **CATalog?**

Click on a keyword to view the command details.

Blue commands are superseded.

see Also

- [Example Programs](#)
 - **New** [See Calibrating the PNA Using SCPI](#)
 - [Learn about Measurement Calibration](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

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..
ECAL50
- [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 Keysight). Default if not specified.
CHAR1 User characterization #1
CHAR2 User characterization #2
...and so forth up to:
CHAR12 User characterization #12

Examples

```
SENS:CORR:CCHeck ECAL2  
sense2:correction:ccheck:acquire ecal1,char1
```

Query Syntax Not applicable

Default 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 SENS:CORR:CCH:DONE sense2:correction:ccheck:done

Query Syntax Not applicable

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

```
SENS:CORR:CCH: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

SENSe<cnum>:CORRection:COLLect[:ACQuire] <class> [subclass][,sync]

(Write-only) 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:

Label	SOLT (Forward)	SOLT (Reverse)	TRL
STAN1	SA	SA	TRL "R"
STAN2	SB	SB	N/A
STAN3	SC	SC	TRL "L"
STAN4	FWD TRANS	REV TRANS	TRL "T"
STAN5	Generic Isolation; not associated with calibration kit definition.		
ECAL1	through ECAL modules		
ECAL50			
RESPonse			
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 Keysight)

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 about this argument](#)

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 Not applicable

Default Not applicable

SENSe<cnum>: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<cnum>:CORRection:COLLect:SAVE` calculates and then automatically applies error terms after you use `SENS:CORR:COLL:ACQuire` to measure cal standards.

Parameters

<cnum> 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:WIND**.

Examples

```
SENS:CORR:COLL:DISP:WIND: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 to view the measurements/channels. [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

```
SENSe: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 SENSE:CORRection:COLLect:ISOLation:ECAL:STATe?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF

SENSe<cnum>: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

<cnum> 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 REFL1	Response Short
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
SPARSOLT3	Full SOLT 3 port
SPARTRL	TRL Cal (Delta Match Cal may be required)

Examples

```
SENS:CORR:COLL:METh REFL1  
sense2:correction:collect:method sparsolt
```

Query Syntax SENSE<num>:CORRection:COLLect:METhod?

Return Type Character

Default Not Applicable

SENSe<ch>:CORRection:COLLect:NOISe:ENR:ADAPter:DEEM [STATe] <bool>

(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

```
SENS:CORR:COLL:NOIS:ENR:ADAP:DEEM 0
```

```
sense2:correction:collect:noise:enr:adapter:deembed:state ON
```

Query Syntax

```
SENSe:CORRection:COLLect:NOISe:ENR:ADAPter:DEEMbed:  
[STATe]?
```

Return Type

Boolean

Default

0 - OFF

SENSe<ch>:CORRection:COLLect:NOISe:LO<n>:PCAL[:STAT <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 0 - OFF

SENSe<ch>:CORRection:COLLect:NOISe:PSENSor:ADAPter:D[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

```
SENS:CORR:COLL:NOIS:PSEN:ADAP:DEEM 0  
sense2:correction:collect:noise:psensor:adapter:deembed:state  
ON
```

Query Syntax SENSe:CORRection:COLLect:NOISe:PSENSor:ADAPter:DEEMbed:
[STATe]?

Return Type Boolean

Default 0 - 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:PREFerece: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

Examples **SENS:CORR:COLL:SAVE**
sense2:correction:collect:save

Query Syntax Not applicable

Default Not applicable

SENSe:CORRection:COLLect:SWEEp:CHANnel:AOff

(Write-only) Clears ALL flags for channels to sweep during calibration. To flag a channel, see **SENSe:CORRection:COLLect:SWEEp:CHANnel:ON**.

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

```
SENS:CORR: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** to load, and **MMEM:STORE:ENR CAL** to 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

```
SENS:CORR:ENR:CAL:TABL: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

```
SENS:CORR:ENR:CAL:TABL:ID:DATA "346C"  
sense:correction:enr:calibration:table:id:data "ENR Table"
```

Query Syntax SENSE:CORRection:ENR:CALibration:TABLE:ID:DATA?

Return Type String

Default Not Applicable

SENSe: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

```
SENS:CORR:ENR:CAL:TABL:SER:DATA "ABCD1234"  
sense:correction:enr:calibration:table:serial:data "ABCD1234"
```

Query Syntax SENSE:CORRection:ENR:CALibration:TABLE:SERial:DATA?

Return Type String

Default Not Applicable

SENSe<ch>:CORRection:GCSetup:POWer <num>

(Read-Write) Set and read the power level at which to perform the Source Power Calibration 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

```
SENS:CORR:GCS:POW 0  
sense:correction:gcsetup:power 5
```

Query Syntax SENSe: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

<string> Cal Kit. Use **SENS:CORR:COLL:GUID:CKIT:PORT1:CAT?** to return a list of valid cal kits.

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

<string> Power sensor connector type. Use **SENS:CORR:COLL:GUID:CONN:CAT?** to return a list of valid connector types.
Select "Ignored" to NOT compensate for the adapter.

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. The set and query of this command will NOT return an error.

To perform isolation as part of an unguided calibration, you must explicitly measure the isolation standard using **SENS:CORR:COLL:ACQ Stan5**.

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<num>:CORRection:ISOLation[:STATe]?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF - (Isolation disabled)

SENSe<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

SENSe: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** - Each Cal is saved ONLY to the channel Cal Register. If the error terms from a new Cal can co-exist with those in the Cal Register, they are appended.

USER - Each Cal is saved to its own new User Cal Set file. 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. If the error terms from a new Cal can co-exist with those in the Cal Set, they are appended.

Examples

```
SENS:CORR:PREF:CSET:SAVE USER  
sense:correction:preference:cset:save reuse
```

Query Syntax SENSe:CORRection:PREFereNce:CSET:SAVE?

Return Type Character

Default CALRegister

SENSe:CORRection:PREFeRence:CSET:SAVUser <bool>

Superseded

This command is replaced 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](#).

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](#).

Examples

```
SENS:CORR:PREF:CSET:SAVU 1  
sense:correction:preference:cset:savuser 0
```

Query Syntax SENSe:CORRection:PREFeRence:CSET:SAVUser?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF (0)

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:REF: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:REF: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:OVERrange[:STATe] <ON|OFF>

(Read-Write) Sets and returns the ECAL over range state.

Parameters

<bool> ECAL over range state. Choose from:
ON or **1** - Enable ECAL over range.
OFF or **0** - Disable ECAL over range.

Examples

```
SENS:CORR:PREF:ECAL:OVER OFF  
sense:correction:preference:ecal:overrange:state on
```

Query Syntax SENSe:CORRection:PREFeRence:ECAL:OVERrange[: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:

ECAL1

.through.

ECAL50

<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

If either the receive port or source port (or load port for 2-port cal) of the **CALC:PAR:SELEcted** measurement is not in this string and orientation is OFF, an attempt to perform an ECal calibration will fail.

Examples `SENS:CORR:PREF:ECAL:PMAP ECAL2, 'A1,B2'`
`sense:correction:preference:ecal:pmap ecal3, 'a2,b1,c3'`

Query Syntax `SENSe:CORRection:PREFeRence:ECAL:PMAP? <module>`

Return Type String

Default Null string ()

SENSe:CORRection:PREFerence:SIMCal <bool> **Obsolete**

This command is no longer supported. [Learn more about old and new behaviors.](#)

(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/Keysight/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 **SENSe:CORR:COLL:SAVE**.
 - 1 - ON** - (WinCal compatible) Prevents **SENSe:CORR:COLL:SAVE** from failing when standard data has not, and will not, be acquired.

Examples

```
SENSe: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

```
SENS:CORR:PREF:TRIG:FREE GUID,1
```

```
sense:correction:preference:trig:free unguided,0
```

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

```
SENS:CORR:RPOW:OFFS .5  
sense2:correction:rpower:offset:amplitude .-5
```

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**
(.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

```
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>:CORRection:TCOLd: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

```
SENS:CORR:TCOL:USER:VAL 295  
sense2:correction:tcold:user:value 298  
See an example using this command
```

Query Syntax SENSe<cnum>:CORRection:TCOLd:USER:VALue?

Return Type Numeric

Default Not Applicable

SENSe<cnum>:CORRection: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

```
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>

Return Type Comma-separated string

Default Not Applicable

Last modified:

- 16-Jan-2014 Obsolete WinCal Pref command
- 16-Jan-2014 Fixed link at ENR:CAI:Table:Data
- 16-Jan-2014 Added note to Obsolete Isolation command
Reordered list of commands
- 16-Jan-2014 Removed errant link to Window:STATE
Updated ACQUIRE command with ECal# and char
Fixed TCOLd example
Added Cal method Sparsolt3
- 16-Jan-2014 Edited Pref:Save
- 16-Jan-2014 Edited TRL limitations
Added SENS:CORR:METH:MATC

16-Jan-2014	Updated confidence check to 12 User chars
16-Jan-2014	Added Noise commands and Normalization method. (9.1)
16-Jan-2014	Added Cal Pref FOM
16-Jan-2014	Added Acq, sync
16-Jan-2014	Added 'if meas deleted' to Swe:Chan
16-Jan-2014	Added Noise TCOLd
16-Jan-2014	Added missing <cnum> arguments
January 16, 2014	Added ENR commands
January 16, 2014	Add 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**

| **CLISt?**

| **DMEMory**

| **CLEAr**

| **IMPort**

| **EXPort**

| **INFormation**

| **KNAMe**

| **INFormation**

| **LIST?**

| **ORient?**

| **PATH**

| **COUNT?**

| **DATA?**

EXPort

IMPort

INITialize

LOAD

- Click on a **red** keyword to view the command details.
- Red is a superseded command
- **New** See Calibrating the PNA Using SCPI
- Learn about Modifying Cal Kits
- Synchronizing the Analyzer and Controller

- SCPI Command Tree



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:ckit: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>:CLISt?

(Read-only) Returns a list of characterizations stored in the specified ECal module.

Parameters

<mod> ECal module from which to read user characterization numbers. Choose from 1 to 50. 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.

**SENS:CORR:CKIT:ECAL:DMEM:CLE "N4433A
MyUserChar ECal 00001"**

'All User characterizations are deleted from disk memory.

SENS:CORR:CKIT:ECAL:DMEM:CLE

Query Syntax Not Applicable

Default Not Applicable

SENSe:CORRection:CKIT:ECAL:DMEMory:IMPort <file>

(Write-only) 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

```
SENS:CORR:CKIT:ECAL:DMEM:IMP  
"c:\users\public\network analyzer\ECal User  
Characterizations/myDiskUserChar.euc"
```

Query Syntax Not Applicable

Default Not Applicable

SENSe:CORRection: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:IMPort** 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/Keysight/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.

Note: If this argument is specified, the second argument (<file>) must also be specified.

Examples

These examples all use "MyUserChar" as the User characterization name.

All parameters specified

```
SENS:CORR:CKIT:ECAL:EXP "N4433A MyUserChar ECal  
00001", "myUserChar.euc", "NewUserChar"
```

First two parameters are specified

```
sense:correction:ckit:ecal:export "N4691B MyUserChar ECal  
00500", "myUserChar.euc"
```

'Only first parameter is specified

**SENS:CORR:CKIT:ECAL:EXP "N4433A MyUserChar ECal
00001"**

Query Syntax Not Applicable

Default Not Applicable

SENSe:CORRection:CKIT:ECAL<mod>:INFormation? [<char>]

(Read-only) Reads the identification and characterization information from the specified ECal module.

Note: To read user-characterization information that is stored in PNA disk memory, then use **SENS:CORR:CKIT:ECAL:KNAM:INF?**

Parameters

<mod> ECal module from which to read characterizations. Choose from 1 through 50. If unspecified, value is set to 1.

Do NOT assume the <mod> number is the order in which ECal modules were connected.

Use **SENS:CORR:CKIT:ECAL:LIST?** to read a list of <mod> numbers of currently-attached ECal modules.

<char> Optional argument. Specifies which characterization 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 Keysight)
- CHAR1 User characterization #1
- CHAR2 User characterization #2
- - through -
- CHAR12 User characterization #12

Examples **SENS:CORR:CKIT:ECAL2:INFormation? char5**

'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) Reads the identification and characterization information from the specified ECal module or PNA disk memory.

[Learn more about User Characterization in PNA Disk Memory.](#)

Parameters

<kitName> String. ECal model and characterization to read information from, enclosed in quotes, in the following format:

<model> <name> **ECal** <serial number>

Where:

<model>: Always required

<name>:

- For the factory characterization, do not specify.
- For a user-characterization stored in the module, use **User <n>** in the string, where <n> is the user-characterization number. Not case sensitive. Separate User and <n> with a space.
- For a user-characterization stored in PNA disk memory, use <charName> from **SENS:CORR:CKIT:ECAL:CHAR:DMEM:SAVE <charName>**

ECal - not case sensitive

<serial number>: Optional. Include when two or more ECal modules with same model number are attached to the PNA,

Each item is separated with a space.

Examples

'For a factory characterization in module memory:

```
SENS:CORR:CKIT:ECAL:KNAM:INF? "N4433A ECal"
```

'For user characterization in module memory with optional serial number:

```
SENS:CORR:CKIT:ECAL:KNAM:INF? "N4433A User 1 ECal 00028"
```

'For user characterization "foo" in disk memory:

**SENS:CORR:CKIT:ECAL:KNAM:INF? "N4433A foo ECal
00028"**

'Example return string:

**"ModelNumber: N4433A, SerialNumber: 00028,
ConnectorType: N5FN5F, PortAConnector: Type N (50) female,
PortBConnector: Type N (50) female, MinFreq: 30000,
MaxFreq: 9100000000, NumberOfPoints: 250, Calibrated: July
4 2002"**

Return Type String

Default Not Applicable

SENSe:CORRection:CKIT:ECAL:LIST?

(Read-only) Returns a list of index numbers for ECal modules that are currently attached to the PNA. Use these numbers (called <mod> in PNAHelp) to refer to the ECal module using SCPI commands.

Examples **SENS:CORR:CKIT:ECAL:LIST?**
'If 2 modules are attached to the PNA
'then the returned list will be:
+1,+2
'If NO modules are attached to the PNA
'then the returned list will be:
+0
See example program using this command.

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. Choose from 1 through 50.

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 Keysight)
- **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

```
SENS1:CORR: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. Choose from 1 to 50.

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)
- **AD** (4-port modules)
- **BC** (4-port modules)
- **BD** (4-port modules)
- **CD** (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. Choose from 1 through 50.

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.

<stateNum>	N4432A and N4433A States	N4431A States	N469x States**	8509x States
One-Port Reflection States				
1	Open	Open	Impedance 1	Open
2	Short	Short	Impedance 2	Short
3	Impedance 1	Impedance 1	Impedance 3	Impedance 1
4	Impedance 2	Impedance 2	Impedance 4	Impedance 2
5			Impedance 5	
6			Impedance 6	
7			Impedance 7	

Two-Port Transmission States

TWO-PORT TRANSMISSION STATES				
1	Thru	Thru	Thru	Thru
2	Confidence	Confidence	Confidence	Confidence

** 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 Keysight)
- **CHAR1** User characterization #1
- **CHAR2** User characterization #2

and so forth up to:

- **CHAR12** User characterization #12

Examples **SENS:CORR:CKIT:ECAL1:PATH:DATA? A,1**

Return Type S1P or S2P

Default Not Applicable

SENSe:CORRection: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/Keysight/Network Analyzer/PNACalKits/User.

Examples

'File unspecified'

```
SENS:CORR:CKIT:EXP "MyCalKit"
```

'Both parameters are specified'

```
sense:correction:ckit:export  
"MyCalKit", "C:/myBackupCalKit.ckt"
```

Query Syntax Not Applicable

Default Not Applicable

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.

Note: Although there is no limit to the number of cal kits that can be imported, during an **Unguided cal**, you can access ONLY mechanical cal kits #1 through #95.

Parameters

<string> Path and cal kit name.

Examples

```
SENSe:CORRection:CKIT:IMPort "c:\users\public\network  
analyzer\documents\85033D.ckt"
```

Query Syntax Not Applicable

Default Not Applicable

SENSe:CORRection:CKIT:INITialize[:IMMEDIATE] [ckit]

(Write-only) Restores default factory installed cal kits. This command also selects kit number 1, as you would using **SENS:CORR:COLL:CKIT:SEL 1**. Therefore, if you intend to work with a Cal Kit remotely, select the Cal Kit **AFTER** sending this command.

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

```
SENS:CORR:CKIT:INITialize  
sense:correction:ckit:initialize:immediate "85052B"
```

Query Syntax Not Applicable

Default Not Applicable

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/Keysight/Network Analyzer/PnaCalKits/factory/wMyCalKits.wks"`

Query Syntax Not Applicable

Default Not Applicable

Last modified:

17-Jan-2014	Removed superseded from INF? command
5-Nov-2013	Edits to SENS:CORR:CKIT:INIT
16-Aug-2013	Updated ECal nums
23-Jan-2013	Fixed several string examples
12-Oct-2011	Edit Import command
3-Aug-2011	Edit ECal list
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:Ckit Commands

Use to change the definitions of calibration kit standards.

SENSe:CORRection:COLLect:CKIT:

- | **CATalog?**
- | **CONNector**
 - | **ADD**
 - | **CATalog?**
 - | **DELeTe**
 - | **FNAME**
 - | **SNAME**
- | **DESCRiption**
- | **INFormation?**
- | **NAME**
- | **OLAB**
- | **OLISt?**
- | **ORDer**
- | **PORT[:SElect]**
- | **RESet**
- | **SElect**
- | **STANdard**
 - | **CO, C1, C2, C3**
 - | **CHARacter**
 - | **DELay**
 - | **FMAXimum**
 - | **FMINimum**
 - | **IMPedance**
 - | **LO, L1, L2, L3**
 - | **LABel**
 - | **LOSS**
 - | **REMove**
 - | **SDEscription**

[SElect]
TYPE
TZReal
TZImag
TRLOption
IMPedance
LRLChar
RPLane

Click on a keyword to view the command details.

Blue 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 Analyzer 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 Keysight Vector Network Analyzers (Application Note 1287-11)**

SENSe:CORRection:COLLect:CKIT:CATalog?

(Read-only) Returns the names of the first 95 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:CORRection:COLLect:CKIT:CONNector:ADD **<family>,<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 cal 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

SENSe:CORRection:COLLect:CKIT:CONNector:SEtNAME.

Examples **SENSe:CORRection:COLLect:CKIT:CONNector: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 **SENSe:CORR:COLL:CKIT:CONN:SNAME "Type-N (50)",MALE,1**

Query Syntax SENSe:CORRection:COLLect:CKIT:CONNector:SNAME?

Return Type String

Default Not Applicable

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

```
SENS:CORR:COLL: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

<module> Specifies which ECal module to read from. Choose from:

ECAL1

.through.

ECAL50

[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 Keysight)

CHAR1 User characterization #1

CHAR2 User characterization #2

- through -

CHAR12 User characterization #12

Examples

```
SENS:CORR:COLL:CKIT:INF? ECAL4  
sense:correction:collect:ckit:information? ecal2,char1
```

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: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

```
SENS:CORR:COLL: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:

	Class	Description
Port 1		
1	SA	Reflection standard
2	SB	Reflection standard
3	SC	Reflection standard
4	FWD TRANS	Thru/Delay standard
Port 2		
5	SA	Reflection standard
6	SB	Reflection standard
7	SC	Reflection standard
8	REV TRANS	Thru/Delay standard

3-port analyzers only

Port 3		
9	S33A	Reflection standard
10	S33B	Reflection standard
11	S33C	Reflection standard
12	S32T	Thru/Delay standard
13	S23T	Thru/Delay standard

14	S31T	Thru/Delay standard
15	S13T	Thru/Delay standard

TRL Calibrations

16	TRL "T"	Thru standard
17	TRL "R"	Reflect standard
18	TRL "L"	Line standard

<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.

This command ALWAYS applies to the Cal Kit that is selected (using SENS:CORR:COLL:CKIT:SEL) when this ORDER command is sent.

Parameters

<class> Number of the calibration class to be queried. The <class> numbers are associated with the following calibration Classes:

	Class	Description
Port 1		
1	SA	Reflection standard
2	SB	Reflection standard
3	SC	Reflection standard
4	FWD TRANS	Thru/Delay standard
Port 2		
5	SA	Reflection standard
6	SB	Reflection standard
7	SC	Reflection standard
8	REV TRANS	Thru/Delay standard

3-port analyzers ONLY (N3381A/2A/3A)

4-port analyzers use S11 and S22 classes (see example program)

Port 3		
9	S33A	Reflection standard
10	S33B	Reflection standard
11	S33C	Reflection standard
12	S32T	Thru/Delay standard

13	S23T	Thru/Delay standard
14	S31T	Thru/Delay standard
15	S13T	Thru/Delay standard

TRL Calibration

16	TRL "T"	Thru standard
17	TRL "R"	Reflect standard
18	TRL "L"	Line standard

Examples

SENS:CORR:COLL:CKIT:OLIS8?

Always returns 7 standard numbers. Unassigned standards return 0

Return Type Numeric; returns the <class> number of the selected standard.

Default Not Applicable

SENSe:CORRection:COLLect:CKIT:ORDeR<class> <std> [, <std>] [,<std>] [,<std>] [,<std>] [,<std>] [,<std>]

(Read-Write) Sets a standard number to a calibration class. This command does **NOT** set or dictate the order for measuring the standards. For more information, see Assigning Standards to a Calibration Class.

This command **ALWAYS** applies to the Cal Kit that is selected (using **SENSe:CORR:COLL:CKIT:SEL**) when this **ORDeR** command is sent.

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:

	Class	Description	STAN#
Port 1			
1	SA	Reflection standard	STAN1
2	SB	Reflection standard	STAN2
3	SC	Reflection standard	STAN3
4	FWD TRANS	Thru/Delay standard	STAN4
Port 2			
5	SA	Reflection standard	STAN1
6	SB	Reflection standard	STAN2
7	SC	Reflection standard	STAN3
8	REV TRANS	Thru/Delay standard	STAN4

3-port analyzers ONLY (N3381A/2A/3A)			
4-port analyzers use S11 and S22 classes (see example program)			
Port 3			

9	S33A	Reflection standard	STAN1
10	S33B	Reflection standard	STAN2
11	S33C	Reflection standard	STAN3
12	S32T	Thru/Delay standard	STAN4
13	S23T	Thru/Delay standard	STAN4
14	S31T	Thru/Delay standard	STAN4
15	S13T	Thru/Delay standard	STAN4

TRL Calibration

16	TRL "T"	Thru standard	STAN4
17	TRL "R"	Reflect standard	STAN1
18	TRL "L"	Line standard	STAN3

<std> Standard number to be assigned to the class; Choose a standard between 1 and 30. One standard is mandatory; up to six additional standards are optional.

Examples

'Assigns standard 3 to S11A class:
SENSe: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.

This command effectively does the same task as **SENS:CORR:COLL:CKIT** but specifies the cal kit by name.

Note: During an **Unguided cal**, you can access ONLY mechanical cal kits #1 through #95. However, there is no limit to the number of cal kits that can be imported.

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 **SENSe:CORR:COLL:CKIT:RESet 1**
sense:correction:collect:ckit:reset 4

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.

Note: During an **Unguided cal**, you can access **ONLY** mechanical cal kits #1 through #95. However, there is no limit to the number of cal kits that can be imported.

This command effectively does the same task as **SENS:CORR:COLL:CKIT:PORT** which specifies the cal kit by name instead of this command which specifies by number.

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.

<num> The number of the calibration kit. Choose from:

Use **SENSe:CORRection:COLLect:CKIT:RESet** to restore Cal Kits to default values.

Name

1 Cal Kit 1

2 Cal Kit 2

3 Cal Kit 3

"

"

94 Cal Kit 94

95 Cal Kit 95

99 ECal module

Examples

```
SENS:CORR:COLL:CKIT 2  
sense2:correction:collect:ckit:select 7
```

Query Syntax SENSe<cnum>:CORRection:COLLect:CKIT?

Return Type Numeric

Default Last kit selected

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.Keysight.com.

Parameters

<num> Value for C0 in femtofarads (1E-15)

Examples

The following commands set C0=15 femtofarads:

```
SENSe:CORR:COLL:CKIT:STAN:C0 15
```

```
sense:correction:collect:ckit:standard:c0 15
```

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.Keysight.com.

Parameters

<num> Value for C1.

Examples

The following two commands set C1=15:

```
SENSe:CORR:COLL:CKIT:STAN:C1 15
```

```
sense:correction:collect:ckit:standard:c1 15
```

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.Keysight.com.

Parameters

<num> Value for C2.

Examples

The following two commands set C2:

```
SENSe: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.Keysight.com.

Parameters

<num> Value for C3.

Examples

The following two commands set C3

```
SENSe: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:CORR: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>

(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>

(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.Keysight.com.

Parameters

<num> Value for L0 in femtohenries (1E-15)

Examples

The following two commands set L0=15 femtohenries:

```
SENSe: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.Keysight.com.

Parameters

<num> Value for L1.

Examples

The following two commands set L1=15:

```
SENSe: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.Keysight.com.

Parameters

<num> Value for L2.

Examples

The following two commands set L2=15:

```
SENSe: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.Keysight.com.

Parameters

<num> Value for L3.

Examples

The following two commands set L3=15:

```
SENSe: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 **SENS:CORR:COLL:CKIT:STAN:SDS "My New Standard"**

Query Syntax SENSE:CORRection:COLLect:CKIT:STANdard:SDEscription?

Return Type String

Default Not Applicable

SENSe:CORRection:COLLect: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 **SENSe:CORR:COLL:CKIT:SEL**

Parameters

<num> Number of the standard. Choose any number between:
1 and 30

Examples

```
SENSe:CORR:COLL:CKIT:STAN 3  
sense:correction:collect:ckit:standard:select 8
```

Query Syntax SENSe:CORRection:COLLect:CKIT:STANdard[:SELEct]?

Return Type Numeric

Default 1

SENSe:CORRection:COLLect: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)
DATAbased (data-based)

Examples

```
SENSe: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

SENSe:CORRection:COLLect:CKIT:TRLOption:IMPedance <char>

(Read-Write) Sets the reference impedance when using this TRL cal kit. [Learn more.](#)

Before sending this command, select a cal kit using [SENS:CORR:COLL:CKIT:SElect](#).

Parameters

<char> Choose from:

SYSTEM - The system impedance is used as the reference impedance. During a Guided or Unguided Cal, the Z0 of the Cal standard's connector definition sets the System Z0.

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 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.

Examples

```
SENS:CORR:COLL:CKIT:TRL:IMP SYST
```

```
sense:correction:collect:ckit:trloption:impedance line
```

Query Syntax SENSe:CORRection:COLLect:CKIT:TRLOption:IMPedance?

Return Type Character

Default LINE

SENSe:CORRection:COLLect:CKIT:TRLOption:LRLChar **<bool>**

(Read-Write) This setting **ONLY** applies if an LRL Cal Kit is being modified **AND** Testport Reference Plane is set to THRU **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.

Before sending this command, select a cal kit using **SENS:CORR:COLL:CKIT:SElect**.

Parameters

<bool> Choose from:
1 or **ON** - Automatically correct for line loss and dispersion characteristics.
0 or **OFF** - Select when anomalies appear during a calibrated measurement which may indicate different loss and impedance values for the Line standards.

Examples

```
SENS:CORR:COLL:CKIT:TRL:LRLC 1  
sense:correction:collect:ckit:trloption:lrchar off
```

Query Syntax SENSE:CORRection:COLLect:CKIT:TRLOption:LRLChar?

Return Type Boolean

Default OFF

SENSe:CORRection:COLLect:CKIT:TRLOption:RPLane <char>

(Read-Write) Sets the reference impedance when using this cal kit. [Learn more](#).

Before sending this command, select a cal kit using `SENSe:CORR:COLL:CKIT:SElect`.

Parameters

<char> Choose from:

THRU The THRU standard definition is used to establish the measurement reference plane. Select if the THRU standard is zero-length or very short.

REFlect 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.

Examples

```
SENSe:CORR:COLL:CKIT:TRL:RPL THRU  
sense:correction:collect:ckit:trloption:rplane reflect
```

Query Syntax SENSe:CORRection:COLLect:CKIT:TRLOption:RPLane?

Return Type Character

Default THRU

Last Modified:

28-Mar-2014	Edit to OLISt
5-Nov-2013	Edit to ORD command. Extend to 30 stds.
16-Aug-2013	Updated Ecal nums
21-Dec-2012	Added TRL commands
12-Oct-2011	Select Cal kit edits
11-Jan-2011	Minor edits
17-Mar-2010	Added Ckit:CAT and CKIT:Select by string (9.2)
10-Nov-2008	Clarified Select command

30-Oct-2008 Fixed SDES query per CN

14-Apr-2008 Added link to app note

17-Sep-2007 Fixed 'select' command

Sense:Correction:Cset Commands

Performs actions on calibration sets.

SENSe:CORRection:CSET

ACTivate

CATalog?

COPY

CREate

DATA

DEACTivate

DELete

DESCription

ETERm

| **CATalog?**

| **[DATA]**

FLATten

GUID

ITEM

| **CAT?**

| **[[:DATA]?**

NAME

SAVE

[SElect]

STANdard

STIMulus?

TSET

| **ALLPorts?**

| **TYPE?**

TYPE

| **CATalog?**

Click on a keyword to view the command details.

[Blue](#) keywords are superseded commands. Learn more.

see Also

- [Creating Cal Sets](#)
 - [Example Programs](#)
 - [Learn about Cal Sets](#)
 - [Synchronizing the Analyzer 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.

Return Type String

Default Not Applicable

SENSe:CORRection:CSET:CATalog? [char] - **Superseded**

This command is replaced by **CSET:CAT?**

(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

Default Not Applicable

SENSe<cnum>:CORRection:CSET:COPIY <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:COPIY 'My2Port'**

Query Syntax Not Applicable

Default Not Applicable

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 **SENS:CORR:CSET:CRE 'My2Port'**

Query Syntax Not Applicable

Default Not Applicable

SENSe<cnum>:CORRection:CSET:DATA <eterm, portA, portB,>[<rec>,<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.

<rec> <string> - Specify the PNA receiver for which the Eterm applies.

Required ONLY when Eterm is response tracking (**ERSPT**) or response isolation (**ERSPI**).

Logical receiver notation is allowed.

A full 4-port calibration requires the following terms be uploaded:

		PORT B			
		1	2	3	4
P O R T	1	EDIR,1,1	ELDM,1,2	ELDM,1,3	ELDM,1,4
	ERFT,1,1	ETRT,1,2	ETRT,1,3	ETRT,1,4	
	ESRM,1,1	EXTLK,1,2	EXTLK,1,3	EXTLK,1,4	
2	ELDM,2,1	EDIR,2,2	ELDM,2,3	ELDM,2,4	
ETRT,2,1	ERFT,2,2	ETRT,2,3	ETRT,2,4		

A		EXTLK,2,1	ESRM,2,2	EXTLK,2,3	EXTLK,2,4
	3	ELDM,3,1	ELDM,3,2	EDIR,3,3	ELDM,3,4
		ETRT,3,1	ETRT,3,2	ERFT,3,3	ETRT,3,4
		EXTLK,3,1	EXTLK,3,2	ESRM,3,3	EXTLK,3,4
	4	ELDM,4,1	ELDM,4,2	ELDM,4,3	EDIR,4,4
		ETRT,4,1	ETRT,4,2	ETRT,4,3	ERFT,4,4
		EXTLK,4,1	EXTLK,4,2	EXTLK,4,3	ESRM,4,4

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**

```
SENS1:CORR:CSET:DATA EDIR, 1, 1, +6.12569600000E-002,-7.27163800000E-003,-3.63812000000E-003,+1.33521800000E-002,-4.36775100000E-003,+1.87792400000E-002,-4.09239100000E-003,+4.24291200000E-002,-2.03784900000E-002,+3.21425100000E-002"
```

Query Syntax SENSE<cnum>:CORRection:CSET:DATA? <eterm,portA, portB>, <rec>

Query Examples **'Read the response isolation eterms for the port 1 reference receiver**

```
sens:corr:cset:data? ERSPI,1,1,'R1'
```

'Same receiver using logical receiver notation

```
sens:corr:cset:data? ERSPI,1,1,'a1'
```

Return Type Block data

Default Not Applicable



SENSe<cnum>:CORRection:CSET:DEACtivate

(Write-only) Unselects a Cal Set from the specified channel.

Parameters

<cnum> Channel number to have Cal Set unselected.

Examples **SENS:CORR:CSET:DEAC**
sense2:correction:cset:deactivate

Query Syntax Not Applicable

Default Not Applicable

SENSe:CORRection:CSET:DELeTe <string> - **Superseded**

This command is replaced by **CSET:DEL**.

(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 **SENS:CORR:CSET:CAT?** to list the available Cal Sets in either format.

Examples

```
SENS:CORR:CSET:DEL '{2B893E7A-971A-11d5-8D6C-00108334AE96}'  
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 **SENS:CORR: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

```
SENS:CORR:CSET:DESC 'MyCalSet'  
sense2:correction:cset:description 'thisCalSet'
```

Query Syntax SENSe<cnum>:CORRection:CSET:DESCription?

Return Type String

Default Not Applicable

SENSe<cnum>:CORRection:CSET:ETERm[:DATA] <string>, <data>

(Read-Write) Sets or returns error term data for all VNA 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 Keysight 4691 family of Ecal modules, n can range from 1 to 7.

<data> (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<cnum>:CORRection:CSET:ETERm[:DATA]? <string>`

Return Type Block data

Default Not Applicable

SENSe<cnum>:CORRection:CSET:ETERm:CATalog?

(Read-only) Returns a list of error term names found in the current Cal Set that is applied to the specified channel.

Parameters

Examples

```
SENS:CORR:CSET:ETER:CAT?
```

'For a 1-port cal, returns

```
"Directivity(1,1),ReflectionTracking(1,1),SourceMatch(1,1)"
```

Return Type String

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

- <cnum> Channel number on which the modified Cal Set resides. If unspecified, value is set to 1
- <string> Name of the new Cal Set. Spaces or punctuation NOT allowed.

Examples `SENS:CORR:CSET:FLAT "MyCalSet"`

Query Syntax Not Applicable

Default Not Applicable

SENSe<cnum>:CORRection: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

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <string> GUID of the desired Cal Set. The curly brackets and hyphens must be included.

Examples

```
SENS:CORR:CSET:GUID '{2B893E7A-971A-11d5-8D6C-00108334AE96}'  
sense2:correction:cset:guid '{2B893E7A-971A-11d5-8D6C-00108334AE96}'
```

Query Syntax

SENSe<cnum>:CORRection:CSET:GUID?

Returns the GUID of the currently-selected Cal Set for the specified channel.

Return Type

String

Default Not Applicable

SENSe<cnum>:CORRection:CSET:ITEM:CATalog?

(Read-only) Returns a list of all name-value pairs (items) in the Cal Set.

Parameters

Examples SENS:CORR:CSET:ITEM:CAT?

Return Type String

Default Not Applicable

SENSe<cnum>:CORRection:CSET:ITEM[:DATA] <string>, <data>

(Read-Write) Add or change a name-value pair in the Cal Set, or read the value associated with a name.

After editing, Save the CalSet to the VNA.

About Name-Value pairs

A Cal Set name-value pair is a general purpose data structure that maps a name to a value. This command allows you to associate a name with a value. Then, using this same command, you can read the value using the name.

For example, one of the items added by the VNA firmware to every Cal Set is named 'Created By'. The value attached to this item is the name of the VNA 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 VNA firmware may behave unpredictably.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1

<string> (String) Name of the name-value pair.

<data> (Variant) - Can be an integer, float, double, string, or a single-dimensioned array of integer, float, double, string.

Examples **SENS:CORR:CSET:ITEM**

Query Syntax SENSe<cnum>:CORRection:CSET:ITEM[:DATA]? <string>

Return Type Variant

Default Not Applicable

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

```
SENS: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: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 **SENS: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 **SENS:CORR:CSET:DATA** (For Advanced Users).

The file name is saved as "**CSETx.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>] Optional argument. Choose from:

USER01

USER02...

and so forth, until...

USER10

If <char> is NOT specified, changes that may have been made are saved to the cal set and NOT to the *.cst file.

Examples

```
SENS:CORR:CSET:SAVE USER03
```

```
sense2:correction:cset:save user09
```

```
'save changes to only the cal set
```

```
SENS:CORR:CSET:SAVE
```

Query Syntax SENSe<cnum>:CORRection:CSET:SAVE?

Queries the last correction set saved.

Return Type Character

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 [SENS: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

Examples

```
SENS:CORR:CSET DEF  
sense2:correction:cset:select user02
```

Query Syntax SENSe<cnum>:CORRection:CSET[:SElect]?

Return Type Character

Default DEF

SENSe<cnum>:CORRection:CSET:STANdard <string>,<data>

(Read-Write) Sets or returns standard data. Standard data is available for Unguided Cals ONLY.

Note: The “Standards data” container in the calset is intended for internal use only. External access is provided for use in diagnosing calibration problems. Users should not form any expectations as to the presence of the data or the naming conventions used.

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 **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

```
SENS:CORR:CSET:STIM?  
sense:correction:cset:stimulus 1
```

Return Type Numeric

Default Not Applicable

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"`
`sens:correction:cset:tset:allports? "{2B893E7A-971A-11d5-8D6C-00108334AE96}"`

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"
sens:correction:cset:tset:type? "{2B893E7A-971A-11d5-8D6C-00108334AE96}"

Return Type String

Default Not Applicable

SENSe<ch>:CORRection: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:CORRection:CSET:TYPE:CAT?
```

Return Type String

Default Not Applicable

Sense:Correction:Extension Commands

Performs and applies Port Extensions.

SENSe:CORRection:EXTension:

AUTO

- | **CONFig**
- | **DCOFFset**
- | **LOSS**
- | **MEASure**
- | **PORT**
- | **RESet**
- | **STARt**
- | **STOP**

PORT

- | **DISTance**
- | **FREQuency**
- | **INCLude**
 - | **[STATe]**
- | **LDC**
- | **LOSS**
- | **MEDium**
- | **SYSMedia**
- | **SYSVelocity**
- | **[TIME]**
- | **UNIT**
- | **VELFactor**
- | **WGCutoff**

RECeiver

- | **[TIME]**
- [STATe]**

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
 - [Learn about Port Extensions](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

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.

USPN - Use custom user span. Use

SENS:CORR:EXT:AUTO:STAR and

SENS:CORR:EXT:AUTO:STOP to specify start and stop frequency.

Examples

```
SENS:CORR:EXT:AUTO: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.
OFF (or 0) - Does NOT include 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.

OFF (or 0) - Does NOT include 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. This command should be preceded by the CALCulate:PARAmeter:MNUMber <num> where num is the trace number of a measurement on the specified channel. [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<cnum>: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<cnum>:CORRection:EXTeNsion:AUTO:STARt <value>?

Return Type Numeric

Default Start frequency of the current active channel.

SENSe<cnum>:CORRection:EXTension:AUTO:STOP <value>

(Read-Write) Set the stop 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 stop value. Must be within the frequency range of the active channel and greater than the value set by SENS:CORR:EXT:AUTO:START

Examples

```
SENS:CORR:EXT:AUTO:STOP 1E9  
sense2:correction:extension:auto:stop 200e6
```

Query Syntax SENSe<cnum>:CORRection:EXTension:AUTO:STOP <value>?

Return Type Numeric

Default Stop frequency of the current active channel.

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:DISTanc<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.
- <value>** Physical length of fixture of added transmission line. First specify units with **SENS:CORR:EXT:PORT:UNIT**.

Examples

```
SENS: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>:FREQue <value>

(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>:CORRection:EXTension:PORT<pnum>:INCLude[: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 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[:STAT

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

```
SENS: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

Examples

```
SENS:CORR:EXT:PORT:LOSS1 1  
sense2:correction:extension:port2:loss2 .1
```

Query Syntax SENSe<cnum>:CORRection:EXTension:PORT<pnum>:LOSS<n>?

Return Type Numeric

Default 0

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

```
SENS:CORR:EXT:PORT:MED COAX  
sense2:correction:extension:port2:medium waveguide
```

Query Syntax SENSe<cnum>:CORRection:EXTeNsion:PORT<pnum>:MEDium?

Return Type Character

Default COAX

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:SYSMed <bool>

(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>:SYSVelo <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

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
```

Query Syntax SENSe<cnum>:CORRection:EXTension:PORT<pnum> [:TIME]?

Return Type Numeric

Default 0

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
```

Query Syntax SENSe<cnum>:CORRection:EXTension:PORT:UNIT?

Return Type Character

Default METer

SENSe<cnum>:CORRection:EXTension:PORT<pnum>:VELFact <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

```
SENS:CORR:EXT:PORT:VELF .6  
sense2:correction:extension:port2:velfactor 1
```

Query Syntax SENSe<cnum>:CORRection:EXTension:PORT<pnum>:VELFactor?

Return Type Numeric

Default System Velocity Factor

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>:WGCutoff?

Return Type Numeric

Default System Media Cutoff Frequency

SENSe<cnum>: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

- <cnum> 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<cnum>:CORRection:EXTension:RECeiver<Rnum>
[:TIME]?

Return Type Numeric

Default 0

SENSe<cnum>: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:

17-Mar-2014	WAVEguide to WAVEguide
26-Aug-13	Another minor edit
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.

Important Notes:

- To perform a **Guided Calibration**, use ONLY **Sens:Corr:Coll:Guided** commands. See the "**Guided**" **example programs** for clarification.
- ALWAYS send ALL measurement setup commands BEFORE initializing a remote calibration.

SENSe:CORRection:COLLect:GUIDed:

ABORt

ACQuire

ADAPter

| **COUNt**

| **ZERO**

| **CREate?**

| **DELay**

| **DESCRiption**

| **PATHs**

CHANnel:MODE

CKIT

| **CATalog?**

| **PORT**

| **CATalog?**

| **[SElect]**

CONNector

| **CATalog?**

| **PORT**

| **[SElect]**

DESCRiption

ETERms

| **COMPute**

| **LOAD[:CSET]**

INITiate

ISOLation

| **AVERage**

| **INCRement**

| **PATHs**

ITERations

| **COUNT?**

| **MINimum?**

| **RESet**

METHod

PACQuire

PATH

| **CMETHod**

| **TMETHod**

PREFerence

| **SLIDingload**

SAVE

| **CSET**

STEPS?

THRU

Click on a keyword to view the command details.

Blue keywords are superseded commands.

see Also

- [ECal Orientation commands](#)
 - [Examples](#) using these commands.
 - [Calibrating the PNA Using SCPI](#)
 - [Learn about Measurement Calibration](#)
 - [Synchronizing the Analyzer and Controller](#)
-

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] STAN<n>[,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.

STAN<n> Choose from:STAN1, STAN2, etc. through STANn where n is the number of cal standard connection steps for the calibration.

Note: You do not necessarily have to invoke these connection steps in sequential order, but you must issue this command for all of the steps to be able to complete 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 about this argument](#)

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>

(Read-only) 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

(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  
sense2:correction:collect:guided:adapter:count:zero
```

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>: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

SENSe<ch>:CORRection:COLLect:GUIDed:ADAPter<n>:DESC <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>:PATH <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

Default 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.

ON (1) Legacy behavior. Behavior is specified by the following table:

<ch> channel type Std or App	Active channel type Std or App	Behavior
Std	Std	Active chan cal'd
Std	App	"Channel not found" error
App	Std	<ch> chan cal'd
App	App	<ch> chan cal'd

Learn about Standard vs Application channels.

Examples **SENS: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 OFF - This is the default beginning with A.09.50
ON - Default before A.09.50

SENSe:CORRection:COLLect:GUIDed:CKIT:CATalog? <connector>

(Read-only) This command replaces **SENS:CORR:COLL:GUID:CKIT:PORT:CAT?**

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> String. Connector type. Use **SENS:CORR:COLL:GUID:CONN:CAT?** to return a list of valid connector types.

Examples **SENSe:CORR:COLL:GUID:CKIT:CAT? "Type N (50) male"**

Return Type String

Default Not Applicable

SENSe:CORRection:COLLect:GUIDed:CKIT:PORT<pnum>:CA **Superseded**

(Read-only) This command is replaced by **SENSe: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

```
SENS:CORR:COLL:GUID:CKIT:PORT1:CAT?
```

'When "Type N (50) male" is specified for connector type, returns:

```
"85054D, 85032F"
```

'When two identical ECal modules are connected for the connector type, the return string includes serial numbers

```
"85092-60010 ECal 10675, 85092-60010 ECal 00758"
```

Return Type String

Default Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:CKIT:PORT<pnum>[:SELection] <kit>

(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.

1. Specify the connector type for the port with **SENSe:CORR:COLL:GUID:CONN:PORT**.
2. Query the valid available kits for the connector on each port with **SENSe: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.

When using this command to specify the cal kit for the output of a VMC calibration mixer, specify port 3. If port 3 is already used for the output of the DUT mixer, then specify port 4. Learn more.

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

SENSe:CORR:COLL:GUID:CKIT:PORT1 '85055A'

' Standard ECal modules

SENSe:CORR:COLL:GUID:CKIT:PORT1 "N4691-60004

ECal"

' **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"**

Query Syntax SENSE:CORRection:COLLect:GUIDed:CKIT:PORT<pnum>
[:SElect]?

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 **SENS:CORR:COLL:GUID:CONN:PORT**

Here are the more common connector types:

W-band waveguide	Type B	1.00 mm female
V-band waveguide	Type A (50) female	1.00 mm male
U-band waveguide	Type A (50) male	1.85 mm male
R-band waveguide	Type F (75) female	1.85 mm female
Q-band waveguide	Type F (75) male	2.92 mm female
K-band waveguide	Type N (75) female	2.92 mm male
P-band waveguide	Type N (75) male	APC 2.4 female
X-band waveguide	Type N (50) female	APC 2.4 male
7-16 female	Type N (50) male	APC 3.5 female
7-16 male		APC 3.5 male
		APC 7

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<[:SElect] <conn>

(Read-Write) Specifies a DUT connector type for every port during the Guided Calibration procedure. Valid DUT connector names are stored within calibration kits. Some cal kits may include both male and female DUT connectors. Therefore, specifying the DUT connector gender may be required.

The PNA remembers previous Guided Cal settings. Therefore, for completeness, unused ports can either be defined as "Not used" or use the **SENS:CORR:COLL:GUID:ABORt** command to clear all ports. The ABORt command is a more thorough approach and more convenient. 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:

1. Use **SENS:CORR:COLL:GUID:CONN:CAT?** to query available connectors before specifying the port connector.
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:ETERms:COMPUte [cal set name]

(Write-only) Computes the error correction terms, turns Correction ON, and saves the calibration to an existing, specified Cal Set.

The cal acquisition process does not conclude as with the **SAVE** command. This command leaves the cal acquisition in memory to allow re-measuring/re-computing. To conclude the cal acquisition process, use the **SENS:CORR:COLL:GUID:ABOR** command.

[Learn all about Cal Sets.](#)

Note: This command is NOT supported for application channels (Gain Compression, SMC/VMC, Noise Figure, IMD and so forth). Use **SENS:CORR:COLL:GUID:SAVE** and save to a cal register. You can then use **SENS:CORR:CSET:COPY** to copy the cal register to a named Cal Set.

- Use this command instead of specifying the optional name or GUID argument in **SENS:CORR:COLL:GUID:INIT**.
- Use **SENS:CORRection:CSET** commands to get names 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.

For Calibrate All Channels

When this command is used during a Cal All session, the <cal set name> argument sets the User Cal Set prefix. All generated Cal Sets will be preceded with this string name.

- Cal Set prefix can also be set using **SYST:CAL:ALL:CSET:PREFix**. When the Cal Set prefix has already been set with **SYST:CAL:ALL:CSET:PREFix**, this command overwrites it.
- When <cal set name> is an empty string, a User Cal Set will not be saved. Only Cal Registers will be saved.

Parameters

<ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.

<cal set name> **String** - Name of an existing Cal Set to be overwritten.

See [Calibrate All Channels](#) note (above).

Examples

```
SENS:CORR:COLL:ETER:COMP "{2B893E7A-971A-11d5-8D6C-00108334AE96}"
```

```
sense:correction:collect:eterms:compute "MyCalSet"
```

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:ETERms:LOAD[:CS] <cset>,<calPort> [,csPort]

(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.

This command must be sent after the INIT command. This command was implemented to facilitate calibrating a large matrix of external ports and most users will not need to use this command.

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[:IMMediat [string][, bool][,char]

(Write-only) Initiates a guided calibration.

- The PNA determines the measurements needed to perform the calibration using the settings specified from the **SENS:CORR:COLL:GUID:CONN:PORT** and **SENS:CORR:COLL:GUID:CKIT:PORT** commands.
- 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. [See example calibration programs.](#)

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 NOT specified, behavior depends on the **SENS:CORR:PREF:ASET:SAVE** setting. If specified, choose an **existing** Cal Set, either by name or by GUID.

- By Cal Set name: include quotes.
- By Cal Set 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. To set this argument, also set the first optional argument. See example below.

OFF (0) If Cal Set stimulus settings differ from the existing channel, do not change channel stimulus settings. The Cal Set is saved 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.

[char] Optional argument. To set this argument, also set the first two optional arguments. See example below.

SYNChronous - blocks further SCPI commands while processing this command.. (default setting).

ASYNchronous - does NOT block further SCPI commands while processing this command.

[Learn more about this argument](#)

Examples

```
SENS:CORR:COLL:GUID:INIT
```

```
'set first optional argument
```

```
SENS:CORR:COLL:GUID:INIT "MyCalSet"
```

```
'set two optional arguments
```

```
SENS:CORR:COLL:GUID:INIT " ",1
```

```
'set all optional arguments
```

```
SENS:CORR:COLL:GUID:INIT "MyCalSet",1,ASYN
```

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:ISOLation:AVERage:<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 measure isolation

```
sense:correction:collect:guided:isolation:paths REMOVE,1,2,1,3
```

Query Syntax SENSe<ch>:CORRection:COLLect:GUIDed:ISOLation:AVERage:INCR

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 SENSE<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<ch>:CORRection:COLLect:GUIDed:ITERations:COUNT<step>

(Read-only) Designed to be used for an iterative cal standard such as a sliding load, this command returns the number of iterative measurement acquisitions that has been made for the specified step.

Zero (0) is returned if the step has not yet been measured.

For most cal steps that have already been measured, this command returns 1.

Set **SENS:CORR:COLL:GUID:PREF:SLID ITER** to count acquisition steps.

Parameters

<ch> Channel being calibrated, depending on the **CHAN:MODE** setting.
If unspecified, value is set to 1.

<step> Guided Cal step number for which the acquisition number will be returned.

Use **SENS:CORR:COLL:GUID:STEP?** to query the number of steps in the calibration.

Examples SENS:CORR:COLL:GUID:ITER:COUN? 4

Example return:

5

[See example program](#)

Return Type Numeric

Default Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:ITERations:MINImu<step>

(Read-only) Designed to be used for an iterative cal standard such as a sliding load, this command returns the minimum number of required iterative measurement acquisitions for the specified step.

For most connection steps this will return 1, but for an iterative cal standard such as a sliding load, it will return a number such as 5.

Set **SENS:CORR:COLL:GUID:PREF:SLID ITER** to count acquisition steps.

Parameters

- <ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- <step> Guided Cal step number for which to return the number of iterative measurement acquisitions that have been made. Use **SENS:CORR:COLL:GUID:STEP?** to query the number of steps in the calibration.

Examples SENS:CORR:COLL:GUID:ITER:MIN? 4

Example return:

5

[See example program](#)

Return Type Numeric

Default Not Applicable

SENSe<ch>:CORRection:COLLect:GUIDed:ITErations:RESet <step>

(Write-only) Resets the specified guided cal connection step as unmeasured. This clears all previous measurements made for that step.

Parameters

- <ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- <step> Guided Cal step number to reset. Use **SENS:CORR:COLL:GUID:STEP?** to query the number of steps in the calibration.

Examples

SENS:CORR:COLL:GUID:ITER:RESet? 4

[See example program](#)

Return Type Not Applicable

Default Not Applicable

SENSe:CORRection:COLLect:GUIDed:METHod <char>

Superseded

This command is replaced with **SENS:CORR:COLL:GUID:PATH:CMETHod** and **SENS:CORR:COLL:GUID:PATH:TMETHod**.

(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 calcs. Valid for "TRL ready" Cal Kits with properly assigned TRL cal classes.
- SOLT** - Select SOLT Cal Type for guided calcs. Valid for any kit with properly assigned SOLT cal classes.

Examples

```
SENS:CORR:COLL:GUID:METH ADAP  
sense:correction:collect:guided:method unkn
```

Query Syntax SENSe:CORRection:COLLect:GUIDed:METHod?

Return Type Character

Default DEFAULT

SENSe<ch>:CORRection:COLLect:GUIDed:PACQuire STAN<n>

(Write-only) 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 to facilitate the connection of standards.

Parameters

<ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.

STAN<n> Choose from:STAN1, STAN2, etc. through STANn where n is the number of cal standard connection steps for the calibration.

Note: You do not necessarily have to invoke these connection steps in sequential order.

Examples

```
SENS:CORR:COLL:GUID:PACQuire STAN2  
sense:correction:collect:guided:pacquire STAN5  
See an example that uses this command.
```

Query Syntax Not Applicable

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: Sending this command will overwrite the PNAs SmartCal determinations for the most accurate cal method for your connector settings and Cal Kits. Send this command **ONLY** if you have a deliberate reason for overwriting the SmartCal logic. You can send the query form of this command to learn the cal method determined by SmartCal.

See **Thru Pairs Sequence** to learn how to send this and other Thru commands.

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 type for the port pair, enclosed in a single pair of quotes. NOT case-sensitive.
- caltype1** 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.

Examples **SENS:CORR:COLL:GUID:PATH:CMETHOD**
2,3,"QSOLT2"
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: Sending this command will overwrite the PNAs SmartCal determination for the thru method. Send this command **ONLY** if you have a deliberate reason for overwriting the SmartCal logic. You can send the query form of this command to learn the THRU method determined by SmartCal.

See **Thru Pairs Sequence** to learn how to send this and other Thru commands.

[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 port pair, enclosed in a single pair of quotes. NOT case-sensitive.

thruType1 Calibration Thru method. Choose from:

- **Defined Thru** Measures a Thru for which there is a stored definition in the Cal kit of the lowest-numbered port of the pair. For example, if the port pair is 1,2, then the cal kit port 1 **MUST** contain a Defined Thru.
- **Zero Thru** Measures a Zero length Thru, also known as a Flush-Thru.
- **Undefined Thru** (Also known as Unknown Thru) A Thru type for which there is **NOT** a stored definition in the Cal Kit. Valid **ONLY** for SOLT cal type.
- **Undefined Thru using a Defined Thru** (ECal module **ONLY**) Measures the internal Thru as an Unknown Thru.

thruType2 Optional argument. Use **ONLY** when Adapter Removal Cal is specified for the pair using

removal can be specified for the pair using **SENS:CORR:COLL:GUID:PATH:CMETHOD**. When specifying ThruType2, this is the only valid argument: "**Defined Thru, Defined Thru**"

Examples **SENS:CORR:COLL:GUID:PATH:TMETHOD 2,3,"Zero Thru"**
sense:correction:collect:guided:path:tmethod 2,3,"Define Thru,Defined Thru"

Query Syntax SENSE<ch>:CORREction:COLLect:GUIDed:PATH:TMETH<port1>,<port2>

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:PREFerence:SLIDIn <char>

(Read-Write) Specifies the behavior for guided cal steps that involve a sliding load in a cal that is about to be performed. Send this command BEFORE sending the Guided INIT command.

Although the term 'Preference' is used in the command, this is NOT a PNA preference. This setting does NOT survive instrument preset or reboot. It remains ONLY for the duration of the Guided Cal.

Parameters

<char> Behavior when measurements of sliding load are acquired. Choose from:

DIALog - The Sliding load dialog box appears when the acquire command is received for a sliding load step. All slide positions are measured (with a user-interface prompt) from a single invocation of the acquire command.

ITERate - Each invocation of the acquire command for a sliding load step measures a single slide position and increments the slide position counter. No Move Sliding Load prompt is presented on the PNA screen.

Examples **SENS:CORR:COLL:GUID:PREF:SLID ITER**

[See example program](#)

Query Syntax SENSe<ch>:CORRection:COLLect:GUIDed:PREFerence:SLIDingload?

Return Type Character

Default DIALog

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.

[Learn all about Cal Sets.](#)

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.
- For application channels (Gain Compression, SMC/VMC, Noise Figure, IMD and so forth), this command saves ONLY to a Cal Register. Use **SENS:CORR:CSET:COPY** to copy the cal register to a named calset.
 - For a **Calibrate All Channels** session, this argument is ignored. Instead, use **SYST:CAL:ALL:CSET:PREFIX**.

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 an existing, specified Cal Set. This command performs the same function as **SENSe:CORRection:COLLect:GUIDed:SAVE**, except this command allows the name or GUID of the Cal Set to be specified.

[Learn all about Cal Sets.](#)

Note: This command is NOT supported for application channels (Gain Compression, SMC/VMC, Noise Figure, IMD and so forth). Use **SENS:CORR:COLL:GUID:SAVE** and save to a cal register. You can then use **SENS:CORR:CSET:COPY** to copy the cal register to a named Cal Set.

- Use this command instead of specifying the optional name or GUID argument in **SENS:CORR:COLL:GUID:INIT**.
- Use **SENS:CORRection: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.

For Calibrate All Channels

When this command is used during a Cal All session, the <cal set name> argument sets the User Cal Set prefix. All generated Cal Sets will be preceded with this string name.

- Cal Set prefix can also be set using **SYST:CAL:ALL:CSET:PREFix**. When the Cal Set prefix has already been set with **SYST:CAL:ALL:CSET:PREFix**, this command overwrites it.
- When <cal set name> is an empty string, a User Cal Set will not be saved. Only Cal Registers will be saved.

Parameters

<ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.

<cal set name or > **String** - Name or GUID of an existing Cal Set to be overwritten. If

guid> specifying a GUID, curly brackets must be included.
See **Calibrate All Channels** note (above).

Examples

```
SENS:CORR:COLL:GUID:SAVE:CSET "{2B893E7A-971A-11d5-8D6C-00108334AE96}"  
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 **SENS:CORR:COLL:GUID:INIT**, **SENS:CORR:COLL:GUID:CONN:PORT** and **SENS: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. Send the query form of this command to learn the Thru pairs determined by SmartCal.

Note: Sending this command will overwrite the PNAs SmartCal determinations for the thru ports. Send this command ONLY if you have a deliberate reason for overwriting the SmartCal logic.

See [Thru Pairs Sequence](#) to learn how to send this and other Thru commands.

Parameters

- <ch> Channel being calibrated, depending on the **CHAN:MODE** setting. If unspecified, value is set to 1.
- <t1a,t1b...> Always specify port numbers in pairs: For example: 1,2 or 1,2,1,3
- For 3-port cals, specify two or three pairs.
 - For 4-port cals, specify from three up to six pairs.

Examples

```
SENS:CORR:COLL:GUID:THRU:PORT 1,2,1,3,1,4 '4-port  
measurement  
sense:correction:collect:guided: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.

THRU Pairs sequence

The SmartCal logic always determines the best calibration based on your specified connectors and ports.

The following three commands overwrite the SmartCal logic. Send these commands ONLY if you have a deliberate reason for overwriting the SmartCal

logic.

- **SENS:CORR:COLL:GUID:THRU:PORTS** <p1, p2>
- **SENS:CORR:COLL:GUID:PATH:TMET** <p1,p2, thrutype>
- **SENS:CORR:COLL:GUID:PATH:CMET** <p1,p2, calmethod>

When sending one or more of these commands, they must be sent in the following sequence with the other commands listed here.

Note: The **GUID:INIT** command is sent before and after these commands.

1. **SENS:CORR:COLL:GUID:CONN:PORT(N)**
 2. **SENS:CORR:COLL:GUID:CKIT:PORT(N)**
 3. **SENS:CORR:COLL:GUID:INIT**
 4. **SENS:CORR:COLL:GUID:THRU:PORTS** <P1, P2>
 5. **SENS:CORR:COLL:GUID:PATH:TMET** <P1,P2, THRUTYPE>
 6. **SENS:CORR:COLL:GUID:PATH:CMET** <P1,P2, CALMETHOD>
 7. **SENS:CORR:COLL:GUID:PATH:CMET?** <P1,P2> (RECOMMENDED)
 8. **SENS:CORR:COLL:GUID:INIT**
-

Sense: Couple Commands

SENSe:COUPle

|
PARAmeter

|
[STATe]

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
 - [Synchronizing the Analyzer 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

```
SENS:COUP 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 SENSe<cnum>:COUPle:PARAmeter[:STATe]?

Return Type Boolean

Default ON (or 1)

Sense:Frequency Commands

Sets the sweep frequencies of the analyzer.

SENSe:FREQuency

| [CENTer](#)

| [CW](#) | [FIXed](#)

| [SPAN](#)

| [STARt](#)

| [STOP](#)

Click on a keyword to view the command details.

see Also

- [Example](#) using some of these commands.
 - [Learn about Frequency Sweep](#)
 - [Synchronizing the Analyzer 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 from **70** (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

- <cnum> 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 the stop frequency is set to the start frequency + frequency span.
- 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<cnum>:FREQuency:STARt?

Return Type Numeric

Default Minimum frequency of the analyzer

SENSe<cnum>:FREQuency:STOP <num>

(Read-Write) Sets the stop frequency of the analyzer.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <num> Stop frequency. Choose any number between 70 (minimum) and **maximum** frequency limits of the analyzer. Units are Hz.
If **FREQ:STOP** is set less than **FREQ:START**, then the start frequency is set to the stop frequency - frequency span.
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<cnum>:FREQuency:STOP?

Return Type Numeric

Default Maximum frequency of the analyzer

Sense:Roscillator Command

SENSe:ROSCillator:SOURce <state>

(Write-only) Set and read 10 MHz Reference Oscillator state.

Note: This setting is NOT cleared with Preset. However, it does clear when the M937xA software is restarted.

Parameters

<state> Choose from the following:
INTernal - Use the internal 10 MHz Reference Oscillator
EXTernal - Use an external Reference Oscillator. Use **SENSe:ROSCillator:SOURce:CONDition?** to determine if the M937xA is locked to the external oscillator.

Examples **SENS:ROSC:SOUR INT**
sense:roscillator:source external

Query Syntax Not applicable

Return Type Not applicable

Default INTernal

SENSe:ROSCillator:SOURce:CONDition?

(Read-only) Reads the 10 MHz Reference Oscillator 'locked' condition.

When SENS:ROSC:SOUR is set to Internal, this command will always return "LOCKed".

When SENS:ROSC:SOUR is set to External, then this function takes about 100 usec to read the state of the hardware.

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**

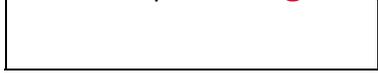
| **TIME**

|

CONTRol

| **X**

| **SPACing**



Click on a keyword to view the command details.

See Also

- Example: [Upload and Download a Segment List](#)
 - [Learn about Segment Sweep](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SENSe<cnum>:SEGMENT<snum>:ADD

(Write-only) Adds a segment.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <snum> 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<cnum>: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 Varies with 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 independently set for each segment.

OFF (or 0) - turns Bandwidth control OFF. Use the channel IF bandwidth setting **SENS:BWID**.

Examples

```
SENS:SEGM:BWID:CONT ON
```

```
sense2:segment:bwid: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:SEGM:COUNT?
sense2:segment:count?

Return Type Numeric

Default 1 segment

SENSe<cnum>:SEGMent<snum>:DELeTe

(Write-only) Deletes the specified sweep segment. When ALL segments are deleted, **SENS:SWE:TYPE** is automatically set to Linear because there are no segments to sweep.

<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:SEGM:DEL**
 sense2:segment2:delete

Query Syntax Not applicable

Default Not Applicable

SENSe<cnum>:SEGMent: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:SEGM:DEL:ALL**
sense2:segment:delete:all

Query Syntax Not applicable

Default Not Applicable

SENSe<cnum>:SEGMENT<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>:SEGMENT<snum>:FREQUENCY:CENTer?

Return Type Numeric

Default Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

SENSe<cnum>:SEGMENT<snum>:FREQUENCY:SPAN <num>

(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:SEGM:FREQ: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

```
SENS:SEGM:FREQ:STAR 1MHZ  
sense2:segment2:frequency:start minimum
```

Query Syntax SENSe<cnum>:SEGMENT<snum>:FREQUENCY:STARt?

Return Type Numeric

Default Stop Frequency of the previous segment. If first segment, start frequency of the analyzer.

SENSe<cnum>: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

- <cnum> 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

```
SENS:SEGM:FREQ:STOP 1MHZ  
sense2:segment2:frequency:stop maximum
```

Query Syntax SENSe<cnum>:SEGMENT<snum>:FREQUENCY:STOP?

Return Type Numeric

Default If first segment, stop frequency of the analyzer. Otherwise, start frequency of the segment.

SENSe<cnum>:SEGMENT:LIST <char>,<numSegs>,<data>

(Read-Write) Reads or writes the entire list of values in the segment sweep table.

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:BOReR** 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

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.

CouplePorts	SourcePowerOption	Number of Elements
False	False	Each port has its own channel-wide power setting, which is set using SOURce:POWer[:LEVel] . Provide exactly 7 elements per segment. The last element (power) is ignored.
False	True	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.
True	False	Provide exactly 7 elements per segment. The last element (power)

		is ignored.
True	True	Provide exactly 7 elements per segment. The last element (power) is honored.

Examples **SENS:SEGM: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 **FORMat[:DATA]**.

Default Not Applicable

SENSe<cnum>:SEGMent<snum>:POWer[<port>][:LEVel] <num>

(Read-Write) Sets the Port Power level for the specified sweep segment. First set SENS: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.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <snum> 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

```
SENS:SEGM:POW 0  
sense2:segment2:power1:level -10
```

Query Syntax SENSe<cnum>:SEGMent<snum>:POWer[<port>][:LEVel]?

Return Type Numeric

Default 0

SENSe<cnum>:SEGMENT:POWer[:LEVel]:CONTRol <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:SWEep: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

```
SENS:SEGM:SWE:TIME:CONT ON  
sense2:segment:sweep:time:control off
```

Query Syntax SENSe<cnum>:SEGMENT:SWEep:TIME:CONTRol?

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF

SENSe<cnum>:SEGMent: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

<char> **LINear** - turns X-axis point spacing OFF

OBASe - turns X-axis point spacing ON

Examples **SENS:SEGM:X:SPACing LIN**
sense2:segment:spacing obase

Query Syntax SENSe<cnum>:SEGMent:X:SPACing?

Return Type Character

Default LINear

Last Modified:

- 20-Aug-2013 Fixed error in finder map.
- 7-May-2012 Removed preset
- 30-Aug-2011 Removed Snum from X:Spacing
- 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:

BLOCKed

DWELI

| **AUTO**

| **SDELay**

GENeration

| **POINTsweep**

GRoups

| **COUNT**

MODE

POINTs

SPEed

SRCPort

STEP

TIME

| **AUTO**

TRIGger

| **DELAY**

| **MODE**

| **POINT**

TYPE

| **FACW**

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- **Example** [Triggering the PNA using SCPI](#)
- [Learn about Sweeping](#)

- Synchronizing the Analyzer and Controller
 - SCPI Command Tree
-

SENSe<cnum>:SWEep:BLOCked?

(Read-only) Reads whether the specified channel is currently 'blocked' from sweeping. Learn more about the Mechanical Devices dialog.

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1

Examples `SENS:SWE:BLOC?`
`sense2:sweep:blocked?`

Return Type Boolean
0 - No, the channel is NOT blocked.
1 - Yes, the channel is blocked.

Default N/A

SENSe<cnum>:SWEep:DWELl <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

<cnum> 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 **SENS:SWE:DWEL .1**
 sense2:sweep:dwell min

Query Syntax SENSE<cnum>:SWEep:DWELl?

Return Type Numeric

Default 0 - (**Note:** dwell time set to 0 is the same as dwell:auto ON)

SENSe<cnum>:SWEep:DWELl: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<cnum>:SWEep:DWELl:AUTO?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

SENSe<cnum>: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 the following two External Trigger delays if enabled.

- **Trig:Delay** (global scope)
- **Sens:Swe:Trig:Delay** (channel scope)

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:DWELL:SDEL .1`
`sense2:sweep:dwel:sdelay .5`

Query Syntax SENSe<cnum>:SWEep:DWELL:SDELay?

Return Type Numeric

Default 0

SENSe<cnum>: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

```
SENS:SWE:GEN STEP  
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

```
SENS:SWE:GEN:POIN 1
```

```
sense2:sweep:generation:pointsweep off
```

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

```
SENS:SWE:GRO:COUN 10  
sense2:sweep:groups:count 50
```

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:

HOLD - channel will not trigger

CONTInuous - channel triggers indefinitely

GROups - channel accepts the number of triggers specified with the last **SENS:WE:GRO:COUN** <num>. This is one of the PNA overlapped commands. [Learn more](#).

SINGLE - channel accepts ONE trigger, then goes to HOLD.

Note: To perform simple, single-triggering, use SINGLE which requires that **TRIG:SOURce** remain in the default (internal) setting.

Examples **SENS:SWE:MODE CONT**
sense2:sweep:mode hold

Query Syntax SENSe<cnum>:SWEep:MODE?

Return Type Character

Default CONTInuous

SENSe<cnum>:SWEep:POINts <num>

(Read-Write) Sets the number of data points for the measurement.

Parameters

- <cnum> 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 `SENS:SWE:POIN 51`
`sense2:sweep:points max`

Query Syntax `SENSe<cnum>:SWEep:POINts?`

Return Type Numeric

Default 201

SENSe<cnum>: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

- <cnum> 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 `SENS:SWE:SRCP 1`
`sense2:sweep:srcport 2`

Query Syntax `SENSe<cnum>:SWEep:SRCPort?`

Return Type Character

Default 1

SENSe<cnum>:SWEep:SPEed <char>

(Read-Write) Sets and returns the state of Fast Sweep mode. [Learn more about Fast Sweep.](#)

Parameters

<cnum> Any existing channel number. If unspecified, value is set to 1

<char> Fast Sweep mode. Choose from:

FAST - turns Fast Sweep Mode ON

NORMal - turns Fast Sweep Mode OFF (Normal Mode).

Examples

```
SENS:SWE:SPE NORM
```

```
sense2:sweep:speed fast
```

Query Syntax SENSe<cnum>:SWEep:SPEed?

Return Type Character

Default NORMal

SENSe<cnum>:SWEep:STEP <num>

(Read-Write) Sets the frequency step size across the selected frequency range. This effectively sets the number of data points. Available ONLY when **Sweep Type** = Linear.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <num> Frequency step size in Hz. Select any value up to the frequency range of the analyzer.

Examples

```
SENS:SWE:STEP 1e6  
sense2:sweep:step 1000000
```

Query Syntax SENSe<cnum>:SWEep:STEP?

Return Type Numeric

Default NA

SENSe<cnum>:SWEep:TIME <num>

(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

<cnum> Any existing channel number. If unspecified, value is set to 1

<num> Sweep time in seconds.

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.

The MAX value will change based on point count, IFBW, and dwell time.

Examples `SENS:SWE:TIME 1ms`
`sense2:sweep:time .001`

Query Syntax SENSe<cnum>:SWEep:TIME?

Return Type Numeric

Default NA

SENSe<cnum>:SWEep:TIME:AUTO <ON | OFF>

(Read-Write) Turns the automatic sweep time function ON or OFF.

Parameters

- <cnum> 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

```
SENS:SWE:TIME:AUTO  
sense2:sweep:time:auto off
```

Query Syntax SENSe<cnum>:SWEep:TIME:AUTO?

Return Type Boolean (1 = ON, 0 = OFF)

Default ON

SENSe<cnum>: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

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <num> Trigger delay value in seconds. Range is from 0 to 3 seconds

Examples `SENS:SWE:TRIG:DELay .003`
`sense2:sweep:trigger:delay 1`

Query Syntax `SENSe<cnum>:SWEep:TRIGger:DELay?`

Return Type Numeric

Default 0

SENSe<cnum>: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 SWE  
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

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

X Values Command

SENSe<cnum>:X[:VALues]? - **Superseded**

Replaced with **CALC:X?**

(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: To avoid frequency rounding errors, specify **FORM:DATA** <Real,64> or <ASCIi,0>

Parameters

<cnum> Any existing channel number; if unspecified, value is set to 1.

Examples **SENS:X?**
sense2:x:values?

Return Type Depends on **FORM:DATA** command

Default Not applicable

Last Modified:

16-Aug-2013	Superseded per JE
10-Sep-2012	Added Form:Data note
9-Apr-2010	Added Note

Source Commands

Controls the power delivered to the DUT.

SOURce:
CATalog?
POWER
 | **ATTenuation**
 | **AUTO**
 | **CENTER**
 | **COUPLE**
 | **DETECTOR**
 | **[LEVEL]**
 | **[IMMEDIATE]**
[AMPLITUDE]
 | **SLOPE**
 | **STATE**
 | **MODE**
 | **PORT**
 | **START**
 | **STOP**
 | **SPAN**
 | **START**
 | **STOP**

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
 - [Learn about Power Settings](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
 - [Remotely Specifying a Source Port](#)
-

SOURce<cnum>:CATalog?

(Read-only) 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?  
source: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>: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:POWer: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.

[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: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>.
- [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:POW2:ATT:Auto On  
source2:power:attenuation: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<port>: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.
- <port> If provided, this argument is **ignored** by the PNA.

Examples SOUR:POW:CENT -15
source2:power:center -7

Query Syntax SOURce<cnum>:POWer:CENTer?

Return Type Numeric

Default 0 dBm

SOURce<cnum>:POWer<port>: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.
- <port> If provided, this argument is **ignored** by the PNA.

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**

(Read-Write) 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.

- [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: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<port>[:LEVel]:SLOPe <num>

(Read-Write) Sets the RF power slope value.

Also enable the slope state using **SOUR:POW:SLOP:STAT ON**.

Parameters

- <cnum> Any existing channel number. If unspecified, value is set to 1
- <num> Slope value in db/GHz. Choose any value between -2 and 2 (0 is no slope).
- <port> If provided, this argument is **ignored** by the PNA.

Examples **SOUR:POW:SLOP .5234434**
source2:power:level slope -1.345

Query Syntax SOURce<cnum>:POWer[:LEVel]:SLOPe?

Return Type Numeric

Default 0

SOURce<cnum>:POWer<port>[:LEVel]:SLOPe:STATe <ON | OFF>

(Read-Write) Turns Power Slope ON or OFF. Set the slope using **SOUR:POW:SLOP**.

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.

<port> If provided, this argument is **ignored** by the PNA.

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.

[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: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.

- [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: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.

- [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: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<port>: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.
- <port> If provided, this argument is **ignored** by the PNA.

Examples **SOUR:POW:SPAN -15**
source2:power:span -7

Query Syntax SOURce<cnum>:POWer:SPAN?

Return Type Numeric

Default 0 dBm

SOURce<cnum>:POWer<port>: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

Actual achievable leveled power depends on frequency.

<port> If provided, this argument is **ignored** by the PNA.

Examples SOUR:POW:STAR -15
source2:power:start -7

Query Syntax SOURce<cnum>:POWer:STARt?

Return Type Numeric

Default 0 dBm

SOURce<cnum>:POWer<port>: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 **SOUR:POW: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. (**SOUR:POW:ATT:AUTO** must be set to ON) Example:
SOURce:POWer:STOP? MAX

Actual achievable leveled power depends on frequency.

<port> If provided, this argument is **ignored** by the PNA.

Examples **SOUR:POW:STOP -15**
source2:power:stop -7

Query Syntax SOURce<cnum>:POWer:STOP?

Return Type Numeric

Default 0 dBm

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

DATA

[STATe]

Click on a keyword to view the command details.

- [SCPI Command Tree](#)
-
-

SOURce<ch>: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

<ch> Channel number of the source power cal. 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

[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 **SOURce1:POWer2:CORRection:DATA 0.12, -0.34, 0.56**

Query Syntax SOURce<ch>:POWer<port>:CORRection:DATA? [src]

Return Type Depends on **FORMat:DATA** command

Default Not Applicable



SOURce<ch>:POWer<port>:CORRection[:STATe] <bool>[,src]

(Read-Write) Enables and disables source power correction for the specified port on the specified channel.

Parameters

- <ch> Channel number of the source power cal. 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.
- [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 ON source1:power2:correction:state off, "MXG N5183A"**

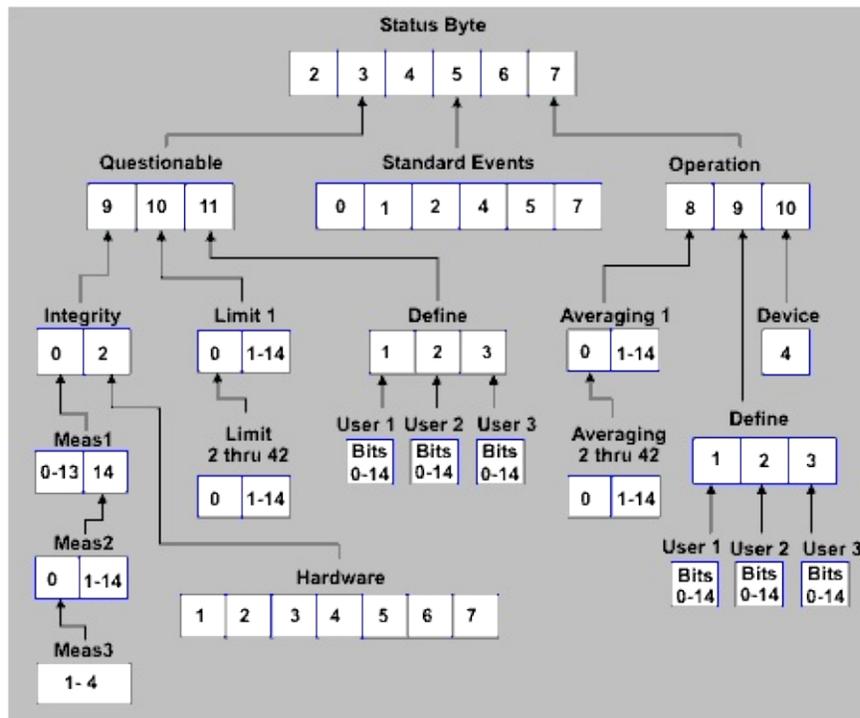
Query Syntax SOURce:POWer:CORRection[:STATe]? "MXG N5183A"

Return Type Boolean (1 = ON, 0 = OFF)

Default OFF (0)

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 **red** keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Status Registers](#)
- [Synchronizing the Analyzer 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.

Commands Description

- *CLS Clears ALL "event" registers and the SCPI Error / Event queue. The corresponding ENABLE registers are unaffected.
- *STB? Reads the value of the analyzer's status byte. The byte remains after being read.
- *SRE? Reads the current state of the Service Request **Enable** Register.
- *SRE <num> 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.
 <num> Combined value of the weights for bits to be set.

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
2	4	Error / Event queue Summary (EAV)	the Error / Event queue is not empty. To read the error message, use SYST:ERR?
3	8	Questionable Register Summary	any enabled bit in the questionable event status register is set to 1
4	16	Message Available	the output queue is not empty
5	32	Standard Event Register Summary	any enabled bit in the standard event status register is set to 1
6	64	Request Service	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.

7	128	Operation Register Summary	any enabled bit in the standard operation event status register is set to 1
---	-----	----------------------------	--

STATUS:QUESTIONABLE:<keyword>

Summarizes conditions that monitor the quality of measurement data.

<keyword> **Example**

:CONDition? **STAT:QUES:COND?**

:ENABle **STAT:QUES:ENAB 1024**
<bits>

[:EVENT] ? **STAT:QUES?**

:NTRansition **STAT:QUES:NTR 1024**
<bits>

:PTRansition **STAT:QUES:PTR 0**
<bits>

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
9	512	Integrity Reg summary	any enabled bit in the Integrity event register is set to 1
10	1024	Limit Registers summary	any enabled bit in the Limit event registers is set to 1
11	2048	Define Registers summary	any enabled bit in the Define event registers is set to 1

STATus:QUEStionable:INTEgrity <keyword>

Summarizes conditions in the Measurement Integrity register.

<keyword> **Example**

:CONDition? **STAT:QUES:INT:COND?**

:ENABle <bits> **STAT:QUES:INT:ENAB 1024**

[:EVENT]? **STAT:QUES:INT?**

:NTRansition <bits> **STAT:QUES:INT:NTR 1024**

:PTRansition <bits> **STAT:QUES:INT:PTR 0**

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
0	1	Measurement Summary	any bit in the Measurement Integrity event register is set to 1
2	4	Hardware Summary	any bit in the Hardware event register is set to 1

STATUS:QUESTIONABLE:INTEGRITY:HARDWARE<keyword>

Monitors the status of hardware failures.

<keyword> **Example**

:CONDITION? **STAT:QUES:INT:HARD:COND?**

:ENABLE <bits> **STAT:QUES:INT:HARD:ENAB 1024**

[:EVENT]? **STAT:QUES:INT:HARD?**

:NTRANSITION <bits> **STAT:QUES:INT:HARD:NTR 1024**

:PTRANSITION <bits> **STAT:QUES:INT:HARD:PTR 0**

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
1	2	Phase Unlock	the source has lost phaselock, possibly caused by a reference channel open or a hardware failure.
2	4	Unleveled	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.
3	8	Not used	N/A
4	16	EE Write Failed	an attempted write to the EEPROM has failed, possibly caused by a hardware failure.
5	32	Not used	N/A
6	64	Ramp Cal Failed	the analyzer was unable to calibrate the analog ramp generator due to a possible hardware failure.
7	128	Not used	N/A

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> **Example**

:CONDition? **STAT:QUES:INT:MEAS1:COND?**

:ENABle <bits> **STAT:QUES:INT:MEAS2:ENAB 1024**

[:EVENT]? **STAT:QUES:INT:MEAS3?**

:NTRansition <bits> **STAT:QUES:INT:MEAS2:NTR 1024**

:PTRansition <bits> **STAT:QUES:INT:MEAS1:PTR 0**

		Measurement Register <n>			
Bit	Weight	1	2	3	Bit is set to 1 when the following conditions exist:
0	1	1	Summary from Meas Reg 3		a setting change on this channel has occurred and the data does not yet reflect that change.
1	2	2	15	29	a setting change on this channel has occurred and the data does not yet reflect that change.
2	4	3	16	30	a setting change on this channel has occurred and the data does not yet reflect

					that change.
3	8	4	17	31	a setting change on this channel has occurred and the data does not yet reflect that change.
4	16	5	18	32	a setting change on this channel has occurred and the data does not yet reflect that change.
5	32	6	19		a setting change on this channel has occurred and the data does not yet reflect that change.
6	64	7	20		a setting change on this channel has occurred and the data does not yet reflect that change.
7	128	8	21		a setting change on this channel has occurred and the data does not yet reflect that change.
8	256	9	22		a setting change on this channel has occurred and the data does not yet reflect that change.
9	512	10	23		a setting change on this channel has occurred and the data does not yet reflect that change.
10	1024	11	24		a setting change on this channel has occurred and the data does not yet reflect that change.
11	2048	12	25		a setting change on this channel has occurred and the data does not yet reflect that change.
12	4096	13	26		a setting change on this channel has occurred and the data does not yet reflect that change.

13	8192	14	27	a setting change on this channel has occurred and the data does not yet reflect that change.
14	16384	Summary from Meas Reg 2	28	a setting change on this channel has occurred and the data does not yet reflect that change.

STATus:QUEStionable:LIMit<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:LIM4:COND?**

:ENABle <bits> **STAT:QUES:LIM1:ENAB 1024**

[:EVENTt]? **STAT:QUES:LIM3?**

:NTRansition <bits> **STAT:QUES:LIM2:NTR 1024**

:NTRansition? **STAT:QUES:LIM1:NTR?**

:PTRansition <bits> **STAT:QUES:LIM5:PTR 0**

:PTRansition? **STAT:QUES:LIM1:PTR?**

		Limit Register <n>											
Bit	Weight	1	2	3	4	5	6	7	8	...	41	42	Bit is set to 1 when the following conditions exist:
0	1	2-42	3-42	4-42	5-42	6-42	7-42	8-42	9-42	...	42	--	Summary Bit - If any bit from that register fails, it propagates to the previous register, bit 0.
		Trace Numbers											
1	2	1	15	29	43	57	71	85	99	...	561	575	any point on trace

													fails the limit test
2	4	2	16	30	44	58	72	86	100	...	562	576	any point on trace fails the limit test
3	8	3	17	31	45	59	73	87	101	...	563	577	any point on trace fails the limit test
4	16	4	18	32	46	60	74	88	102	...	564	578	any point on trace fails the limit test
5	32	5	19	33	47	61	75	89	103	...	565	579	any point on trace fails the limit test
6	64	6	20	34	48	62	76	90	104	...	566	580	any point on trace fails the limit test
7	128	7	21	35	49	63	77	91	105	...	567	--	any point on trace fails the limit test
8	256	8	22	36	50	64	78	92	106	...	568	--	any point on trace fails the limit test
9	512	9	23	37	51	65	79	93	107	...	569	--	any point on trace fails the limit test
10	1024	10	24	38	52	66	80	94	108	...	570	--	any point on trace fails the limit test
11	2048	11	25	39	53	67	81	95	109	...	571	--	any point on trace fails the limit test
12	4096	12	26	40	54	68	82	96	110	...	572	--	any point on trace fails the limit test
13	8192	13	27	41	55	69	83	97	111	...	573	--	any point on trace fails the limit test
14	16384	14	28	42	56	70	84	98	112	...	574	--	any point on trace fails the limit test

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.
 - $((400-1)/14) + 1 = \text{Register\#} + 1 \text{Bit}$
 - $399/14 = 28 \text{ r}7$
 - $28+1 = \text{Register } 29$
 - $7+1 = \text{Bit number } 8$
 - 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.

<keyword> **Example**

:CONDition? **STAT:QUES:DEF:COND?**

:ENABle <bits> **STAT:QUES:DEF:ENAB 1024**

[:EVENTt]? **STAT:QUES:DEF?**

:NTRansition <bits> **STAT:QUES:DEF:NTR 1024**

:PTRansition <bits> **STAT:QUES:DEF:PTR 0**

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.

<keyword> **Example**

:ENABle <bits> **STAT:QUES:DEF:USER1:ENABle 1024**

[:EVENT]? **STAT:QUES:DEF:USER1?**

:MAP <bit>,<error> **STAT:QUES:DEF:USER1:MAP 0,-113 '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

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).

Description

Commands

- *ESE? Reads the settings of the standard event **ENABLE** register.
- *ESE
<*bits*> Sets bits in the standard event **ENABLE** register. The current setting is saved in non-volatile memory.
<*bits*> The sum of weighted bits in the register. Use *ESE 0 to clear the enable register.
- *ESR? Reads and clears the **EVENT** settings in the Standard Event Status register.
- *OPC Sets bit 0 when the overlapped command is complete. (see Understanding Command Synchronization / OPC).
- *OPC? Operation complete query - read the Operation Complete bit (0).

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
0	1	Operation Complete	the two following events occur in order : <ol style="list-style-type: none"> 1. the *OPC command is sent to the analyzer 2. the analyzer completes all pending overlapped commands
1	NA	Request Control	Not Supported - the analyzer application is not configured to control GPIB operation

2	4	Query Error	a query error is detected indicating: - an attempt to read data from the output queue when no data was present OR - data in the output queue was lost, as in an overflow
4	16	Execution Error	an execution error is detected indicating: - a <PROGRAM DATA> element was outside the legal range or inconsistent with the operation of the analyzer OR - the analyzer could not execute a valid command due to some internal condition
5	32	Command Error	a command error is detected indicating that the analyzer received a command that: <ul style="list-style-type: none"> • did not follow proper syntax • was misspelled • was an optional command it does not implement
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.

<keyword> **Example**

:CONDition? **STAT:OPER:COND?**

:ENABle <bits> **STAT:OPER:ENAB 1024**

[:EVENT]? **STAT:OPER?**

:NTRansition <bits> **STAT:OPER:NTR 1024**

:PTRansition <bits> **STAT:OPER:PTR 0**

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
8	256	Averaging summary	either enabled bit in the Averaging summary event register is set to 1
9	512	User Defined summary	
10	1024	Device summary	either enabled bit in the Device summary event register is set to 1

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

<keyword> **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**

		Averaging Register <n>											
Bit	Weight	1	2	3	4	5	6	7	8	...	41	42	Bit is set to 1 when the following conditions exist:
0	1	2-42	3-42	4-42	5-42	6-42	7-42	8-42	9-42	...	42	--	Summary Bit - If any bit from that register fails, it propagates to the previous register, bit 0.
		Trace Numbers											
1	2	1	15	29	43	57	71	85	99	...	561	575	Averaging on this trace is complete
2	4	2	16	30	44	58	72	86	100	...	562	576	Averaging on this trace is complete

3	8	3	17	31	45	59	73	87	101	...	563	577	Averaging on this trace is complete
4	16	4	18	32	46	60	74	88	102	...	564	578	Averaging on this trace is complete
5	32	5	19	33	47	61	75	89	103	...	565	579	Averaging on this trace is complete
6	64	6	20	34	48	62	76	90	104	...	566	580	Averaging on this trace is complete
7	128	7	21	35	49	63	77	91	105	...	567	--	Averaging on this trace is complete
8	256	8	22	36	50	64	78	92	106	...	568	--	Averaging on this trace is complete
9	512	9	23	37	51	65	79	93	107	...	569	--	Averaging on this trace is complete
10	1024	10	24	38	52	66	80	94	108	...	570	--	Averaging on this trace is complete
11	2048	11	25	39	53	67	81	95	109	...	571	--	Averaging on this trace is complete
12	4096	12	26	40	54	68	82	96	110	...	572	--	Averaging on this trace is complete
13	8192	13	27	41	55	69	83	97	111	...	573	--	Averaging on this trace is complete
14	16384	14	28	42	56	70	84	98	112	...	574	--	Averaging on this trace is complete

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.
 - $((400-1)/14) + 1 = \text{Register\# } r+1\text{Bit}$
 - $399/14 = 28 \text{ r}7$
 - $28+1 = \text{Register } 29$
 - $7+1 = \text{Bit number } 8$
 - 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.

<keyword> **Example**

:CONDition? **STAT:OPER:DEF:COND?**

:ENABle <bits> **STAT:OPER:DEF:ENAB 12**

[:EVENT]? **STAT:OPER:DEF?**

:NTRansition <bits> **STAT:OPER:DEF:NTR 12**

:PTRansition <bits> **STAT:OPER:DEF:PTR 0**

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:OPERation:DEFine:USER<1|2|3><keyword>

Monitors conditions that you define and map in any of the three OPER:DEF:USER event registers.

<keyword> Example

:ENABle <bits> **STAT:OPER:DEF:USER1:ENAB 1024**

[:EVENT]? **STAT:OPER:DEF:USER1?**

:MAP <bit>,<error> **STAT:OPER:DEF:USER1:MAP 0,-113 '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> **Example**

:CONDition? **STAT:OPER:DEV:COND?**

:ENABle <bits> **STAT:OPER:DEV:ENAB 16**

[:EVENTt]? **STAT:OPER:DEV?**

:NTRansition <bits> **STAT:OPER:DEV:NTR 16**

:PTRansition <bits> **STAT:OPER:DEV:PTR 0**

Bit	Weight	Description	Bit is set to 1 when the following conditions exist:
0	1	Unused	
1	2	Unused	
2	4	Unused	
3	8	Unused	
4	16	Sweep Completed	When sweep is complete
5	32	Unused	
6	64	Unused	
7	128	Unused	
8	256	Unused	
9	512	Unused	
10	1024	Unused	
11	2048	Unused	

12	4096	Unused	
13	8192	Unused	
14	16384	Unused	

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 VNA system.

SYSTem:

ABORt:THReshold

ACTive

| **CHANnel**

| **MEASurement**

BEEPer:VOLume

CAL:ALL **More commands**

CAPability **More commands**

CHANnels

| **CATalog?**

| **COUPle[:STATe]**

| **DELete**

| **HOLD**

| **RESume**

| **SINGle:COMBine**

CLOCK[:STATe]

COMMunicate **More commands**

CONFigure

| **DIRectory**

| **REVision**

| **CPU?**

| **DSP?**

| **DSPFpga?**

CORRection

| **WIZard[:IMMEDIATE]**

DATE?

ERRor?

| **COUNt?**

| **REPort**

| **SUNLeveled**

FIFO **More commands**

FPReset

HISTory

| **FPANel?**

| **SCPI?**

MACRO:COPY

CHANnel[:TO]
SOURce
MCLass
CATalog?
PARAmeter:CATalog?
MEASurement
CATalog?
NAME?
TRACe?
WINDow?
POWer:
LIMit
LOCK
STATe
PREFerences More commands
PRESet
SECurity
[LEVel]
SET
SHORtcut
ARGuments
DELete
EXECute
PATH
TITLe
TIME?
TOUCHscreen[:STATe]
WINDows
CATalog?

Click on a **red** keyword to view the command details.

see Also

- [Referring to Traces Channels Windows and Meas Using SCPI](#)
- [Learn about PNA Preferences](#)
- [Example Programs](#)
- [Synchronizing the Analyzer and Controller](#)

- SCPI Command Tree



SYSTem:ABORt:THReshold <value>

(Read-Write) When a PNA setting is made while a sweep is in progress, the sweep is immediately aborted by default. This command allows you to change that behavior by specifying a time threshold. When a setting change is made during a sweep and if the total sweep time is less than the threshold time, then the sweep is allowed to finish instead of immediately aborting.

In general, PNA setting changes that could cause an aborted sweep are changes that affect how a measurement is made, such as changes in stimulus conditions.

For example, with a threshold setting of 60 seconds:

- Sweeps that require 60 seconds or less from start to finish will be allowed to complete if a PNA setting change is made at any time during the sweep.
- Sweeps that require MORE than 60 seconds from start to finish will be immediately aborted when a PNA setting change is made at any time during the sweep.

Notes:

- Preset clears this setting.
- Save state saves this setting.
- Sweep times are estimated.
- This setting affects ALL channels.

Parameters

<value> Threshold time in seconds. Set to 0 to immediately abort a sweep when a PNA setting is made.

Examples

SYST:ABOR:THR 10

'When a setting is made during a sweep, if that sweep requires less than 10 seconds more to complete, it will be allowed to finish instead of aborting.'

Query Syntax SYSTem:ABORt:THReshold?

Default 0 - No threshold time; all sweeps are immediately aborted.

SYSTem:ACTive:CHANnel?

(Read-only) Returns the number of the active channel. The active channel is the channel number that contains the active measurement. The active measurement is the trace that has a highlighted **Tr#** in the **Trace Status** area.

If there is no active channel, 0 is returned.

Examples

```
SYST:PRES
SYST:ACT:CHAN?
'Returns 1
```

Return Type Integer

Default Not Applicable

SYSTem:ACTive:MEASurement?

(Read-only) Returns the name of the active measurement. While looking at the PNA display, the active measurement is the trace that has a highlighted **Tr#** in the **Trace Status** area. Only displayed measurements can be active.

If there is no active measurement, " " (empty string) is returned.

Examples **SYST:PRES**
 SYST:ACT:MEAS?
 'Returns "CH1_S11_1"'

Return Type String

Default Not Applicable

SYSTem:BEEPer:VOLume <num>

(Read-Write) Sets and reads the volume of the internal speaker.

Parameters

<num> Relative volume of the internal speaker.
Choose a volume between 0 (off) and 100.

Examples

```
SYST:BEEP:VOL 5  
system:beeper:volume
```

Query Syntax SYSTem:BEEPer:VOLume?

Default 0

SYSTem:CHANnels:CATalog?

(Read-only) Returns the channel numbers currently in use.

Examples SYST:CHAN:CAT?
system:channels:catalog?
Returns:
"1,2,3"

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 Keysight 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).

Examples SYST:CHAN:RES

Query Syntax Not Applicable

Default Not Applicable

SYSTem:CHANnels:SINGle:COMBine <chanNums>

(Write-only) Sets the trigger count on the list of channels to ONE, and then combines the channels into a single efficient acquisition. The index line stays high during the entire acquisition.

Parameters

<chanNums> Existing comma separated list of channel numbers to combine.

Examples

```
SYST:CHAN:SING:COMB 1,3,4  
System:channels:single:combine 1,3,4
```

Query Syntax Not Applicable

Default Not Applicable

SYSTem:CLOCK[:STATe] <bool>

(Read-Write) Sets and reads the clock visibility state in the VNA status bar.

Parameters

<bool> **ON** (or 1) Clock is visible in the PNA status bar.
OFF (or 0) Clock is NOT visible in the PNA status bar.

Examples **SYST:CLOC 1**
system:clock:state OFF

Query Syntax SYSTem:CLOCK[:STATe]?

Default ON

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:DIRectory? <char>

(Read-only) Returns the directory path location for the specified file type.

Parameters None

<char> Type of file. Choose from:

STATe - This is the location for the storage of state files.

APPLication - This is the location of the PNA firmware executable files.

SUPPort - This is the location of private support files for the PNA firmware.

See these file locations.

Example SYST:CONF:DIR? SUPP

Return Type String

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.

Reported CPU version - Clock speed

1.0 - 266 MHz

2.0 - 500 MHz

3.0 - 1100 MHz

4.0 - 1600 MHz

5.0 - 2000 MHz

6.0 - 2000 MHz dual core.

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:DATE?

(Read-only) Returns the system date.

Parameters None

Example SYST:DATE?

Return Type Comma separated numbers representing year, month, day.

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	SYST:ERR? system:error?
-----------------	--

Default Not Applicable

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 SYST:ERR:COUN?
system:error:count?

Default Not Applicable

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

- <bool> **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:FPRreset

(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:HISTory:FPANel?

(Read-only) Returns front panel history.

Parameters None

Examples SYST:HIST:FPAN?

Return Type String of comma-separated history of front panel actions.

Default Not Applicable

SYSTem:HISTory:SCPI?

(Read-only) Returns SCPI history.

Parameters None

Examples SYST:HIST:SCPI?

Return Type String of comma-separated history of SCPI commands sent to the PNA.

Default Not Applicable

SYSTem:MACRo:COPIY: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:COPIY:CHAN1 2  
system:macro:copy:channel2:to 3
```

Query Syntax Not Applicable

Default Not Applicable

SYSTem:MACRo:COPIY: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:COPIY: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:MCLass:PARAmeter:CATalog? <name>

(Read-only) Returns ALL parameters that are supported by the specified measurement class

Parameters

<name> String. Measurement Class name. See the complete list of measurement classes

Examples

Returns all parameters for Gain Compression.

```
SYST:MCL:PAR:CAT? "Gain Compression"
```

Return:

```
"S11,S12,S13,S14,S21,S22,S23,S24,S31,S32,S33,S34,S41,S42,S43,S44,A"
```

Return String of comma-separated parameters

Type

Default Not Applicable

SYSTem:MEASurement:CATalog? [chan]

(Read-only) Returns ALL measurement numbers, or measurement numbers from a specified channel.

Parameters

[chan] Optional. Channel number to catalog. If not specified, all measurement numbers are returned.

Examples

'Returns all measurement numbers

SYST:MEAS:CAT?

'Returns the measurement numbers 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

Returns 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 of the specified measurement number.

Parameters

<n> Measurement number for which to return the window number. If unspecified, value is set to 1.

Examples

'Returns 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** - Disables the ability to change the power limit values from the user interface.
OFF or **0** - Enables 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:PRESet 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**.

Examples	SYST:PRES system:preset
-----------------	--

Default	Not applicable
----------------	----------------

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.

Examples `SYST:SEC LOW`
`system:security:level high`

Query Syntax SYSTem:SECurity[:LEVel]?

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 of the macros for the desired macro title.

<string> Arguments for the specified macro.

Examples

```
SYST:SHOR1:ARG  
"http://na.support.keysight.com/pna/help/PNAWebHelp/help.htm"
```

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 of the 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 of the 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 of the macros for the desired macro title.

Examples **SYST:SHOR1:PATH "C:/Program Files/Keysight/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 of the 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:TIME?

(Read-only) Returns the system time.

Parameters None

Example SYST:TIME?

Return Type Comma separated numbers representing hours, minutes, seconds.

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:WINDows:CATalog?

(Read-only) Returns the window numbers 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:

- 17-Mar-2014 Added note to Abort thres
- 7-Feb-2014 Changed Active commands to return 0 or "". (A.09.90.10)
Moved communicate commands.
- 15-Nov-2013 Fixed couple in tree
- 29-Apr-2013 Removed reference to 12 macros
- 6-Aug-2012 Fixed Meas:CAT and windows cat?
- 23-Mar-2012 Added Cal All
- 24-Jan-2012 Updated PSEnSor command with 'any'.
- 5-Jan-2012 Added 6.0 CPU to list
- 26-Oct-2011 Added Abort command
- 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

28-Aug-2008 Updated Launch Cal Wiz command

11-Feb-2008 Added Noise switch preference (8.2)

5-Feb-2007 Added Extra security and USB power meter commands

23-Feb-2007 Added touchscreen command

15-Nov-2006 Added Unleveled Error reporting

31-Oct-2006 Added PSRTrace command

SYSTem:CALibrate:ALL Commands

Contains the settings to configure a "Cal All" Calibration.

Use the [Guided Cal](#) interface to perform the calibration.

SYSTem:CALibrate:ALL:

CHANnel:

| [PORTs\[:SElect\]](#)

CSET:

| [CATalog?](#)

| [PREFix](#)

GUIDed:

| [CHANnel?](#)

| [PORTs?](#)

IFBW

MCLass:

| [PROPerty:](#)

| [NAME:CATalog?](#)

| [VALue:](#)

| [CATalog?](#)

| [\[STATe\]](#)

PORT<n>:

| [RECeiver:ATTen](#)

| [SOURce:POWer:](#)

| [ATTen](#)

| [OFFSet](#)

| [\[VALue\]](#)

RESet

SElect

Click on a [red](#) keyword to view the command details.

see Also

- [About Calibrate All Channels](#)
 - [Example Program](#)
 - [Guided Cal commands](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:CALibrate:ALL:CHANnel<ch>:PORTs[:SElect] <value>

(Write-Read) For each channel to be calibrated, sets and returns the ports to be calibrated. Specify port numbers **ONLY** for standard channels. Application channels are not necessary because they have designated input/output/LO ports.

Parameters

<ch> Channel number to be calibrated.

<value> Ports to be calibrated for the specified channel. Select any of the native PNA ports (1,2,3,4).

Examples **SYST:CAL:ALL:CHAN2:PORT 1,2,3**

Query Syntax SYSTem:CALibrate:ALL:CHANnel<ch>:PORTs[:SElect]?

Return Type Comma-separated port numbers.

Default 1,2

SYSTem:CALibrate:ALL:CSET:CATalog?

(Read-only) Returns the User Cal Set or cal register names that were produced by the cal all session.

Parameters None

Examples SYST:CAL:ALL:CSET:CATalog?

'returns this format:

"MyCalAll_STD_001, MyCalAll_SMC_002"

See example program

Return Type String of comma-separated Cal Set or cal register names

Default Not Applicable

SYSTem:CALibrate:ALL:CSET:PREFix<value>

(Write-Read) Sets and returns the prefix to be used when saving User Cal Sets that result from the Cal All session. The Meas Class and channel number are appended to this prefix for each calibrated channel. Use **SYST:CAL:ALL:CSET:CATalog?** to read the saved cal set names.

- **SENS:CORR:COLL:GUID:SAVE:CSET** can also be used to set the Cal Set prefix.
- If a Cal Set prefix is NOT set using either command, the cal data for each channel will be saved only to cal registers. [Learn about cal registers.](#)

Parameters

<value> (String) User Cal Set prefix.

Examples `SYST:CAL:ALL:CSET:PREFix "MyCalAll"`

Query Syntax `SYSTem:CALibrate:ALL:CSET:PREFix?`

Return Type String

Default " " (Empty string)

SYSTem:CALibrate:ALL:GUIDed:CHANnel?

(Read-only) Reads the channel number of the Cal All Calibration. Use this value as the <ch> argument for the subsequent **Guided:Cal** commands.

Parameters None

Examples `chan = SYST:CAL:ALL:GUID:CHAN?`

Return Type Numeric

Default Not applicable

SYSTem:CALibrate:ALL:GUIDed:PORTs?

(Read-only) Returns the ports to be calibrated during the Cal All Channels calibration. Specify connectors and cal kits for these ports using the **Guided:Cal** commands.

Specify the ports to be calibrated for each channel using **SYST:CAL:ALL:CHAN<ch>:PORT**.

Parameters None

Examples `ports = SYST:CAL:ALL:GUID:PORT?`

Return Type Comma-separated list of port numbers

Default Not applicable

SYSTem:CALibrate:ALL:IFBW <value>

(Write-Read) Sets and returns the IFBW for a Cal All calibration. [Learn more about this setting.](#)

Parameters

<value> IF Bandwidth in Hz. The list of valid IF Bandwidths is different depending on the PNA model. [See the list of valid settings.](#) If an invalid number is specified, the PNA will round up to the closest valid setting.

This command supports MIN and MAX as arguments. [Learn more.](#)

Examples `SYST:CAL:ALL:IFBW 10e3`

Query Syntax SYSTem:CALibrate:ALL:IFBW?

Return Type Numeric

Default 1 kHz

SYSTem:CAL:ALL:MCLass:PROPerTy:NAME:CATalog? **[mclass]**

(Read-only) Returns the unique, settable properties for the current cal all session.
See a list of valid properties and values for each measurement class.

Parameters

[mclass] Optional argument. String name of the measurement class for which properties are to be returned. See a list of valid measurement class Application names. The measurement class must be included in the current Cal All calibration.

Examples

```
SYST:CAL:ALL:MCL:PROP:NAME:CAT?
```

'with NFX app, returns:

```
"Noise Cal Method,Noise Tuner,AutoOrient Tuner,Tuner  
In,Tuner Out,Receiver Characterization Method,ENR  
File,Noise Source Connector,Noise Source CalKit"
```

Return Type String of comma-separated properties.

Default Not applicable

SYSTem:CAL:ALL:MCLass:PROPerTy:VALue:CATalog? <prop>

(Read-only) Returns the valid property values for a specific property name.
See a list of valid properties and values for each measurement class.

Parameters

<prop> (String) Property name for which valid values are to be returned.

Examples SYST:CAL:ALL:MCL:PROP:VAL:CAT? "Noise Cal Method"
'with NFX app, returns:
"Scalar,Vector"

Return Type String of comma-separated values

Default Not applicable

SYSTem:CALibrate:ALL:MCLass:PROPerTy:VALue[:STATe] <prop>,<value>

(Write-Read) Sets and returns the property value for a specific property name.
See a list of valid properties and values for each measurement class.

Parameters

<prop> (String) Property name for which value is to be set or returned.

<value> Property value. To read a list of valid values, use
SYST:CAL:ALL:MCL:PROP:VAL:CAT?

Examples

```
SYST:CAL:ALL:MCL:PROP:VAL "Noise Cal  
Method","Noise:Scalar"
```

```
SYST:CAL:ALL:MCL:PROP:VAL? "Noise Source Connector"
```

Query Syntax SYSTem:CALibrate:ALL:MCLass:PROPerTy:VALue[:STATe]?
<prop>

Return Type String

Default Varies with the property name.

SYSTem:CALibrate:ALL:PORT<n>:RECeiver:ATTen<value> [,src]

(Write-Read) Sets and returns the Receiver Attenuator setting for a Cal All calibration.

Parameters

<n> Receiver port number.

<value> Attenuation value in dB for a Cal All calibration. Choose a valid value for the PNA model. **See valid settings.**

[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 **SYST:CAL:ALL:PORT2:REC:ATT 10**

Query Syntax SYSTem:CALibrate:ALL:PORT<n>:RECeiver:ATTen?

Return Type Numeric

Default 0

SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer:ATTen<va [,src]

(Write-Read) Sets and returns the Source Attenuator setting for the Cal All calibration.

Parameters

<n> Source port number.

<value> Attenuation value in dB for the Cal All calibration. Choose a valid value for the PNA model. See [valid settings](#).

[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 `SYST:CAL:ALL:PORT2:SOUR:POW:ATT 10`

Query Syntax SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer:ATTen?

Return Type Numeric

Default 0

SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer:OFFSet <value>[,src]

(Write-Read) Sets and returns the power offset value for a Cal All calibration.

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 reflects the added components.

Parameters

<n> Source port number.

<value> Power offset value in dB for a Cal All calibration.

- For amplification, use positive offset.
- For attenuation, use negative offset.

[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 **SYST:CAL:ALL:PORT2:SOUR:POW:OFFS 10**

Query Syntax SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer:OFFSet?

Return Type Numeric

Default 0

SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer[:VALue] <value>[,src]

(Write-Read) Sets and returns the power level at which a Cal All calibration is to be performed.

Parameters

<n> Source port number.

<value> Power level at which the calibration is to be performed.

[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 **SYST:CAL:ALL:PORT2:SOUR:POW 0**

Query Syntax SYSTem:CALibrate:ALL:PORT<n>:SOURce:POWer[:VALue]?

Return Type Numeric

Default Preset power of the PNA model.

See the data sheet for the power level for each model.

SYSTem:CALibrate:ALL:RESet

(Write-only) Resets all properties associated with the Cal All session to their default values.

Parameters None

Examples SYST:CAL:ALL:RES

Query Syntax Not Applicable

Default Not Applicable

SYSTem:CALibrate:ALL:SElect <value>

(Write-Read) Sets and returns the list of channels to be calibrated during the Cal All session.

Parameters

<value> Channel numbers to be calibrated. These channels must already exist.

Examples SYST:CAL:ALL:SEL 1,2,3

Query Syntax SYSTem:CALibrate:ALL:SElect?

Return Type Comma-separated channel numbers.

Default Existing channels

Last modified:

18-Mar-2014 Added opt argument to CAT?
10-Feb-2014 Added links to properties and values
19-Dec-2013 Added optional source port strings
19-Sep-2012 Removed LO cal and minor edits (JK)
5-Jan-2012 New topic

SYSTem:CAPacity:HARDware:MODule Commands

Returns various capabilities of the PXIe Module.

SYSTem:CAPacity:HARDware:MODule:

COUNT?

SERial?

MODEl?

OPT?

FPGA?

| **COUNT?**

Click on a **red** keyword to view the command details.

see Also

- [Example Programs](#)
 - [Guided Cal commands](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:CAPacity:HARDware:MODule:COUNT?

(Read-only) Returns the number of modules that are part of the current VNA instance.
[Learn more about Multiport VNA.](#)

Parameters None

Examples SYST:CAP:HARD:MOD:COUNT?

Return Type Numeric

Default Not Applicable

SYSTem:CAPacity:HARDware:MODUle<n>:SERial?

(Read-only) Returns the serial number of the specified module.

Parameters

<n> Module number. This is the order in which the module appears in the Multiport VNA. [Learn more about module number.](#)

Examples SYST:CAP:HARD:MOD3:SERial?

Return Type String

Default Not Applicable

SYSTem:CAPacity:HARDware:MODule<n>:MODel?

(Read-only) Returns the model number of the specified module. [See frequency ranges of VNA models.](#)

Parameters

<n> Module number. This is the order in which the module appears in the Multiport VNA. [Learn more about module number.](#)

Examples SYST:CAP:HARD:MOD3:MODel?

Return Type String

Default Not Applicable

SYSTem:CAPacity:HARDware:MODule<n>:OPT?

(Read-only) Returns the options that are installed in the specified module. [See a list of VNA options.](#)

Parameters

<n> Module number. This is the order in which the module appears in the Multiport VNA. [Learn more about module number.](#)

Examples

```
SYST:CAP:HARD:MOD3:OPT?
```

Possible return string: "010,551"

Return Type A comma-separated string.

Default Not Applicable

SYSTem:CAPacity:HARDware:MODule<n>:FPGA:COUNt?

(Read-only) Returns the number of FPGA boards in the module.

Parameters

<n> Module number. This is the order in which the module appears in the Multiport VNA. [Learn more about module number.](#)

Examples SYST:CAP:HARD:MOD3:FPGA:COUNt?

Return Type Numeric

Default Not Applicable

SYSTem:CAPacity:HARDware:MODule<n>:FPGA<x>?

(Read-only) Returns the version number of specified FPGA board.

Parameters

- <n> Module number. This is the order in which the module appears in the Multiport VNA. [Learn more about module number.](#)
- <x> FPGA board number. Choose from 1 to the value returned by [SYST:CAP:HARD:MOD<n>:FPGA:COUNt?](#)

Examples SYST:CAP:HARD:MOD3:FPGA2?

Return Type String

Default Not Applicable

System:FIFO Commands

The 4 GB FIFO data buffer is available with Option 118 on the [PNA-X](#) and [N5264A](#). These commands control data in and out of FIFO data buffer. The FIFO can be emptied as it is being filled, which means that the PNA can be used to acquire an infinite amount of data.

The data placed into the FIFO is the raw data after averaging and ratioing has been applied, but prior to any calibration, formatting, or data analysis functions.

SYSTem:FIFO

DATA

| [CLEar](#)

| [COUNT?](#)

[\[:STATe\]](#)

Click on a [red](#) keyword to view the command details.

see Also

- FIFO and other Antenna Features
 - [Fast CW command](#)
 - FIFO Example Program
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:FIFO:DATA? <dpoints>

(Read-only) Reads the next specified number of data points from the FIFO buffer. Each data point is returned as a real/imaginary pair. Data is cleared as it is read.

Parameters

<dPoints> Number of data points to read. An error is returned if the amount of requested data is larger than the available data.

Examples `SYST:FIFO:DATA? 1e6`
`system:fifo:data? 1e3`

Return Type Use **FORMat:DATA** to change the data type (<REAL,32>, <REAL,64> or <ASCii,0>). For best results, use REAL,32
Use **FORMat:BOReR** to change the byte order. Use “NORMal” when transferring a binary block from LabView or Vee. For other programming languages, you may need to "SWAP" the byte order.
Each data point is returned as a real/imaginary pair.

Default Not applicable

SYSTem:FIFO:DATA:CLEar

(Write-only) Clears the data from the FIFO buffer.

Parameters None

Examples SYST:FIFO:DATA:CLE
system:fifo:data:clear

Return Type None

Default Not applicable

SYSTem:FIFO:DATA:COUNT?

(Read-only) Returns the total number of data points in the FIFO buffer.

Parameters None

Examples SYST:FIFO:DATA:COUN?
' returns 5.07e6

Return Type Numeric

Default Not applicable

SYSTem:FIFO[:STATe] <bool>

(Write-Read) Sets and returns the state of data storage to the FIFO buffer. Syst:Preset or an instrument state recall also ends storage to the FIFO buffer. The FIFO buffer is cleared when set to OFF.

Parameters

<bool> FIFO buffer state. Choose from:
ON or 1 Data is stored in the FIFO buffer.
OFF or 0 Data is NOT stored in the FIFO buffer.

Examples `SYST:FIFO 1`
`system:fifo:state off`

Query Syntax SYSTem:FIFO[:STATE]?

Return Type Boolean

Default 0 OFF

System Preferences Commands

Sets and reads the PNA Preferences settings.

SYSTem:PREFerences

| **DEFault**

| **ITEM**

| **EEXTrapolate**

| **EDEV: DPOLicy**

| **GDELay:TWOPoint**

| **KEYS**

| **MCControl**

| **MCMethod**

| **MCPreset**

| **MRU**

| **OFFSet**

| **RCV**

| **SRC**

| **PRESet:POWer:STATe**

| **PSRTrace**

| **QStart**

| **RECeivers**

| **CERRor**

| **OVERload:POWer**

| **REDLimits**

| **REFMarker**

| **RETRace:POWer**

| **RTOF**

| **SWITCh:DEF**

Click on a keyword to view the command details.

see Also

- [SENS:CORRection:PREFErences](#)
 - [Learn about PNA Preferences](#)
 - [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:PREFerences:DEFault

(Write-only) Resets the PNA preferences to their default settings. Some default settings vary depending on the PNA Model. [Learn more about PNA Preferences.](#)

Examples SYST:PREF:DEF system:preferences:default

Query Syntax Not Applicable

Default Not Applicable

SYSTem:PREFerences:ITEM:EDEV:DPOLicy <bool>

(Read-Write) 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.

Parameters

<bool> Choose from:

OFF (0) External devices **remain active** when the PNA is Preset or when a Instrument State is recalled.

ON (1) External devices are **de-activated** (SYST:CONF:EDEV:STAT to OFF) when the PNA is Preset or when a Instrument State is recalled.

Examples

```
SYST:PREF:ITEM:EDEV:DPOL 1  
system:preferences:item:edev:dpolicy OFF
```

Query Syntax SYSTem:PREFerences:ITEM:EDEV:DPOLicy?

Return Type Boolean

Default ON or 1

SYSTem:PREFerences:ITEM:EEXTrapolate <bool>

(Read-Write) Sets whether a Swept IMD or IMDx calibration can exceed the stop frequency limit of an ECal module. [Learn more.](#)

Parameters

<bool> Choose from:
ON (1) Allow extrapolation.
OFF (0) Do NOT allow extrapolation.

Examples

```
SYST:PREF:ITEM:EEXT 1  
system:preferences:item:eextrapolate OFF
```

Query Syntax SYSTem:PREFerences:ITEM:EEXTrapolate?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:GDELaY:TWOPoint <bool>

(Read-Write) Sets the default group delay aperture setting. [Learn more about group delay aperture.](#)

Parameters

<bool> Choose from:

ON (1) Sets default group delay aperture to 2 points.

OFF (0) Sets default group delay aperture to 11 points.

Examples

```
SYST:PREF:ITEM:GDELaY:TWOPoint 1  
system:preferences:item:gdelay:twopoint OFF
```

Query Syntax SYSTem:PREFerences:ITEM:GDELaY:TWOPoint?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:KEYS <bool>

(Read-Write) Set and return whether the keys are displayed or not.

Parameters

<bool> Choose from:
ON (1) – Turn keys on.
OFF (0) – Turn keys off.

Examples

```
SYST:PREF:ITEM:KEYS 1  
system:preferences:item:keys OFF
```

Query Syntax SYSTem:PREFerences:ITEM:KEYS?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:MCCControl <bool>

(Read-Write) Set and return whether the Coupled Markers setting controls the ON|OFF state of markers that are coupled. [Learn more about Coupled Markers](#). Refer also to [CALC:MARK:COUP:STATe ON](#).

Parameters

<bool> Choose from:

ON (1) – With Coupled Markers ON, when a marker is turned on, the same-numbered marker on all coupled traces will also be turned on. Likewise, turning off a marker will turn it off on all coupled traces.

OFF (0) – Turning a marker on or off will have no effect on the markers on other traces.

Examples

```
SYST:PREF:ITEM:MCC 1
```

```
system:preferences:item:mcccontrol OFF
```

Query Syntax SYSTem:PREFerences:ITEM:MCCControl?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:MCMMethod <bool>

(Read-Write) Set and return whether Coupled Markers is set to Channel or All after Preset. [Learn more about Coupled Markers](#). Refer also to [CALC:MARK:COUP:STATE ON](#) and [SYST:PREF:ITEM:MCPR ON](#).

Parameters

<bool> Choose from:
ON (1) – Marker Coupling Method is set to Channel after Preset.
OFF (0) – Marker Coupling Method is set to ALL after Preset.

Examples

```
SYST:PREF:ITEM:MCM 1  
system:preferences:item:mcmethod OFF
```

Query Syntax SYSTem:PREFerences:ITEM:MCMMethod?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:MCPRest <bool>

(Read-Write) Set and return whether Coupled Markers is set to ON or OFF after Preset.
[Learn more about Coupled Markers.](#)

Parameters

<bool> Choose from:
OFF (0) – Coupled Markers is OFF after Preset.
ON (1) – Coupled Markers is ON after Preset.

Examples

```
SYST:PREF:ITEM:MCPR 1  
system:preferences:item:mcpreset OFF
```

Query Syntax SYSTem:PREFerences:ITEM:MCPRest?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:MRU <bool>

(Read-Write) Set and return whether to list files for recall on softkeys by most-recently used or alphabetically.

Parameters

<bool> Choose from:
ON (1) – Recall softkeys show most recently-used files.
OFF (0) – Recall softkeys show alphabetically-ordered files.

Examples

```
SYST:PREF:ITEM:MRU 1  
system:preferences:item:mru OFF
```

Query Syntax SYSTem:PREFerences:ITEM:MRU?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:OFFSet:RCV <bool>

(Read-Write) Set and return whether to offset the test port receivers by the amount of receiver attenuation. [Learn more.](#)

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.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

Parameters

<bool> Choose from:
ON (1) Offset the test port receivers
OFF (0) Do NOT offset the test port receivers

Examples `SYST:PREF:ITEM:OFFS:RCV 1`
`system:preferences:item:offset:rcv OFF`

Query Syntax `SYSTem:PREFerences:ITEM:OFFSet:RCV?`

Return Type Boolean

Default PNA-L and E836xB: **OFF** (does NOT offset the display).
PNA-X: **ON** (offsets the display).

SYSTem:PREFerences:ITEM:OFFSet:SRC <bool>

(Read-Write) Set and return whether to offset the reference receiver by the amount of source attenuation. [Learn more.](#)

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.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

Parameters

<bool> Choose from:
ON (1) Offset the reference receivers.
OFF (0) Do NOT Offset the reference receivers.

Examples

```
SYST:PREF:ITEM:OFFS:SRC 1  
system:preferences:item:offset:src OFF
```

Query Syntax SYSTem:PREFerences:ITEM:OFFSet:SRC?

Return Type Boolean

Default All models: **ON** (offset the display).

SYSTem:PREFerences:ITEM:PRESet:POWer[:STATe] <char>

(Read-Write) Set and return the Preset Power ON/OFF state. [Learn more.](#)

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

Parameters

<char> Choose from:

ON - Instrument Preset always turns RF power ON.

AUTO - When the current power setting is OFF, leave power OFF after Preset. When the current power setting is ON, turn power ON after Preset.

Examples

```
SYST:PREF:ITEM:PRE:POW ON
system:preferences:item:preset:power:state auto
```

Query Syntax SYSTem:PREFerences:ITEM:PREset:POWer[:STATe]?

Return Type Character

Default ON

SYSTem:PREFerences:ITEM:PSRTrace <char>

(Read-Write) 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.

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.

This setting remains until changed again using this command, or until the hard drive is changed or reformatted.

Parameters

<char> Choose from:

START - Maintain source power at the start power level.

STOP - Maintain source power at the stop power level.

Examples

```
SYST:PREF:ITEM:PSRT STOP  
system:preferences:item:psrtrace start
```

Query Syntax SYSTem:PREFerences:ITEM:PSRTrace?

Return Type Character

Default START

SYSTem:PREFerences:ITEM:QSTart <bool>

(Read-Write) This command controls the on/off state of the preference, "On PRESET show Quick Start dialog".

Parameters

<bool> Choose from:

ON (1) Display the Quick Start dialog on PRESET.

OFF (0) Do not display the Quick Start dialog on PRESET.

Examples

```
SYST:PREF:ITEM:QST 1  
system:preferences:item:qstart OFF
```

Query Syntax SYSTem:PREFerences:ITEM:QST?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:RECeivers:CERRor[:STATe] <bool>

(Read-Write) Set and return whether to display receiver overload warnings. [Learn more.](#)

Parameters

<bool> Choose from:
ON (1) Display overload warnings,
OFF (0) Do NOT display overload warnings.

Examples

```
SYST:PREF:ITEM:REC:CERR 1  
system:preferences:item:receivers:cerror:state OFF
```

Query Syntax SYSTem:PREFerences:ITEM:RECeivers:CERRor[:STATe]?

Return Type Boolean

Default ON

SYSTem:PREFerences:ITEM:RECEivers:OVERload:POWer[:ST <bool>

(Read-Write) Set and return whether to turn source power OFF when a receiver is overloaded. [Learn more.](#)

Parameters

<bool> Choose from:
ON (1) Turn OFF source power to ALL ports when a receiver is overloaded.
OFF (0) Power remains ON when a receiver is overloaded.

Examples **SYST:PREF:ITEM:REC:OVER:POW 1**
system:preferences:item:receivers:overload:power:state OFF

Query Syntax SYSTem:PREFerences:ITEM:RECEivers:OVERload:POWer[:STAtE]?

Return Type Boolean

Default OFF (0)

SYSTem:PREFerences:ITEM:REDLimits <bool>

(Read-Write) Set and return whether to draw limits lines in Red or the trace color.

Parameters

<bool> Choose from:

ON (1) All Limit lines are drawn in Red.

OFF (0) Limit lines are drawn the same color as the trace.

Examples SYST:PREF:ITEM:REDL 1

system:preferences:item:redlimits OFF

Query Syntax SYSTem:PREFerences:ITEM:REDLimits?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:REFMarker <bool>

(Read-Write) Set and return whether to treat marker 10 as a reference marker. [Learn more.](#)

Parameters

<bool> Choose from:

ON (1) Marker 10 is always a reference marker (Pre A.10.40 behavior).

OFF (0) Marker 10 is just another marker. See [Reference Marker commands](#)

Examples

```
SYST:PREF:ITEM:REFM 1
```

```
system:preferences:item:refmarker OFF
```

Query Syntax SYSTem:PREFerences:ITEM:REFMarker?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:RETRace:POWer <char>

(Read-Write) For single-band frequency or segment sweeps ONLY, specify whether to turn RF power ON or OFF during a retrace. [Learn more about RF power during sweep retrace.](#)

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.

This setting remains until changed using this command, or until the hard drive is changed or reformatted.

Parameters

<char> Choose from:

AUTO: Power is left ON during retrace of single-band frequency or segment sweeps ONLY.

OFF: Power is turned OFF during retrace of single-band frequency or segment sweeps ONLY.

Examples

```
SYST:PREF:ITEM:RETR:POW OFF
```

```
system:preferences:item:retrace:power auto
```

Query Syntax SYSTem:PREFerences:ITEM:RETRace:POWer?

Return Type Character

Default AUTO

SYSTem:PREFerences:ITEM:RTOF <bool>

(Read-Write) Set and return whether to display limit line failures as red trace segments or red data points (dots). [Learn more.](#)

Parameters

<bool> Choose from:
ON (1) Display failures as red trace segments. (Red Trace On Fail).
OFF (0) Display failures as red data points (dots).

Examples

```
SYST:PREF:ITEM:RTOF 1  
system:preferences:item:rtof OFF
```

Query Syntax SYSTem:PREFerences:ITEM:RTOF?

Return Type Boolean

Default OFF

SYSTem:PREFerences:ITEM:SWITCh:DEF <string>, <int>

(Read-Write) Sets the default setting for the Noise Tuner switch. This is the setting that occurs when a new channel is created. [Learn more.](#)

This command will return an error on PNA models with a built-in Noise tuner.

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.

This setting remains until changed using this command, or until the hard drive is changed or reformatted.

Parameters

<string> Name of the switch to set. Choose from:

- **"Port1NoiseTuner"**

<int> Value to set. Choose from:

- 0** Sets the default (preset) to INTERNAL
- 1** Sets the default (preset) to EXTERNAL

Examples

```
SYST:PREF:ITEM:SWIT:DEF "Port1NoiseTuner" 1 'Write  
system:preferences:item:switch:def? "Port1NoiseTuner" 'Read
```

Query Syntax SYSTem:PREFerences:ITEM:SWITCh:DEF? <switch>

Return Type Integer

Default 1 (External)

Last Modified:

- 19-Feb-2015 Added Coupled Markers (3), Ref Marker, and Red Limits
- 1-Aug-2011 Added MRU
- 30-Aug-2010 Added receiver overload
- 30-Aug-2010 New topic started from System level

PXI Shared Memory Data Transfer

The fastest way to transfer data out of a PXIe VNA is to use shared memory. Due to the nature of shared memory, these commands can ONLY be used to transfer data between your program and the VNA when your program runs locally on the same computer as the VNA.

Shared Memory Background: Usually, each process has its own memory, and anything saved in one process isn't visible to another process. With shared memory, both processes actually connect to the same memory, so what happens in one process is now visible to another process.

How to setup Shared Memory

1. Send **SYST:DATA:MEM:INITialize**. This tells the VNA to prepare for the creation of shared memory.
2. Decide which measurements to include in the shared memory. For each of those measurements, send **SYST:DATA:MEM:ADD**. This adds a request to copy the contents of those measurements into the shared memory. This does not start adding data.
3. Send **SYST:DATA:MEM:COMMit**. This command tells the VNA to create the Windows shared memory resource and to give that shared memory resource a name.
4. You can now create a handle to the shared memory and start reading the data from that shared memory. Once setup, the VNA remembers the setup, and the shared memory will be filled on each sweep. You do not send any more SCPI commands unless you need to change the list of requested measurements to include in the shared memory.

[See C# example](#)

SYSTem: DATA:MEMory Commands

Controls Shared Memory.

The commands are listed in order of recommended use.

SYSTem:DATA:MEMory :

[INITialize](#)

[ADD](#)

[OFFSet?](#)

[NAME?](#)

[COMMit](#)

[SIZE?](#)

[DELete](#)

[RESet](#)

Click on a keyword to view the command details.

see Also

- [SCPI Command Tree](#)
-

SYSTEM:DATA:MEMORY:INITIALIZE

(Write only) Initializes the shared memory setup buffers.

Parameters None

Examples [See an example program](#)

Query Syntax Not Applicable

Default Not Applicable

SYSTem:DATA:MEMory:ADD <string>

(Write only) Add a request to copy the contents of the specified measurement into shared memory. Call this command once for each measurement definition. Once the shared memory is setup, there is no need to send more SCPI commands. The shared memory is filled automatically on every sweep.

Parameters

<string> The following elements separated with colons (:).

<Ch>:<MeasNum>:<dataFormat>:<numPoints>

where:

- <Ch> - Channel number of the measurement to share memory. Use **SYST:ACTive:CHAN** to return the active channel number. Use **SYST:CHAN:CAT?** to return the channel numbers in use.
- <MeasNum> - Measurement (**Tr**) number to share memory. Use **CALC<ch>:PAR:MNUM?** just after the trace is created to read the measurement number. See also: **Referring to Traces, Measurements, Channels, and Windows Using SCPI**.
- <dataFormat> - Choose from:
 - **SDATA** - Complex measurement 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.
 - **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 one number per data point for all other formats.
 - Format of the read data is same as the displayed format.
- <numPoints> - Number of data points in the measurement

trace.

Examples

SYSTem:DATA:MEMory:ADD "2:3:SDATA:201"

'copies the data for channel #2, measurement #3, complex data, 201 points into the shared memory buffer.

[See an example program](#)

Query Syntax Not Applicable

Default Not Applicable

SYSTem:DATA:MEMory:OFFSet?

(Read only) The shared memory is a contiguous block of memory. Each measurement takes up a subset of this contiguous block. This command returns the offset (in bytes) into the shared memory for the most recently added parameter. The offset is a number that specifies the starting index (in bytes) of the data.

This query can be sent after sending **SYST:DATA:MEM:ADD**.

Parameters None

Examples [See an example program](#)

Return Type Numeric

Default Not Applicable

SYSTEM:DATA:MEMORY:NAME?

(Read only) Returns a unique, auto-generated name that can be used in the COMMIT command. By using this generated name, a client can be sure not to conflict with any other used shared memory regions.

Parameters None

Examples [See an example program](#)

Return Type String

Default Not Applicable

SYSTem:DATA:MEMory:COMMit <memName>

(Write only) Allocates the memory mapped buffer.

Parameters

<memName> String. Name of the memory mapped buffer. This must be a unique name, and cannot conflict with other shared memory buffer names. Use this command in your program when connecting to the shared memory.

See [SYSTem:DATA:MEMory:NAME?](#)

Examples

[See an example program](#)

Query Syntax Not Applicable

Default Not Applicable

SYSTem:DATA:MEMory:SIZE?

(Read only) Returns the size of the memory mapped region. Send this immediately after **SYST:DATA:MEM:COMMit**. The result is the total size (in bytes) of all the measurements in the shared memory region.

Parameters None

Examples [See an example program](#)

Return Type Numeric

Default Not Applicable

SYSTEM:DATA:MEMORY:CATALOG?

(Read only) Returns a list of all the allocated shared memory buffers

Parameters None

Examples [See an example program](#)

Return Type String

Default Not Applicable

SYSTEM:DATA:MEMORY:RESet

(Write only) Deletes all allocated shared memory buffers

Parameters None

Examples [See an example program](#)

Query Syntax Not Applicable

Default Not Applicable

SYSTem:DATA:MEMory:DELete <memName>

(Write only) Allocates the specified memory mapped buffer.

Parameters

<memName> String. Name of the memory mapped buffer. This is the unique name that is used in the **COMMit** command.

Examples [See an example program](#)

Query Syntax Not Applicable

Default Not Applicable

Example Program

```
static void Main(string[] args)
{
    // Connect to the hislip VISA address of localhost
    ResourceManager resourceManager = new ResourceManager();
    FormattedIO488 formattedIO = new FormattedIO488();
    formattedIO.IO =
    (IMessage)resourceManager.Open("TCPIP0::localhost::hislip0::INSTR");

    // Destroy all measurements and add a window
    formattedIO.WriteString("SYST:FPR\n");
    formattedIO.WriteString("DISP:WIND:STAT 1");

    // initialize memory mapped structures in VNA
    formattedIO.WriteString("SYST:DATA:MEM:INIT\n");

    // Create 4 SParameters
    string[] parameters = new string[] { "S11", "S21", "S12", "S22" };
    int[] offsets_for_complex_data = new int[parameters.Length];
```

```

int[] offsets_for_formatted_data = new int[parameters.Length];
for (int i = 0; i < parameters.Length; i++)
{
    // Create a new parameter
    formattedIO.WriteString("CALC:PAR:DEF " + parameters[i] + "," +
parameters[i]);
    formattedIO.WriteString("DISP:WIND:TRAC" + (i + 1).ToString() + ":FEED "
+ parameters[i] + "");

    // Configure a new section of the memory map to monitor the complex data of
this parameter
    formattedIO.WriteString("SYST:DATA:MEM:ADD '1:" + (i + 1).ToString() +
":SDATA:201"); // add parameter to memory mapped
    formattedIO.WriteString("SYST:DATA:MEM:OFFSet?");
    offsets_for_complex_data[i] = int.Parse(formattedIO.ReadString());

    // Configure a new section of the memory map to monitor the formatted data of
this parameter
    formattedIO.WriteString("SYST:DATA:MEM:ADD '1:" + (i + 1).ToString() +
":FDATA:201"); // add parameter to memory mapped
    formattedIO.WriteString("SYST:DATA:MEM:OFFSet?");
    offsets_for_formatted_data[i] = int.Parse(formattedIO.ReadString());
}

// Tell the VNA to allocate the memory map. Name it "VNA_MemoryMap"
formattedIO.WriteString("SYST:DATA:MEM:COMM 'VNA_MemoryMap");

// Query the size of the memory map
formattedIO.WriteString("SYST:DATA:MEM:SIZE?");
int size = int.Parse(formattedIO.ReadString());

// Create the memory map in C#. This requires .NET 4.5 framework
MemoryMappedFile mappedFile =
MemoryMappedFile.CreateOrOpen("VNA_MemoryMap", size);
MemoryMappedViewAccessor mappedFileView =
mappedFile.CreateViewAccessor();

```

```

// Trigger a single sweep, and wait for it to complete
formattedIO.WriteString("SENS:SWE:MODE SING");
formattedIO.WriteString("*OPC?");
formattedIO.ReadString();

// Allocate buffers to hold the output data
float[][] complexData = new float[parameters.Length][];
for (int i = 0; i < complexData.Length; i++)
{
    complexData[i] = new float[402];
}

float[][] formattedData = new float[parameters.Length][];
for (int i = 0; i < formattedData.Length; i++)
{
    formattedData[i] = new float[201];
}

// Copy the data from the memory map into the output buffers
// These copy the data from the in-process memory map.
// This runs very fast - and is just a "memcpy" under the hood
for (int i = 0; i < parameters.Length; i++)
{
    ReadBytes(mappedFileView, offsets_for_complex_data[i], 402, complexData[i]);
    ReadBytes(mappedFileView, offsets_for_formatted_data[i], 201,
formattedData[i]);
}

// Output some data to show that it worked
System.Console.WriteLine(complexData[0][0].ToString()); // Output first point of
S11 in complex
System.Console.WriteLine(formattedData[3][200].ToString()); // Output last point
of S22 as formatted

```

```
}

static public unsafe void ReadBytes(MemoryMappedViewAccessor mappedFileView,
    int offset, int num, float[] arr)
{
    // This is equivalent to:
    //     //m_mappedFileView.ReadArray<float>(m_sharedMemoryOffsets[i-1],
complexArray, 0, points*2);
    // But, using this "unsafe" code is 30 times faster. 100usec versus 3ms
    byte* ptr = (byte*)0;
    mappedFileView.SafeMemoryMappedViewHandle.AcquirePointer(ref ptr);
    System.Runtime.InteropServices.Marshal.Copy(IntPtr.Add(new IntPtr(ptr), offset),
arr, 0, num);
    mappedFileView.SafeMemoryMappedViewHandle.ReleasePointer();
}
}
```

SYSTem:CAPability Commands

Reads various capabilities of the analyzer.

SYSTem:CAPability:

ALC:POWer:

| **MAXimum[:LEVel]?**

| **MINimum[:LEVel]?**

CHANnels:MAXimum[:COUNT]?

FOM:EXISts?

HARDware:

| **ATTenuator:RECeiver:**

| **EXISts?**

| **MAXimum?**

| **STEP[:SIZE]?**

| **ATTenuator:SOURce:**

| **MAXimum?**

| **STEP[:SIZE]?**

| **DC:RECeiver**

| **INTernal:CATalog?**

| **INTernal:COUNT?**

| **DC:SOURce**

| **INTernal:CATalog?**

| **INTernal:COUNT?**

| **PORTs:**

| **CATalog?**

| **COUNT?**

| **INTernal**

| **CATalog?**

| **COUNT?**

| **PNUMBER?**

| **SOURce**

<ul style="list-style-type: none"> CATalog? COUNt? INTernal <ul style="list-style-type: none"> CATalog? COUNt? RBSWitch:EXISts? RECeiver: <ul style="list-style-type: none"> INTernal <ul style="list-style-type: none"> COUNt? SOURce:COUNt?
<p>NBW:</p> <ul style="list-style-type: none"> NOISe:CATalog? STD:CATalog?
<p>PRESet:FREQuency:</p> <ul style="list-style-type: none"> MAXimum? MINimum?
<p>RBW:IMS:CATalog?</p>
<p>WINDows</p> <ul style="list-style-type: none"> MAXimum[:COUNt]? TRACes:MAXimum[:COUNt]?

Click on a **red** keyword to view the command details.

see Also

- [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:CAPability:ALC:POWer:MAXimum[:LEVel]? <srcNum>

(Read-only) Returns the maximum leveled source power setting in dB. [Learn more about leveled source power.](#)

Parameters

<srcNum> Source Number. Choose from 1 or 2.

Examples SYST:CAP:ALC:POW:MAX? 1

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:ALC:POWer:MINimum[:LEVel]? <srcNum>

(Read-only) Returns the minimum leveled source power setting in dB with 0 dB attenuation. [Learn more about leveled source power.](#)

Parameters

<srcNum> Source Number. Choose from 1 or 2.

Examples SYST:CAP:ALC:POW:MIN? 1

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:CHANnels:MAXimum[:COUNt]?

(Read-only) Returns the maximum possible number of channels. [Learn more about Channels.](#)

Parameters None

Examples SYST:CAP:CHAN:MAX?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:FOM:EXISts?

(Read-only) Returns whether or not the analyzer has FOM Opt. 080 installed. [Learn more.](#)

Parameters None

Examples SYST:CAP:FOM:EXIS?

Return Type Boolean
1 - Yes, FOM is installed.
0 - No, FOM is NOT installed.

Default Not Applicable

SYSTem:CAPability:HARDware:ATTenuator:RECeiver:EXISts? <portNum>

(Read-only) Returns whether or not there is a receiver attenuator on the specified port.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples SYST:CAP:HARD:ATT:REC:EXIS? 2

Return Type Boolean

1 - Yes, the test port has a receiver attenuator.

0 - No, the test port does NOT have a receiver attenuator.

Default Not Applicable

SYSTem:CAPability:HARDware:ATTenuator:RECeiver:MAXim <portNum>

(Read-only) Returns the maximum amount of receiver attenuation on the specified port.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples **SYST:CAP:HARD:ATT:REC:MAX? 2**

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:ATTenuator:RECeiver:STEP[:S <portNum>

(Read-only) Returns the step size of the receiver attenuator on the specified port.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples SYST:CAP:HARD:ATT:REC:STEP? 2

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:ATTenuator:SOURce:MAXimu <portNum>

(Read-only) Returns the maximum amount of source attenuation on the specified port.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples **SYST:CAP:HARD:ATT:SOUR:MAX? 2**

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:ATTenuator:SOURce:STEP[:SI <portNum>

(Read-only) Returns the step size of the source attenuator on the specified port.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples SYST:CAP:HARD:ATT:SOUR:STEP? 2

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:DC:RECeiver:INTernal:CATalogue

(Read-only) Returns a list of names of the internal DC receivers.

Parameters None

Examples **SYST:CAP:HARD:DC:REC:INT:CAT?**

Return Type String of internal DC receivers separated by commas.
For example, "AI1,AI2,AIG,AOS1,AOS2"

Default Not Applicable

SYSTem:CAPability:HARDware:DC:RECeiver:INTernal:COUN

(Read-only) Returns the number of internal DC receivers in the analyzer.

Parameters None

Examples SYST:CAP:HARD:DC:REC:INT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:DC:SOURce:INTernal:CATalog

(Read-only) Returns a list of names of the internal DC sources.

Parameters None

Examples **SYST:CAP:HARD:DC:SOUR:INT:CAT?**

Return Type String of internal DC sources separated by commas.
For example, "AO1,AO2"

Default Not Applicable

SYSTem:CAPability:HARDware:DC:SOURce:INTernal:COUNt?

(Read-only) Returns the number of internal DC sources in the analyzer.

Parameters None

Examples SYST:CAP:HARD:DC:SOUR:INT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:CATalog?

(Read-only) Returns a list of test port names including external testset ports.

Parameters None

Examples SYST:CAP:HARD:PORT:CAT?

Return Type String of port names separated by commas.
For example, "Port 1,Port 2,Port 3,Port 4".

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:COUNt?

(Read-only) Returns the number of test ports including external testset ports.

Parameters None

Examples SYST:CAP:HARD:PORT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:INTernal:CATalog?

(Read-only) Returns a list of internal test port names.

Parameters None

Examples SYST:CAP:HARD:PORT:INT:CAT?

Return Type String of port names separated by commas.
For example, "Port 1,Port 2,Port 3,Port 4".

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:INTernal:COUNt?

(Read-only) Returns the number of internal test ports.

Parameters None

Examples SYST:CAP:HARD:PORT:INT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:PNUMber?

<portName>

(Read-only) Returns the port number associated with the specified port name.

Parameters None

<portName> String. Port name. Use **SYST:CAP:HARD:PORT:CAT?** to return a list of valid port names.

Examples SYST:CAP:HARD:PORT:PNUM? "Port 1"

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:SOURce:CATalog?

(Read-only) Returns a list of source port names, including any configured external sources.

Parameters None

Examples SYST:CAP:HARD:PORT:SOUR:CAT?

Return Type String of source port names separated by commas.
For example, "Port 1,Port 2,Port 3,Port 4, Port 1 Src 2".

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:SOURce:COUNt?

(Read-only) Returns the number of source ports, including any configured external sources.

Parameters None

Examples SYST:CAP:HARD:PORT:SOUR:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:SOURce:INTernal:CAT

(Read-only) Returns a list of internal source port names.

Parameters None

Examples **SYST:CAP:HARD:PORT:SOUR:INT:CAT?**

Return Type String of internal source port names separated by commas.
For example, "Port 1,Port 2,Port 3,Port 4, Port 1 Src 2"

Default Not Applicable

SYSTem:CAPability:HARDware:PORTs:SOURce:INTernal:COU

(Read-only) Returns the number of internal source ports.

Parameters None

Examples SYST:CAP:HARD:PORT:SOUR:INT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:RBSWitch:EXISts? <portNum>

(Read-only) Returns whether or not the specified port number has a reference bypass switch.

Parameters

<portNum> Port number. Choose from the number of test ports on the analyzer.

Examples SYST:CAP:HARD:RBSW:EXIS? 2

Return Type Boolean

1 - Yes, the test port has a reference bypass switch.

0 - No, the test port does NOT have a reference bypass switch.

Default Not Applicable

SYSTem:CAPability:HARDware:RECeiver:INTernal:COUNt?

(Read-only) Returns the number of receivers in the analyzer.

Parameters None

Examples SYST:CAP:HARD:REC:INT:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:HARDware:SOURce:COUNt?

(Read-only) Returns the number of sources in the analyzer.

Parameters None

Examples SYST:CAP:HARD:SOUR:COUN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:NBW:NOISe:CATalog?

(Read-only) Returns the list of supported Noise Bandwidth values when using a noise receiver (option 029). Learn more about Opt. 029.

Parameters None

Examples SYST:CAP:NBW:NOIS:CAT?

Return Type Variant array of string values

Default Not Applicable

SYSTem:CAPability:NBW:STD:CATalog?

(Read-only) Returns the list of supported Noise Bandwidth values when using the NA receiver for noise measurements (option 028). [Learn more about Opt 028.](#)

Parameters None

Examples SYST:CAP:NBW:STD:CAT?

Return Type Variant array of string values

Default Not Applicable

SYSTem:CAPability:PRESet:FREQuency:MAXimum?

(Read-only) Returns the maximum specified frequency of the analyzer. Does not include any over-sweep. See also: **SYST:CAP:FREQ:MAX?**

Parameters None

Examples SYST:CAP:PRESet:FREQ:MAX?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:PRESet:FREQuency:MINimum?

(Read-only) Returns the minimum specified frequency of the analyzer. Does not include any under-sweep. See also: **SYST:CAP:FREQ:MIN?**

Parameters None

Examples SYST:CAP:PRESet:FREQ:MIN?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:RBW:IMS:CATalog?

(Read-only) Returns the list of supported Resolution BW values for the IMSpectrum channel. Learn more about IMSpectrum.

Parameters None

Examples SYST:CAP:RBW:IMS:CAT?

Return Type Variant array of string values

Default Not Applicable

SYSTem:CAPability:WINDows:MAXimum[:COUNT]?

(Read-only) Returns the maximum number of windows. [Learn more.](#)

Parameters None

Examples SYST:CAP:WIND:MAX?

Return Type Numeric

Default Not Applicable

SYSTem:CAPability:WINDows:TRACes:MAXimum[:COUNt]?

(Read-only) Returns the maximum number of traces per window. [Learn more.](#)

Parameters None

Examples SYST:CAP:WIND:TRAC:MAX?

Return Type Numeric

Default Not Applicable

SYSTem: COMMunicate Commands

Controls and queries settings that affect the PNA system.

SYSTem: COMMunicate:
GPIB
PMETer
ADDReSS
RDEVice
CLOSe
OPeN
READ?
RESet
WBINary
WBLock
WRITe
LAN:HOSTname
PSENSor
TCPip:CONTrol?
USB:PMETer:CAT?
VISA
RDEVice
FIND?
TIMeout

Click on a keyword to view the command details.

see Also

- [Referring to Traces Channels Windows and Meas Using SCPI](#)
 - [Example Programs](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SYSTem:COMMunicate:GPIB:PMETer[:ADDRess] <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 Keysight IO Libraries on the PNA.

Parameters

<num> GPIB address of the power meter. Choose any integer between 0 and 30.

Examples

```
SYST:COMM:GPIB:PMET 13  
system:communicate:gpiB:pmeter:address 14
```

Query Syntax SYSTem:COMMunicate:GPIB:PMETer[:ADDRess]?

Return Type Numeric

Default 13

SYSTem:COMMunicate:GPIB:RDEVice:CLOSe <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.

Examples [See an example program](#)

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

<bus> Bus ID number.

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.

<addr> GPIB Address of the device to be controlled

<timeout> The amount of time (in milliseconds) to wait for a response from the remote device after sending a command. A "timeout" error is displayed after this time has passed without a response.

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

<ID> Session identification number that was returned with the **OPEN?** command.

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.

Examples **SYST:COMM:GPIB:RDEV:RES**

Query Syntax Not Applicable

Default Not Applicable

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.

```
SYST:COMM:GPIB:RDEV:WBIN 1,#210SYST:PRES<EOL>
```

The <EOL> represents your linefeed character.

Query Syntax Not Applicable

Default Not Applicable

SYSTem:COMMunicate:GPIB:RDEVice:WBLock <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:WRIT**.

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.

Parameters None

Example SYST:CONF:LAN:HOSTname?

Return Type String

Default Not applicable

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
- **ANY** Any VISA resource string or a visa alias

<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.

For **ANY**, any VISA resource string or a visa alias.

Examples

```
SYST:COMM:PSEN gpib, "14"  
system:communicate:psensor usb, "Keysight  
Technologies,U2000A,MY12345678"  
syst:comm:psen lan, "mymeter.Keysight.com"  
syst:comm:psen any,  
"TCPIP0::mymeter.Keysight.com::5025::SOCKET"
```

Query Syntax SYSTem:COMMunicate:PSEnSor?

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).

Parameters

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:COMMunicate:VISA:RDEVice:FIND? <VISA regex> [, <ADDRess|ALIAS>]

(Read-only) Returns a comma separated list of either VISA address strings or aliases.

Parameters

<VISA regex> (String) VISA regular expressions are expressions defined by the user to find devices that have been set up on the VISA interface. The following are examples of VISA regular expressions:

Interface	Expression
GPIB	GPIB[0-9]*::?*INSTR
PXI	PXI?*INSTR
VXI	VXI?*INSTR
GPIB-VXI	GPIB-VXI?*INSTR
GPIB and GPIB-VXI	GPIB?*INSTR
All VXI	?*VXI[0-9]*::?*INSTR
ASRL	ASRL[0-9]*::?*INSTR
All	?*INSTR or ?*

Note that using "INSTR" in the VISA regular expression finds "instruments." To search all interfaces, use "?*".

<ADDRess|ALIAS> Optional. Determines whether addresses or aliases are returned.

Note: The list of aliases may have less—or more—entries than the list of addresses because not all addresses will have aliases, and one address can have more than one alias.

Examples

```
SYST:COMM:VISA:RDEV:FIND? "?*",ADDR
system:communicate:visa:rdevice:find? '*INSTR',alias
```

Return Type Variant

Default Addresses returned if no return-type specified

SYSTem:COMMunicate:VISA:RDEVice:TIMEout <ID>, <timeout>

(Read-Write) Sets or returns the timeout value (in milliseconds) for VISA pass-through commands for the specified VISA session ID.

Parameters

<ID> VISA session number that was returned with the **OPEN?** command.

<timeout> The amount of time (in milliseconds) to wait for a response from the remote device after sending a command. A "timeout" error is displayed after this time has passed without a response.

Examples

```
SYST:COMM:VISA:RDEV:TIM 1,6000
```

```
system:communicate:visa:rdevice:timeout 3,6000
```

Query Syntax SYSTem:COMMunicate:VISA:RDEVice:TIMEout? <ID>

Returns the timeout value for the specified session ID.

Return Type Numeric

Default 2000

Trigger Commands

Controls External Triggering on PNA-X and N522x models.

TRIGger:

AUXiliary

| **COUNt**

CHANnel:AUXiliary

| **DELay**

| **DURation**

| **[ENABle]**

| **HANDshake**

| **INTerval**

| **IPOLarity**

| **OPOLarity**

| **POSition**

| **TYPE**

DELay

PREFerence

| **AIGLobal**

READy:POLarity

[SEQuence]

| **LEVel**

| **ROUTE**

| **INPut**

| **READy**

| **SCOPE**

| **SOURce**

| **SLOPE**

| **TYPE**

Click on a keyword to view the command details.

[Blue](#) commands are superseded.

see Also

- **Example program** [Triggering the PNA](#)
 - [See other SCPI Triggering commands](#)
 - [Learn about External / Aux Triggering](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

TRIGger:AUXiliary:COUNT?

(Read-only) Returns the number of AUX trigger input / output connector pairs in the instrument.

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.
Choose from **1** (AUX TRIG 1) or **2** (**AUX TRIG 2**)
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?

Return Type Numeric

Default 1E-6

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.
Choose from **1 (AUX TRIG 1)** or **2 (AUX TRIG 2)**
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.
Choose from **1 (AUX TRIG 1)** or **2 (AUX TRIG 2)**
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.
Choose from **1** (AUX TRIG 1) or **2** (AUX TRIG 2)
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.
Choose from **1** (AUX TRIG 1) or **2** (AUX TRIG 2)
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 **TRIG:CHAN:AUX:IPOL POS**

trigger:channel2:aux2:ipolarity negative

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.
Choose from **1** (AUX TRIG 1) or **2** (AUX TRIG 2)
If unspecified, value is set to 1.

<char> Choose from:

- **POSitive** PNA sends positive going pulse.
- **NEGative** PNA sends negative going pulse.

Examples TRIG:CHAN:AUX:OPOL NEG

trigger:channel2:aux2:opolarity positive

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.

Choose from **1** (AUX TRIG 1) or **2** (AUX TRIG 2)

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.

Choose from **1** (AUX TRIG 1) or **2** (AUX TRIG 2)

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: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 3.

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

Default 0

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.
- **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**.

Examples `TRIG:PREF:AIGL 1`
`trigger:preference:aiglobal 0`

Query Syntax `TRIGger:PREFerence:AIGLobal?`

Return Type Boolean

Default 0

TRIGger:READy:POLarity <char>

(Read-Write) Specifies the polarity of Ready for Trigger output.

All existing Ready for Trigger outputs are configured simultaneously with this command.

Parameters

<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?

Return Type Character

Default LOW

TRIGger[:SEQuence]:ROUTE:INPut <char>

(Read-Write) Specifies the connector to use for the external trigger input.

Parameters

- <char> Choose from:
- SMB** – Meas Trig In BNC
 - DSTARB** – Backplane Trigger Lines (PXIe DSTARB)
 - STAR** – Backplane Trigger Lines (PXI STAR)
 - TRIG0** – Backplane Trigger Lines (PXI TRIG0)
 - TRIG1** – Backplane Trigger Lines (PXI TRIG1)
 - TRIG2** – Backplane Trigger Lines (PXI TRIG2)
 - TRIG3** – Backplane Trigger Lines (PXI TRIG3)
 - TRIG4** – Backplane Trigger Lines (PXI TRIG4)
 - TRIG5** – Backplane Trigger Lines (PXI TRIG5)
 - TRIG6** – Backplane Trigger Lines (PXI TRIG6)
 - TRIG7** – Backplane Trigger Lines (PXI TRIG7)

Examples TRIG:ROUTE:INP SMB

trigger:sequence:route:input smb

Query Syntax TRIGger[:SEQuence]:ROUTE:INPut?

Return Type Character

Default MAIN

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:SWEp: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

Default POSitive

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.

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[:SEQuence]:TYPE <char>

(Read-Write) Specifies the type of EXTERNAL trigger input detection used to listen for signals on the Meas Trig IN connectors. Edge triggers are most commonly used.

Parameters

<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 **TRIG:TYPE EDGE**
trigger:sequence:type level

Query Syntax TRIGger[:SEQuence]:TYPE?

Return Type Character

Default LEVel

Last modified:

12-Nov-2013	Fixed trig:type
23-Jan-2013	Removed <ch> argument from several commands
12-Apr-2012	Removed reference to old models-big syntax fix
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

```
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 Measurments: " & 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

This VBScript program does the following:

- Presets the analyzer, deleting the default trace
- 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

The following notes explain the basic structure of the SCPI tree on the analyzer:

- **SOURce:** and most **SENSe:** commands act on the **channel** that is specified in the command. Channel 1 is default if not specified.
- Most **DISPlay:** 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.

See [Traces, Channels, and Windows on the Analyzer](#)

How to run 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 analyzer. To do this, copy the following code into a text editor file such as Notepad and save it on the analyzer 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
```

```
'Preset the analyzer
```

```
'This command also deletes the default trace
```

```
scpi.execute "SYSTEM:FPReset"
```

```
'Create Measurements
```

```
scpi.execute "CALCulate1:PARAMeter:DEFine:EXT 'Meas1','S11'"
```

```
scpi.execute "CALCulate2:PARAMeter:DEFine:EXT 'Meas2','S21'"
```

```
' Turn on windows - creates if new
```

```
scpi.execute "DISPlay:WINDow1:STATE ON"
```

```
scpi.execute "DISPlay:WINDow2:STATE ON"
```

```
'Associate ("FEED") the measurement name('Meas1') to WINDow(1), and  
give the new TRACe a number(1).
```

```
scpi.execute "DISPlay:WINDow1:TRACe1:FEED 'Meas1'"
```

```
scpi.execute "DISPlay:WINDow2:TRACe2:FEED 'Meas2'"
```

```
'Change each channel's frequency range
```

```
scpi.execute "SENSe1:FREQUency:SPAN 1e9"
```

```
scpi.execute "SENSe2:FREQUency:SPAN 2e9"
```

```
'Select both measurements
```

```
scpi.execute "CALCulate1:PARAMeter:SELEct 'Meas1'"
```

```
scpi.execute "CALCulate2:PARAMeter:SELEct 'Meas2'"
```

```
'Turn marker 1 ON for each measurement
```

```
scpi.execute "CALCulate1:MARKer:STATE ON"
```

```
scpi.execute "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.
- **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

```
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
'
' _____
' 1 -----|      |----- 2 +
'      |  DUT  |
' 4 -----|_____|------ 3 -
```

```
' 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")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SDD21")
```

```
' 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("CALC:FSIM:BAL:PAR:BBAL:DEF SCD21")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SDC21")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SCC21")
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")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SDD11")
```

' 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("CALC:FSIM:BAL:PAR:BBAL:DEF SCD11")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SDC11")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SCC11")
```

```
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")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SDD12")
' 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:FSIM:BAL:PAR:BBAL:DEF SDC12")
scpi.Parse("DISP:WIND3:TRAC3:FEED ""sdc12""")
scpi.Parse("CALC:PAR:DEF:EXT ""scc12"",S11")
scpi.Parse("CALC:PAR:SEL ""scc12""")
scpi.Parse("CALC:FSIM:BAL:PAR:STATe ON")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SCC12")
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")
scpi.Parse("CALC:FSIM:BAL:PAR:BBAL:DEF SDD22")
' 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("CALC:FSIM:BAL:PAR:BBAL:DEF SCD22")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SDC22")
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("CALC:FSIM:BAL:PAR:BBAL:DEF SCC22")
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 = scpi.Parse("CALC:FSIM:BAL:PAR:" & device & ":DEF?")
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
```

Last Modified:

9-May-2011 Modified to make it work per TS

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:FPRReset")
' 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 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 thrus. Be sure to use the following sequence for each newly defined standard.

1. Programmatically select the standard number
2. Programmatically 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
360 !
370 CLEAR SCREEN
380 !
390
!!!!!!
400 ! Designate the kit selection to be used for
performing cal's
410 OUTPUT @Na;":sens:corr:ckit:count?"
420 ENTER @Na;Calkitnum
430 Calkitnum=Calkitnum+1
440 OUTPUT @Na;":sens:corr:coll:ckit
"&VAL$(Calkitnum)
```

```
450  !
460  ! Name this kit with your own name
470  OUTPUT @Na;":sens:corr:coll:ckit:name ""Special
2.4 mm Model 85056""
480  !
490  !
500  DISP "Defining kit std 1..."
510  ! Now set up standard #1
520  OUTPUT @Na;":sens:corr:coll:ckit:stan 1"
530  OUTPUT @Na;":sens:corr:coll:ckit:stan:type
SHORT"
540  Get_std
550  OUTPUT @Na;":sens:corr:coll:ckit:stan:char coax"
560  OUTPUT @Na;":sens:corr:coll:ckit:stan:label ""My
Short""
570  Get_label
580  !
590  DISP "Defining kit std 2..."
600  ! Now set up standard #2
610  OUTPUT @Na;":sens:corr:coll:ckit:stan 2"
620  OUTPUT @Na;":sens:corr:coll:ckit:stan:type OPEN"
630  Get_std
640  OUTPUT @Na;":sens:corr:coll:ckit:stan:char coax"
650  OUTPUT @Na;":sens:corr:coll:ckit:stan:label ""My
Open""
660  Get_label
670  !
680  DISP "Defining kit std 3..."
690  ! Now set up standard #3
700  OUTPUT @Na;":sens:corr:coll:ckit:stan 3"
710  OUTPUT @Na;":sens:corr:coll:ckit:stan:type LOAD"
720  Get_std
730  OUTPUT @Na;":sens:corr:coll:ckit:stan:char coax"
740  OUTPUT @Na;":sens:corr:coll:ckit:stan:label ""My
Fixed Load""
750  Get_label
```

```
760  !
770  DISP "Defining kit std 4..."
780  ! Now set up standard #4
790  OUTPUT @Na;":sens:corr:coll:ckit:stan 4"
800  OUTPUT @Na;":sens:corr:coll:ckit:stan:type THRU"
810  Get_std
820  OUTPUT @Na;":sens:corr:coll:ckit:stan:char coax"
830  OUTPUT @Na;":sens:corr:coll:ckit:stan:label ""My
Thru""
840  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"
1190 OUTPUT @Na;":sens:corr:coll:ckit:stan:type ARBI"
1200 Get_std
1210 OUTPUT @Na;":sens:corr:coll:ckit:stan:char coax"
1220 OUTPUT @Na;":sens:corr:coll:ckit:stan:TZR 15;"
1230 OUTPUT @Na;":sens:corr:coll:ckit:stan:TZI -9;"
1240 OUTPUT @Na;":sens:corr:coll:ckit:stan:label ""Z
Load""
1250 Get_label
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$
1350 !
1360 ! Define your new connectors
1370 OUTPUT @Na;":sens:corr:coll:ckit:conn:add ""PSC
2.4"",0HZ,999GHZ,50.0,MALE,COAX,0.0"
1380 OUTPUT @Na;":sens:corr:coll:ckit:conn:add ""PSC
```

```
2.4"" , 0HZ, 999GHZ, 50.0, FEMALE, COAX, 0.0"
1390 !
1400 ! Verify that the new connectors are installed
1410 OUTPUT @Na;":sens:corr:coll:ckit:conn:cat?"
1420 ENTER @Na;Conn$
1430 PRINT "Verify new connectors: ";Conn$
1440 DISP ""
1450 !
1460 DISP "Defining conn std 1..."
1470 ! Now set up standard #1
1480 OUTPUT @Na;":sens:corr:coll:ckit:stan 1"
1490 Verify_std
1500 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" , FEMALE, 1"
1510 Print_connector
1520 !
1530 DISP "Defining conn std 2..."
1540 ! Now set up standard #2
1550 OUTPUT @Na;":sens:corr:coll:ckit:stan 2"
1560 Verify_std
1570 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" , FEMALE, 1"
1580 Print_connector
1590 !
1600 DISP "Defining conn std 3..."
1610 ! Now set up standard #3
1620 OUTPUT @Na;":sens:corr:coll:ckit:stan 3"
1630 Verify_std
1640 OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" , FEMALE, 1"
1650 Print_connector
1660 !
1670 DISP "Defining conn std 4..."
1680 ! Now set up standard #4
1690 OUTPUT @Na;":sens:corr:coll:ckit:stan 4"
1700 Verify_std
```

```
1710  OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" ,FEMALE,1"
1720  OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" ,MALE,2"
1730  Print_connector
1740  !
1750  DISP "Defining conn std 5..."
1760  ! Now set up standard #5
1770  OUTPUT @Na;":sens:corr:coll:ckit:stan 5"
1780  OUTPUT @Na;":sens:corr:coll:ckit:stan:label
""Sliding Load""
1790  Verify_std
1800  OUTPUT @Na;":sens:corr:coll:ckit:conn:snam ""PSC
2.4"" ,MALE,1"
1810  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, 3rd 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, 3rd 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 !
```

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
- Keysight'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

```
'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 Keysight 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 Keysight'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:/VXIPNP/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 analyzer. 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 SICT 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
```

External Test Set Control using SCPI

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.](#)

```
' 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
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 device would typically be connected to the PNA through the System Controller GPIB port on the PNA rear-panel or alternatively be connected using a **USB/GPIB interface**. Uncomment the line in **Blue text** in the example to open a session for 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

```
option explicit
dim app
set app = CreateObject("AgilentPNA835x.Application")

dim p
set p = app.ScpiStringParser

' Open a new GPIB session on Bus:0 Device:14 Timeout: 100ms
p.Parse "SYST:COMM:GPIB:RDEV:OPEN 0,14,100"
' The following commented-out line shows opening the same session but
' for a USB/GPIB interface with VISA interface number GPIB4
'p.Parse "SYST:COMM:GPIB:RDEV:OPEN 4,14,100"
```

```
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
idn = p.Parse("SYST:COMM:GPIB:RDEV:READ? " & handleAsInt)
msgbox idn

' Close the GPIB session
p.Parse "SYST:COMM:GPIB:RDEV:CLOSE " & handleAsInt
```

Last modified:

30-May-2012 Added uncomment line and other edit (BH)

GPIB using Visual C++

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 the WINDOWS.H and DECL-32.H files. The standard Windows
 * header file, WINDOWS.H, contains definitions used by DECL-32.H and
 * DECL-32.H contains prototypes for the NI GPIB routines and constants.
 */
#include <windows.h>
#include "decl-32.h"

#define ERRMSG_SIZE 1024 // Maximum size of SCPI command string
#define ARRAY_SIZE 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];
char ErrorMnemonic[21][5] = {"EDVR", "ECIC", "ENOL", "EADR",
"EARG",
"ESAC", "EABO", "ENEB", "EDMA", "",
"EOIP", "ECAP", "EFSO", "", "EBUS",
"ESTB", "ESRQ", "", "", "", "ETAB"};

void GPIBWrite(char* SCPIcmd);
char *GPIBRead(void);
void GPIBCleanup(int Dev, char* ErrorMessage);

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
 */
void GPIBWrite(char* SCPIcmd)
{
int length;
char ErrorMessage[ERRMSGSIZE + 1];
length = strlen(SCPIcmd) ;

    ibwrt (pna, SCPIcmd, length);
    if (ibsta & ERR)
    {
        strcpy(ErrorMessage, "Unable to write this command to PNA:/n");
        strcat(ErrorMessage, SCPIcmd);

        GPIBCleanup(pna, ErrorMessage);
        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* ErrorMessage)  
{  
    printf("Error : %s/nibsta = 0x%x iberr = %d (%s)/n",  
        ErrorMessage, 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](#)

See Other SCPI Example Programs

```
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"" ")
scpi.Execute("sens:corr:coll:guid:ckit:port2 ""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.
'scpi.Parse "sens:corr:coll:guid:ckit:port1 'N4691-60004 ECal'"
'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:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 ECal 01234'"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 MyDskChar ECal
01234'"
'
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"" ")
scpi.Execute("sens:corr:coll:guid:conn:port3 ""APC 3.5 female"" ")
scpi.Execute("sens:corr:coll:guid:conn:port4 ""APC 3.5 female"" ")
MsgBox("Connectors defined for Ports 1 to 4")
' 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"" ")
scpi.Execute("sens:corr:coll:guid:ckit:port3 ""85052D"" ")
scpi.Execute("sens:corr:coll:guid:ckit:port4 ""85052D"" ")
' 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
'scpi.Execute("sens:corr:coll:guid:ckit:port1 ""N4431-60003 ECal"" ")
'scpi.Execute("sens:corr:coll:guid:ckit:port2 ""N4431-60003 ECal"" ")
'scpi.Execute("sens:corr:coll:guid:ckit:port3 ""N4431-60003 ECal"" ")
'scpi.Execute("sens:corr:coll:guid:ckit:port4 ""N4431-60003 ECal"" ")
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)
strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
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
' while 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
scpi.Execute("sens:corr:coll:guid:save")
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"
```

Last Modified:

16-Jul-2010 Fixed typo

26-Feb-2009 MX New topic

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](#)

See Other SCPI Example Programs

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"
```

```
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:ID '" & ecalModelNum & ","  
& ecalSerialNum & ""
```

```
MsgBox "ECal module to be characterized is: " &  
scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:ID?")
```

```
' 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
```

```
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:CNUM " &  
CStr(characterizationNumber)
```

```
' 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
```

```
'connTypeList = scpi.Parse("SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?")
```

```
'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
```

```
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:CONN:PORT2 '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.

```
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:DESC:USER 'John Doe, Acme  
Inc.'"
```

' Set user-specified description of the PNA being used.

```
scpi.Parse "SENS1:CORR:CKIT:ECAL:CHAR:DESC:VNA 'SN US12345678'"
```

' 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 32)

' 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
GPIB

' 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 Keysight 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
filled).
' 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.Parse "SENS1:CORR:CKIT:ECAL:CHAR:INIT"
Dim numSteps
numSteps = CLng( scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:STEP?") )
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.
MsgBox scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:DESC? " & i)
' 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
opcReply = scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:ACQ STAN" &
CStr(i) & ";*OPC?")
' 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.
opcReply = scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:SAVE;*OPC?")
'scpi.Execute "SENS1:CORR:CKIT:ECAL:CHAR:SAVE;*OPC"
'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"
opcReply = scpi.Parse("SENS1:CORR:CKIT:ECAL:CHAR:DMEM:SAVE '" &
characterizationName & "';*OPC?")
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.
scpi.Parse "SENS:CORR:CKIT:ECAL:EXP '" & pnaDiskMemCalKitName &
""
' 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".
scpi.Parse "SENS:CORR:CKIT:ECAL:DMEM:CLE '" &
pnaDiskMemCalKitName & ""
scpi.Parse "SENS:CORR:CKIT:ECAL:DMEM:IMP 'C:/Program
Files/Keysight/Network Analyzer/ECal User Characterizations/" &

```

```

pnaDiskMemCalKitName & ".euc'"
Dim moduleMemCalKitName
moduleMemCalKitName = GetCalKitName("User " &
CStr(characterizationNumber))
MsgBox "Information about the characterization from ECal module memory = "
& scpi.Parse("SENS:CORR:CKIT:ECAL:KNAM:INF? '" &
moduleMemCalKitName & "'")
MsgBox "Information about the characterization from PNA disk memory = " &
scpi.Parse("SENS:CORR:CKIT:ECAL:KNAM:INF? '" &
pnaDiskMemCalKitName & "'")
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 SICT-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
```

Last Modified:

18-Sep-2009 Updated with new User Char commands (9.0)

25-Nov-2008 New topic (8.33)

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>[, <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, 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 number (SST<n>).

When performing 2 port calibrations on 4 Port Network Analyzers, 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

--	--	--

Ports	S11A Port	S22A Port
1,2	1	2
1,3	1	3
1,4	1	4
2,3	2	3
2,4	2	4
3,4	3	4

The following example program shows one method of handling two port calcs 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 calcs are performed: 1-2 (insertable male to female), 2-3 (insertable female to male), and 3-4 (noninsertable using an characterized adapter).

```

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" )
PrintKitStandardInfo 1
PrintKitOlist 1
```

```
' -----
' CALIBRATE PORTS 1 and 2, insertable cal
' -----
```

```
wscript.echo
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 ")
```

```
' -----
' CALIBRATE PORTS 2 and 3, insertable cal
' -----
```

```
wscript.echo
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 ")
```

CALIBRATE PORTS 3 and 4, non-insertable cal

```
wscript.echo
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
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 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.](#)

```
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:CKIT:PORT1 """
```

```
Parser.Parse "SENS:CORR:COLL:GUID:CKIT:PORT2 '85054D'"
```

```
Parser.Parse "SENS:CORR:COLL:GUID:INIT"
```

Rem Query the number of steps

```
txtDat = Parser.Parse("SENS:CORR:COLL:GUID:STEP?")
```

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
```

```
    prompt = Parser.Parse("sens:corr:coll:guid:desc? 1")
```

```
MsgBox(prompt)
Parser.Parse ("sens:corr:coll:guid:acq STAN1")
ElseIf i = 2 then
prompt = Parser.Parse("sens:corr:coll:guid:desc? 2")
MsgBox(prompt)
Parser.Parse ("sens:corr:coll:guid:acq STAN2")
ElseIf i = 3 then
prompt = Parser.Parse("sens:corr:coll:guid:desc? 3")
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.](#)

```
' 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

```
connList = scpi.Execute("sens:corr:coll:guid:conn:cat?")
```

' 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
```

```
scpi.Execute "sens:corr:coll:guid:conn:port2 '" & selectedConn2 & """
```

' 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

```
'adapterNum = scpi.Execute("sens:corr:coll:guid:adap:cre? '" &  
selectedConn1 & "', '" & 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.
kitList = scpi.Execute("sens:corr:coll:guid:ckit:cat? '" & selectedConn1 &
"'"")
' 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.
kitList = scpi.Execute("sens:corr:coll:guid:ckit:cat? '" & selectedConn2 &
"'"")
' 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.
kitList = scpi.Execute("sens:corr:coll:guid:ckit:cat? '" & selectedConn1 &
"'" )
' 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'")
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
scpi.Execute("sens:corr:coll:guid:psen1:stat off")
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
' "sens:corr:coll:guid:init" command.
' Read ECAL information from ECal module #1 on the USB bus
' about the Keysight 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).
```

```
'scpi.Execute "sens:corr:pref:ecal:ori OFF"
```

```
'scpi.Execute "sens:corr:pref:ecal:pmap ECAL1,'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 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
```

```
scpi.Execute "sens:corr:coll:guid:init"
```

```
numSteps = scpi.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)
```

```
strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
```

```
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!"
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
```

Last Modified:

8-Oct-2010	Updated for Enhanced Power (BH)
14-May-2007	MX Updated for new CMET and TMET commands

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'"
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:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 ECal 01234'"
' When Disk Memory ECal user characterizations are used,
' specify both the User char and the serial number as follows:
'scpi.Parse "sens:corr:coll:guid:ckit:port2 'N4691-60004 MyDskChar ECal
01234'"
'
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"
numSteps = scpi.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)
strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
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!"

```

Last modified:

5-Apr-2011 edited for ECal options

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"" "
```

```
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"
```

```
' 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"  
numSteps = scpi.Execute("sens:corr:coll:guid:steps?")  
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 = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
```

```
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!"
```

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.](#)

```
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.SCPIStrParser
Parser.Parse "DISPlay:WINDow1:STATE ON"
Parser.Parse "CALCulate:PARAmeter:DEFine:EXT
'MyMeas',S12"
Parser.Parse "DISPlay:WINDow1:TRACe1:FEED 'MyMeas'"
' Initialize state
Set Chan = App.ActiveChannel
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'"
Parser.Parse "SENS:CORR:COLL:GUID:CKIT:PORT1 '85052C'"
Parser.Parse "SENS:CORR:COLL:GUID:CKIT:PORT2 '85052C'"
' 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
txtDat = Parser.Parse("SENS:CORR:COLL:GUID:STEP?")
' 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")
```

Last Modified:

9-Apr-2007 MX Updated for new CMethod command

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.

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.](#)

```
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.](#)

```
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")

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 Keysight 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 Keysight 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 PNAs. When run on select PNA-L models, a Delta Match Cal is required.

- See Delta Match Cal example program
- [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.](#)

```
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'"
scpi.Parse "SENS:CORR:COLL:GUID:CKIT:PORT2 '85052C'"
' Since the 85052C cal kit contains SOLT standards and also TRL
' standards, these next two lines set cal and thru method.
' Always send the init command before and after these two commands
scpi.Parse "SENS:CORR:COLL:GUID:INIT"
scpi.Parse "SENS:CORR:COLL:GUID:PATH:CMETHOD 1,2,"SOLT"
scpi.Parse "SENS:CORR:COLL:GUID:PATH:TMETHOD 1,2,"UNKN"
' To set up the cal as TRL, comment the previous 'CMET' line and uncomment
```

```

' this next line. The TMethod is set by default
'scpi.Parse "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
retStr = scpi.Parse("SENS:CORR:COLL:GUID:DMAT:APPL:PORT?")
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
' 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
prompt = scpi.Parse("SENS:CORR:COLL:GUID:DESC? " & CStr(i))
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:

10-Jan-2013 Added init command before CMET and TMET

14-May-2007 MX Updated for new CMET and TMET commands

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.](#)

See Other SCPI Example Programs

```
Dim na, vi, ret
```

```
Set na = CreateObject("AgilentPNA835x.Application")
```

```
Set vi = na.ScpStringParser
```

```
'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
```

Last Modified:

6-Oct-2008 MX New topic

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:FPRReset")
```

```
' 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:SElect '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

Last Modified:

22-Feb-2010 MX New topic

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

```
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

```
GPIB.Write "SYSTem:PRESet" 'Select the measurement
GPIB.Write "CALCulate:PARAMeter:SElect '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:RPOStion?"

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:SMOothing: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**

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
```

```
'This creates an S11 measurement in channel 1, window 1
```

```
scpi.Execute "SYST:PRreset"
```

```
' 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'"
```

```
'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"
numSteps = scpi.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)
strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
'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
scpi.Execute "SENS:CORR:COLL:SWE:CHAN:AOFF"

```

Last Modified:

1-Nov-2007 [New topic](#)

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
```

See Other SCPI Example Programs

Status Reporting using SCPI

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.

```
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 **MMEM:TRANsfer** command.

Transferring data FROM the PNA -- TO a remote PC:

```
30 !
40 !           Set up I/O paths
50 !
60 ! Network analyzer address
70 ASSIGN @Na TO 716
75 !
77 ! File to be stored on local computer
80 ! First time -- need to create the file.
90 ! After file name, number records set to 0 (ignored by WinOS)
95 ! Use "PURGE" command to delete if desired.
100 CREATE "mytestdata.s2p",0
110 ASSIGN @File TO "mytestdata.s2p"
120 !
122 !           TRANSFER the data (download)
123 !
125 ! Analyzer has file 'testdata.s2p' in default directory
130 OUTPUT @Na;":MMEM:TRAN? ""testdata.s2p""
135 !
137 ! 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 !
142 ! (2) Read the next character which is the number of digits in the file
size
150 ENTER @Na USING "#,A";Digit$
160 !
161 ! (3) Use the value of the number of digits to read back the file byte size
170 ! Create query string using this number of digits
180 Img$="#,"&Digit$&"A"
190 !
```

```
200 ! Byte$ holds the number of bytes in string format
210 ENTER @Na USING Img$;Byte$
220 !
225 ! (4) Read the file contents into a buffer and store the buffer contents
to a local file
230 ! Allocate a buffer for holding the data
240 ALLOCATE Dat$[VAL(Byte$)]
250 !
260 ! Set up a different image for filling the buffer
270 Img$=Byte$&"A"
280 !
290 ! Retrieve the actual file data
300 ENTER @Na USING Img$;Dat$
305 !
307 ! Now save the file locally.
310 OUTPUT @File;Dat$
320 END
```

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 !
122 !          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'
325 ! in default directory.
```

```
326 !  
327 ! A specific block transfer designator must follow the  
328 ! file name:  
329 ! '#' specifies a block transfer.  
330 ! '6' specifies 6 digits to follow.  
331 ! '026236' matches the buffer size allocated above  
332 ! not counting <NL><END> (new line and end of file).  
430 OUTPUT @Na;":MMEM:TRAN ""testupld.s2p"",#6026236",Dat$  
520 END
```

Last Modified:

26-Jul-2007 Added comments to example

Triggering the Analyzer using SCPI

To understand how to trigger the analyzer using SCPI, it is very important to understand the **trigger model**. Here is a very simple explanation. These three separate functions control triggering:

1. **Trigger:Source** - Where the trigger signals originate:
 - Internal Continuous
 - Internal Manual (Single)
 - External - a trigger source that is connected to the 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 uses the **default** Trigger Source = Internal to send a stream of trigger signals.
 - The channel is configured to ACCEPT only a single trigger signal, then HOLD (**Sense<ch>:Sweep:Mode SINGLE**). This is the ONLY required command.
 - 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 analyzer. To do this, copy the following code into a text editor file such as Notepad and save it on the analyzer 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.

```
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:FPRreset")
```

```

' 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
'scpi.Execute ("CONT:SIGN BNC1,TILHIGH")

'Setup Trigger Scope
'WHAT gets triggered
'Pick one using comments
'Set Channel triggering
'scpi.Execute ("TRIG:SCOPE CURRent")
'Set Global triggering (Default)
```

```
scpi.Execute ("TRIG:SCOPE ALL")
```

'Set Channel Settings

'The channels respond to UNLIMITED trigger signals (Default)

```
scpi.Execute ("SENS1:SWE:MODE CONTInuous")
```

```
scpi.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

```
'scpi.Execute ("SENS1:SWE:TRIG:POINT ON")
```

```
'scpi.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:

8-Mar-2013 Slight mods

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.](#)

```
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

```
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 = " & CDbI(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 = "
& CDbI(segData(j))

```

Next

WScript.Echo segInfStr

Last Modified:

29-Apr-2009 MX New topic

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 Keysight 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) 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
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)

' 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 :

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:

- **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

- **<WSP>** - Whitespace characters, such as <tab> and <space>, are generally ignored. There are two important exceptions:
 - Whitespace inside a keyword, such as :CALC ULATE, 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.

Form	Examples
Event commands - cause an action to occur inside the analyzer.	:INITIATE:IMMEDIATE
Query commands - query only; there is no associated analyzer state to set.	:SYSTem:ERRor?
Command and query - set or query an analyzer setting. The query form appends a question mark (?) to the set form	:FORMat:DATA ! Command :FORMat:DATA? ! Query

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.

SOUR - Short form

SOURce - Long form

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:

```
"HCOPY:IMMEDIATE"
```

```
"HCOPY"
```

The syntax in this Help system looks like this:

```
HCOPY[:IMMEDIATE]
```

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:

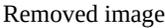
```
SENS:FREQ:START? MIN
```

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 

Configure for GPIB, SCPI, and SICL

The following settings are used to configure the analyzer for remote control using SCPI commands.

How to Configure for SICL / GPIB Operation

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

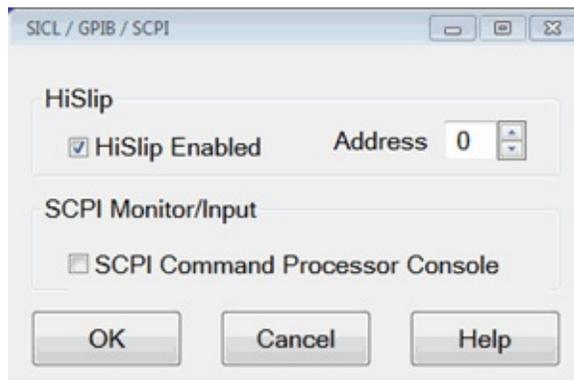
1. Press **SYSTEM**
2. then **[Configure]**
3. then **[SICL/GPIB]**

Using Menus

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **SICL/GPIB/SCPI**

◀ Programming Commands ▶

SICL / GPIB dialog box help



HiSLIP

HiSLIP is available ONLY when the soft front panel has already been launched. When checked, the PXI VNA will also advertise itself using port TCPIP port 4880. You must configure your firewall to enable communication through this port. You can connect to the module and send SCPI commands by using the VISA address: **TCPIP0::::hislip0::INSTR.** ("hislip" is case-sensitive) For example, if your PXI controller is named: myPXI, then the TCPIP connection

string would be:

```
TCPIP0::myPXI::hislip0::INSTR
```

When configuring an embedded controller, or when configuring an external computer that is connected using a PXIe cable, then replace <PXI controller name> with "Localhost". For example:

```
TCPIP0::Localhost::hislip0::INSTR
```

SCPI Monitor / Input

GPIB Command Processor Console Launches a window that is used to send single SCPI/GPIB commands from the analyzer 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 analyzer application. Press **Control+Tab** to bring the console window to the top.

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.

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:BOReR](#) 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

**+1.2300000000E+008; +7.8900000000E+008<NL><^END> 'Analyzer
Response**

Clearing the Output Queue

After receiving a query, the analyzer places the response message in its 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

```
#210ABCDE+WXYZ<nl><end>
```

Where:

- always sent before definite block data

2 - specifies that the byte count is two digits (2)

10 - specifies the number of data bytes that will follow, not counting <NL>
<END>

ABCDE+WXYZ - 10 digits of data

<NL><END> - always sent at the end of block data

Transferring Measurement Data

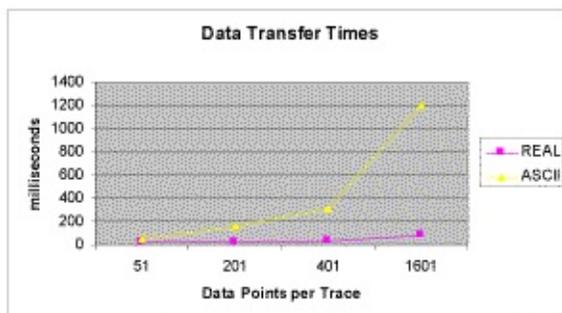
Measurement data is blocks of numbers that result from an analyzer measurement. Measurement data is available from various processing arrays within the analyzer. For more information on the analyzer's data processing flow, see [Accessing Data Map](#). Regardless of which measurement array is read, transferring 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:

- | | |
|-------------|------------------------------|
| 6-Feb-2013 | Changed block data example |
| 13-Jul-2012 | Fixed typo |
| 28-Jul-2009 | Added three zeros to example |
| 26-Jul-2007 | Added link to example |

Synchronizing the VNA and Controller

Synchronizing the VNA (Vector Network Analyzer) and Controller means to keep VNA and the controller working at approximately the same pace. In this topic:

- [The Problem and the Solution](#)
- [VNA Queues](#)
- [Synchronization Methods](#)
- [When To Synchronize the Analyzer and Controller](#)
 - [Completion of a Measurement](#)
 - [Measurements with External Trigger](#)
 - [Averaged Measurements](#)
 - [During Calibration Acquire](#)

See Also

- [Synchronize an External PSG Source](#)
- [Triggering the VNA using SCPI](#)

The Problem

The controller sends commands to the VNA as fast as the bus will allow. The VNA stores these commands in the VNA **Input queue**. However, the VNA executes those commands at a slower rate than they are accepted. If left unchecked, the VNA 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 VNA. The controller will not send more commands until a response is received. It will wait for a response from the VNA for the amount of time set by the Timeout setting. If the VNA 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 VNA "synched" is to send query commands often. This stops the controller from sending more commands until the VNA executes and responds to the query. This limits the number of commands that are waiting in the VNA 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, the VNA 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** (also known as Asynchronous) commands take longer to execute. Therefore, they allow the VNA 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 VNA is executing an overlapped command when a *OPC? is received, it will wait until the overlapped command is complete before processing new commands.

Note: The analyzer has two overlapped commands:

- **INITiate:IMMediate**
- **SENSe:SWEEp:MODE GROUPS** (when INIT:CONT is ON)

Several calibration commands have an optional ASYNchronous argument which allows them to behave like overlapped commands. [Learn more.](#)

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

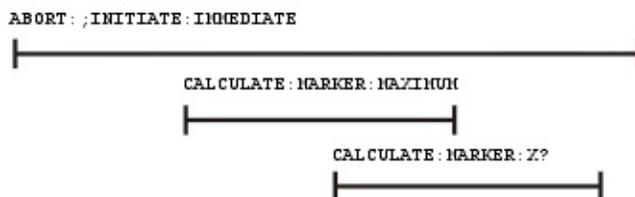
The *WAI command:

- **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

```
"ABORT;:INITIATE:IMMEDIATE" 'Restart the measurement.  
"CALCULATE:MARKER:SEARCH:MAXIMUM" 'Search for max  
amplitude.  
"CALCULATE:MARKER:X?" 'Which frequency?
```

The following time line shows how the processing times of the three commands relate to each other:

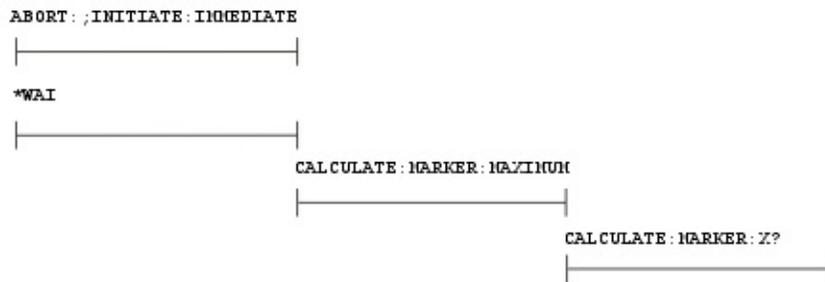


INITIATE:IMMEDIATE is an overlapped command. It allows the immediate processing of the sequential command, CALCULATE:MARKER:SEARCH:MAXIMUM. However, the INITIATE:IMMEDIATE is not considered complete until the measurement is complete. Therefore, the marker searches for maximum amplitude before the measurement completes. **The CALCULATE:MARKER:X? query could return an inaccurate value.**

To solve the problem, insert a *WAI command.

```
"ABORT;:INITIATE:IMMEDIATE" 'Restart the measurement.  
"*WAI" 'Wait until complete.  
"CALCULATE:MARKER:MAXIMUM" 'Search for max amplitude.  
"CALCULATE:MARKER:X?" 'Which frequency
```

The time line now looks like this:



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.

```
"ABORT;:INITIATE:IMMEDIATE"! Restart the measurement  
"*OPC?" 'Wait until complete  
Meas_done = GPIB.Read 'Read output queue, throw away result  
"CALCULATE:MARKER:MAX" 'Search for max amplitude
```

```
"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](#)
- [Measurements with External Trigger](#)
- [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 VNA 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.

During Calibration 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 VNA 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](#).

Note: Do NOT issue the `*OPC?` command when using the `ASYN` argument. If your program is using the `ScpiStringParser`, then you can ONLY use `*OPC?` to detect when the OPC bit is set, so do NOT use the `ASYN` argument with the calibration commands when using that parser.

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 VNA takes a several seconds to respond to additional commands, such as when the processor is calculating error terms.

The following commands have this argument:

Command	Example
<code>SENS:CORR:COLL:GUID:ACQUIRE</code> (Guided Cal)	Guided 2-Port or 4-Port Cal
<code>SENS:CORR:COLL:ACQUIRE</code> (Unguided Cal)	Perform Unguided ECAL
<code>SOUR:POW:CORR:COLL:ACQUIRE</code> (Source Power Cal)	Perform a Source and Receiver Power Cal (shows polling loop)

In addition, the `SENS:CORR:COLL:GUIDED:INITIALIZE` command has this optional argument for long calibration initialization, such as a `CalAll` calibration.

Calibrating the Analyzer Using SCPI

There are several ways to calibrate the analyzer 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](#)
- [Creating Cal Sets](#)
- [Applying Cal Sets and Cal Types](#)
- [Uploading Error Terms](#)
- [Unguided Cals and Calibration Classes](#)

Note: ALWAYS send ALL measurement setup commands BEFORE initializing a remote calibration.

See Also

[Synchronizing the Analyzer and Controller \(During a calibration\)](#)

[See SCPI Calibration Examples](#)

Guided Calibrations

Guided versus Unguided is the style of calibration that is selected on the first page of the [Calibration Wizard](#). A remote 'guided' cal does not present the cal wizard, but prompts for specific standards to be connected. In a remote 'Unguided', the steps must be 'hard-coded'.

- To perform a **Guided Calibration**, use ONLY [Sens:Corr:Coll:Guided](#) commands.
- 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 calcs.
- 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**

Note: To perform an **Unguided Calibration**, use ONLY the **Sens:Corr** commands (NOT Guided).

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 analyzer perform an Unknown Thru measurement of the ECal module Thru state **IF** the analyzer model being used has **1 reference receiver per port**. If your analyzer 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.
- You do NOT have to send the ECal module state 'switch' commands. The ECal algorithm switches ECal states automatically.

- All of these ECal choices are listed in the [Programming Command Finder](#) function in this Help file.

See [Using ECal](#) to learn about all of the ECal features.

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

'Perform 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 analyzer 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 analyzer will reapply the Full 2-Port Cal Type.

See the SCPI and COM commands for [Cal Sets](#) and [Cal Types](#).

Uploading Error Terms

Note: There was a method described here for WinCal 3.x that involved a **preference setting**. That method is no longer supported.

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>  
SENS:CORR:CSET:SAVE
```

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"

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 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 (SST<n>).

See an example: Perform an Unguided 2-port Cal on a 4-port analyzer.

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](#)

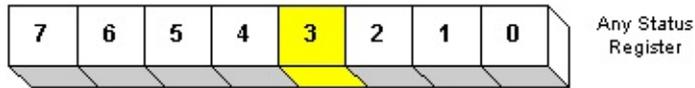
Other Topics about GPIB Concepts

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.



Each PNA Status Register is actually comprised of the following registers. = 4

```
&& typeof(BSPSPopupOnMouseOver) == 'function')
```

```
BSPSPopupOnMouseOver(event);" class="BSSCPopup"
```

```
onclick="BSSCPopup('./GP-IB_Command_Finder/Status.htm');return
```

```
false;">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 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.

8-bit register

Bit	0	1	2	3	4	5	6	7
Weight	1	2	4	8	16	32	64	128

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)	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

Referring to Traces, Measurements, Channels, and Windows

Using SCPI

Sometimes in a SCPI program you may need to refer to traces that you have not created. This can be a bit confusing in the PNA. Here are the THREE ways to refer to a specific measurement trace.

Note: The terms "Trace" and "Measurement" effectively mean the same thing in this discussion.

1. The **Measurement Name** is picked by you when you first create a trace using the `CALCulate<cnum>:PARAmeter[:DEFine]:EXTended <Mname>, <param>` command. The measurement name is only used by SCPI.
2. The **Trace Number** is also picked by you when 'feeding' a newly-created measurement name to a window number using `DISP:WINDow<wmun>:TRACe<tnum>:FEED`. The trace number is used ONLY by SCPI and is mainly used to refer to traces in the DISPLAY node. This is NOT the number that appears as **Tr#** on the screen. While you can assign any Trace number you want, when a measurement is created from the GUI, the PNA assigns numbers to the traces sequentially, starting with one in each window. Therefore, when there is more than one window, these numbers are not unique.
3. The **Tr#** that appears on the PNA screen is the third and most visible way to refer to a trace. Since we already have a "Trace Number", we call this the **Measurement Number** in the PNA Help file. This number is issued sequentially by the PNA regardless of channel and window. It is therefore unique among all traces. Use `CALC<ch>:PAR:MNUM?` just after the trace is created to read the measurement number.

The concept of the **Active measurement** versus **Selected Measurement** is also a bit confusing. As seen on the screen, the Active measurement has the highlighted **Tr#**. While there can only be ONE active measurement, every channel has a selected measurement. The target measurement must first be selected before most CALC node settings can be made. There are two ways to select a measurement for each channel:

1. Use **CALC<ch>:PAR:SEL <measName>** which requires the channel number and measurement name.
2. Use **CALC<ch>:PAR:MNUM <measNum>** which requires the channel and measurement (**Tr**) number.

Here are other relevant commands for referring to traces, measurements, channels, and windows:

- **CALC<cnum>:PAR:CATalog:EXTended?** - Catalog the Measurement Names for the specified channel.
- **CALC<cnum>:PAR:TNUMBER?** - Returns the Trace Number of the selected trace.
- **CALC<cnum>:PAR:WNUMBER?** - Returns the window number of the selected trace.
- **SYSTEM:ACTIVE:CHANNEL?** - Returns the number of the active channel. The active channel is the channel number that contains the active measurement.
- **SYSTEM:ACTIVE:MEAS?** - Returns the name of the active measurement. As seen on the screen, the Active measurement has the highlighted Tr#.
- **SYSTEM:CHANNELS:CATalog?** - Returns the channel numbers currently in use.
- **SYSTEM:WINDOWS:CATalog?** - Returns the window numbers that are currently being used.
- **SYSTEM:MEAS:CATalog? [chan]** - Returns ALL measurement numbers, or optionally measurement numbers from a specified channel.
- **SYSTEM:MEAS<n>:NAME?** - Returns the name of the specified measurement (Tr#) number.
- **SYSTEM:MEAS<n>:TRACE?** - Returns the trace number of the specified measurement number.
- **SYSTEM:MEAS<n>:WINDOW?** - Returns the window number of the specified measurement number.

Last modified:

26-Feb-2013 Added Active Meas

7-Aug-2012 New topic

Configure for SCPI LAN using SICL / VISA

- [PNA Supported Interfaces](#)
- [Keysight 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 Keysight 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.](#)

Keysight I/O Libraries

The Keysight I/O libraries includes the drivers to allow you to communicate with Keysight test instruments. Every PNA is shipped with the Keysight I/O libraries installed. We recommend you do NOT upgrade the Keysight I/O libraries on the PNA as unexpected results may occur. If you choose to upgrade the Keysight I/O libraries on the PNA, do NOT change the default folder path in the InstallShield Wizard.

To communicate with the PNA, the Keysight I/O libraries must also be installed on your external controller. To purchase the Keysight I/O libraries, or download a free upgrade, go to www.Keysight.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 Keysight 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 Keysight 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 Keysight 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:

```
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, Keysight 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.

VISA Interface Name	SICL Interface Name	Dialog box title	Description
GPIB0	gpib0	GPIB Using NI-488.2	PNA Rear-panel GPIB connector. This GPIB interface can be used to control the PNA OR for the PNA to control external equipment. IT CAN NOT DO BOTH IN THE SAME PROGRAM. Learn more about pass-through options.
GPIB1	hpib7	Internal Instrument Configuration	Internal interface for programs running on the PNA to control itself.
GPIB4	inst0	Internal Instrument Configuration	Used for LXI compliance. Do NOT delete this

			interface.
--	--	--	-------------------

Configure the External Controller

Please refer to the Keysight 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

Keysight VEE Pro RunTime Installed

Beginning in Dec. 2005, Keysight VEE Pro RunTime is installed on new PNAs. This means that programs written with Keysight VEE (.vxe files) can be run directly on the PNA.

PNAs **without** Keysight VEE installed can go to the [Keysight VEE website](#) and download Keysight **VEE Pro 6.2** RunTime to the PNA and begin to run VEE programs directly on the PNA. This version does not require Keysight I/O Libraries suite 14. **Do NOT upgrade to Keysight I/O libraries suite 14 on the PNA.**

With Keysight 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 Keysight VEE, see www.Keysight.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.

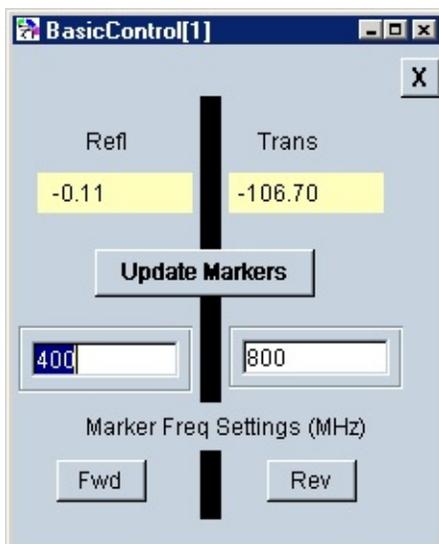
If this Help file is on a PNA and **VEE Pro RunTime is installed**, then:

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.

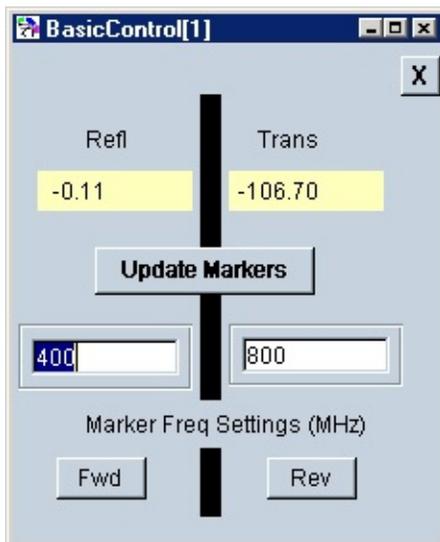
If this Help file is on a PNA and **VEE Pro RunTime is installed**:

- **Run the .vxe example**
- Then click **Open** on the following dialog box to run the program.

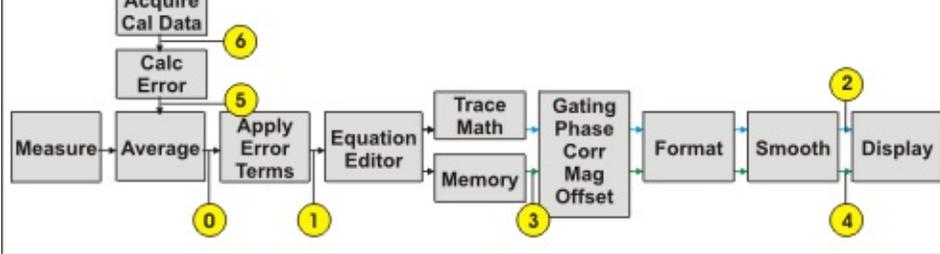
Or to modify the example program using VEE, **save the VEE BasicControl.vee**

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.



Click a box or circle to view details:

See [8510 data processing mode](#).

See larger [Data Processing map](#).

Measurement - Receivers gather complex trace data which is ratioed if required by the parameter, such as S11 or A/B. Otherwise it is raw receiver data, such as A or B. See [Measurement Parameters](#).

Averaging - If turned ON, data is averaged with specified number of measurement traces. See [Averaging](#).

Data Access Point 0 - Get or Put **RAW MEASUREMENT data** using:

SCPI - Write data using [Calc:Data SDATA](#) - Read data from **Data Access Point 1**

COM - getData and putDataComplex - naRawData (0)

Formatting Note:

COM - getData allows you to request data from locations 0 to 5 in a format other than the displayed format. SMOOTHED data is only attainable from locations 2 & 4, and only when you request data in the same format as the displayed format.

SCPI - you can only request data in the displayed format.

Acquired Cal Data - Calibration standards are measured. When the calibration is complete, complex data is stored in a Cal Set.

Data Access Point 6 - Get or Put **RAW CAL ACQUISITION data** using:

SCPI - None

COM - getStandardComplex and putStandardComplex

New PXI Programming Commands

The following are new programming commands for PXIe release A.03.00. [See What's New](#)

CALCulate commands	
Resets the currently-stored data points	CALC:HOLD:CLEAr
Sets the type of trace hold to perform	CALC:HOLD:TYPE
Returns the number of segments used in a limit test	CALC:LIMit:SEGMENT:COUNT?
Sets and reads the scope of Coupled Markers	CALC:MARK:COUP:METHOD

Ground Loop De-embedding/Embedding commands	
De-embedding	
Sets and returns the Capacitance value	CALC:FSIM:GLOop:DEEMbed:C
Sets and returns the Inductance value	CALC:FSIM:GLOop:DEEMbed:L
Sets and returns the Resistance value	CALC:FSIM:GLOop:DEEMbed:R
Turns ON or OFF De-embedding	CALC:FSIM:GLOop:DEEMbed:STATe
Specifies the circuit model type	CALC:FSIM:GLOop:DEEMbed:TYPE
Specifies the filename of the s1p file to load	CALC:FSIM:GLOop:DEEMbed:USER
Embedding	
Sets and returns the Capacitance value	CALC:FSIM:GLOop:EMBed:C
Sets and returns the Inductance value	CALC:FSIM:GLOop:EMBed:L
Sets and returns the Resistance value	CALC:FSIM:GLOop:EMBed:R
Turns ON or OFF Embedding	CALC:FSIM:GLOop:EMBed:STATe
Specifies the circuit model type	CALC:FSIM:GLOop:EMBed:TYPE
Specifies the filename of the s1p file to load	CALC:FSIM:GLOop:EMBed:USER

CSET commands

Returns a list of error term names for the given Cal Set	CSET:ETERm:CATalog?
Sets and returns the error term data	CSET:ETERm:DATA

DISPlay commands	
Sets and returns the X-axis position of the Limit Line Pass/Fail indicator	DISP:WIND:ANN:LIM:XPOStion
Sets and returns the Y-axis position of the Limit Line Pass/Fail indicator	DISP:WIND:ANN:LIM:YPOStion
Display marker symbols above trace or not	DISP:WIND:ANN:MARK:SYMB:ABOVe[STATe]

SENSe commands	
Returns a list of all name-value pairs in cal set	SENS:CORR:CSET:ITEM:CATalog?
Add or change a name-value pair in the Cal Set	SENS:CORR:CSET:ITEM[:DATA]
Sets and returns the ECAL over range state	SENS:CORR:PREF:ECAL:OVERrange[:STATe]
Computes the error correction terms, turns Correction ON, and saves the calibration to an existing, specified Cal Set	SENS:CORR:COLL:GUID:ETER:COMPUte
Sets the frequency step size across the selected frequency range	SENS:SWE:STEP

SYSTem commands	
Returns the maximum possible number of channels	SYSTem: CAPability:CHAN:MAX[:COUNt]?
Returns whether or not the analyzer has FOM Opt. 080 installed	SYSTem: CAPability:FOM:EXISTs?
Returns whether or not there is a receiver attenuator on the	SYSTem: CAPability:HARDware:ATTenuator:RECeiver:EXISTs?

specified port	
Returns the maximum amount of receiver attenuation on the specified port	SYSTem: CAPability:HARDware:ATTenuator:RECeiver:MAXimum?
Returns the step size of the receiver attenuator on the specified port	SYSTem: CAPability:HARDware:ATTenuator:RECeiver:STEP[:SIZE]?
Returns the maximum amount of source attenuation on the specified port	SYSTem: CAPability:HARDware:ATTenuator:SOURce:MAXimum?
Returns the step size of the source attenuator on the specified port	SYSTem: CAPability:HARDware:ATTenuator:SOURce:STEP[:SIZE]?
Returns whether or not the specified port number has a reference bypass switch	SYSTem: CAPability:HARDware:RBSWitch:EXISTs?
Returns the maximum specified frequency of the analyzer	SYSTem: CAPability:PRESet:FREQuency:MAXimum?
Returns the minimum specified frequency of the analyzer	SYSTem: CAPability:PRESet:FREQuency:MINimum?
Returns the maximum number of windows	SYSTem: CAPability:WINDows:MAXimum
Returns the maximum number of traces per window	SYSTem: CAPability:WINDows:TRACes:MAXimum?
Returns the maximum leveled source power setting	SYSTem:CAPability:ALC:POWER:MAXimum[:LEVel]?
Returns the minimum leveled source power setting	SYSTem:CAPability:ALC:POWER:MINimum[:LEVel]?
Returns a list of names of the internal DC receivers	SYSTem:CAPability:HARDware:DC:RECeiver:INTernal:CATalog?
Returns the number of internal DC receivers in the analyzer	SYSTem:CAPability:HARDware:DC:RECeiver:INTernal:COUNt?
Returns a list of names of the internal DC sources	SYSTem:CAPability:HARDware:DC:SOURce:INTernal:CATalog?
Returns the number of internal DC sources in the analyzer	SYSTem:CAPability:HARDware:DC:SOURce:INTernal:COUNt?
Returns a list of test port names	SYSTem:CAPability:HARDware:PORTs:CATalog?
Returns the number of test ports	SYSTem:CAPability:HARDware:PORTs:COUNt?

Returns a list of internal test port names	<code>SYSTem:CAPability:HARDware:PORTs:INTernal:CATalog?</code>
Returns the number of internal test ports	<code>SYSTem:CAPability:HARDware:PORTs:INTernal:COUNT?</code>
Returns the port number	<code>SYSTem:CAPability:HARDware:PORTs:PNUMber?</code>
Returns a list of source port names	<code>SYSTem:CAPability:HARDware:PORTs:SOURce:CATalog?</code>
Returns the number of source ports	<code>SYSTem:CAPability:HARDware:PORTs:SOURce:COUNT?</code>
Returns a list of internal source port names	<code>SYSTem:CAPability:HARDware:PORTs:SOURce:INTernal:CATalog?</code>
Returns the number of internal source ports	<code>SYSTem:CAPability:HARDware:PORTs:SOURce:INTernal:COUNT?</code>
Returns the number of receivers in the analyzer	<code>SYSTem:CAPability:HARDware:RECeiver:INTernal:COUNT?</code>
Returns the number of sources in the analyzer	<code>SYSTem:CAPability:HARDware:SOURCe:COUNT?</code>
Returns the list of supported Noise Bandwidth values when using a noise receiver	<code>SYSTem:CAPability:NBW:NOISe:CATalog?</code>
Returns the list of supported Noise Bandwidth values when using the NA receiver for noise measurements	<code>SYSTem:CAPability:NBW:STD:CATalog?</code>
Returns the list of supported Resolution BW values for the IMSpectrum channel	<code>SYSTem:CAPability:RBW:IMS:CATalog?</code>
Returns a comma separated list of either VISA address strings or aliases	<code>SYSTem:COMMunicate:VISA:RDEVice:FIND?</code>
Sets or returns the timeout value for VISA pass-through commands	<code>SYSTem: COMMunicate:VISA:RDEVice:TIMEout?</code>
Returns the system date	<code>SYSTem:DATE?</code>
Returns front panel history	<code>SYSTem:HISTory:FPANel?</code>
Returns SCPI history	<code>SYSTem:HISTory:SCPI?</code>
Set and return whether the keys	

are displayed or not	SYSTem:PREFerences:ITEM:KEYS
Set and return whether the Coupled Markers setting controls the ON OFF state of coupled markers	SYSTem:PREFerences:ITEM:MCControl
Set and return whether Coupled Markers is set to Channel or All after Preset	SYSTem:PREFerences:ITEM:MCMethod
Set and return whether Coupled Markers is set to ON or OFF after Preset	SYSTem:PREFerences:ITEM:MCReset
Set and return whether to draw limits lines in Red	SYSTem:PREFerences:ITEM:REDLimits
Set and return whether to treat marker 10 as a reference marker	SYSTem:PREFerences:ITEM:REFMarker
Returns the system time	SYSTem:TIME?

The following are new programming commands for PXIe release A.02.00. [See What's New](#)

PXIe Speed Enhancement commands	
Set logic of the index line	CONT:HAND:EXT:INDEX:LOG
Trigger multiple channels	SYST:CHAN:SING:COMB
Shared Memory Data Transfer commands	SYST:DATA:MEM
PXI Capability commands	SYST:CAP:HARD:MODule

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:

```
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

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 Analyzer 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](#)

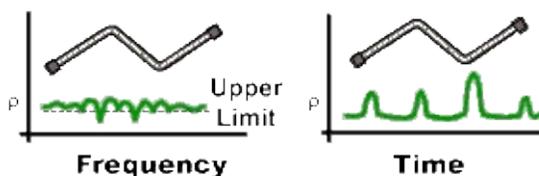
Note: Time Domain measurements are only available on analyzers with Option 010. See [Configurations](#)

See the updated **App Note:** [Time Domain Analysis Using a Network Analyzer](#).

Overview

In normal operation, the analyzer 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.



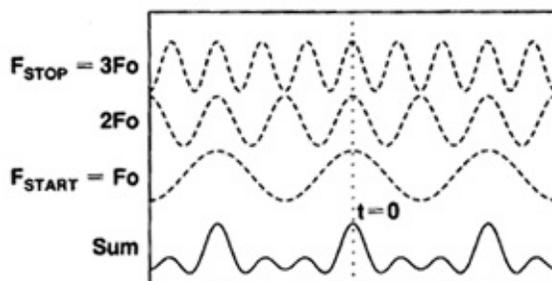
- The frequency domain S11 measurement shows reflections caused by mismatches in the cable. It is impossible to determine where the mismatches physically occur in the cable.
- The time domain response shows both the location and the magnitude of each mismatch. The responses indicate that the second cable 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 Analyzer 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.

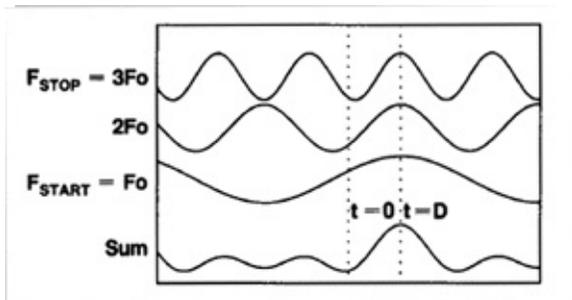
The analyzer does not launch an actual incident impulse or step. Instead, a Fourier Transform algorithm is used to calculate time information from the frequency measurements. The following shows how this occurs.

A single frequency in the time domain appears as a sine wave. In the following graphic, as we add the fundamental frequency (F_0), the first harmonic ($2F_0$), and then the second harmonic ($3F_0$), 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 analyzer sends discrete frequencies to the test device, it is in effect, sending individual spectral pieces of a pulse separately to stimulate the test device.

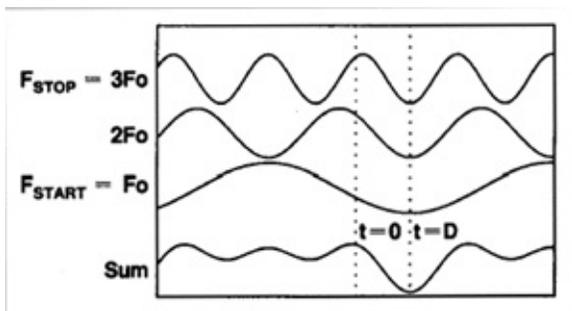


During an S11 reflection measurement, these incident signals reflect from the test device and are measured at the A receiver. This is when the time domain transform calculations are used to add the separate spectral pieces together.

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 analyzer 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.



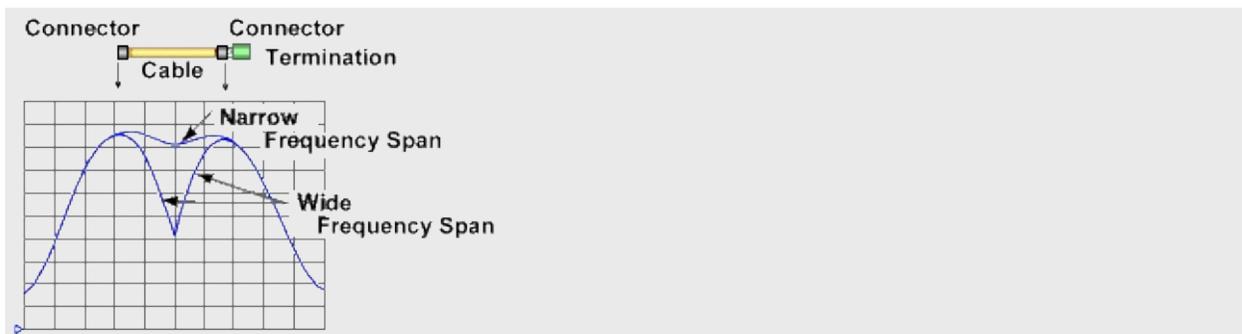
Triple travel is a term used to describe the reflected signal off the output, reflected again off the input, then finally reappearing at the output. This is best seen in a time domain S21 measurement.

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.

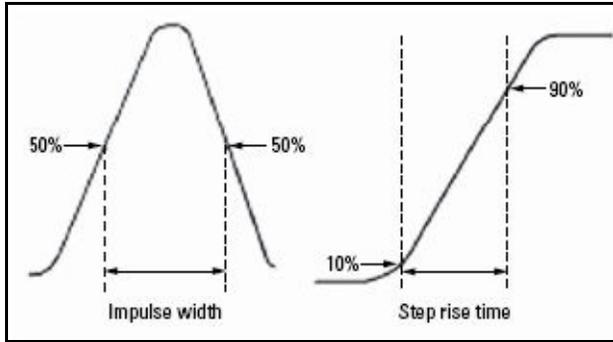
Note: Adjusting the **transform time settings** improves **display** resolution, but not measurement resolution.

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.

Window	Low-pass step (10% to 90%)	Low-pass impulse (50%)	Bandpass impulse
Minimum	0.45 / f span	0.60 / f span	1.20 / f span
Normal	0.99 / f span	0.98 / f span	1.95 / f span
Maximum	1.48 / f span	1.39 / f span	2.77 / f span

For example, using a 10 GHz wide frequency span and a normal window in Bandpass impulse mode, response resolution (in time) equals:

- Time Res = 1.95 / frequency span
- Time Res = 1.95 / 10 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 ps x c x Vf
- Distance Res = 195 ps x (2.997925 E8 m/s) x .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

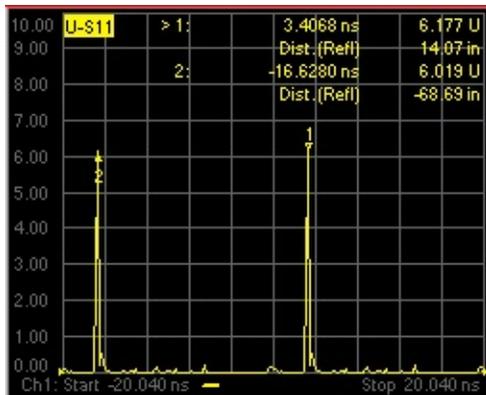
Notes:

- 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 Analyzer Measures in the Time Domain**). Alias responses occur at time intervals that are equal to $1/\text{frequency step size}$.

The analyzer 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 analyzer using the following formula for **TDR** (reflection) measurements:

$$\text{Range (meters)} = (1/\Delta f) \times V_f \times c$$

Where:

- Δf = 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 ($V_f = 0.66$)

- Range = $(1 / (2.5E9 / 400)) \times 2.997925 \text{ E8 m/s} \times 0.66$
- Range = $6.25E6 \times 2.997925 \text{ E8 m/s} \times 0.66$
- 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

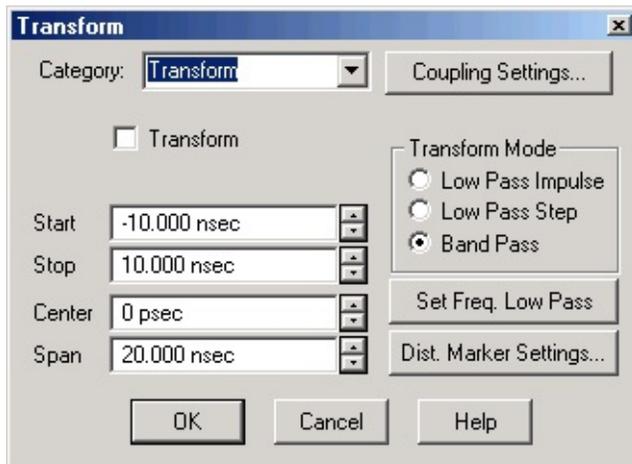
1. Press **ANALYSIS**
2. then **[Transform]**
3. then **[More]**
4. then **[Transform Tool]**

Using Menus

1. Click **Marker/Analysis**
2. then **Transform**

◀ Programming Commands ▶

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 analyzer 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 analyzer screen.

Center Sets the transform center time that is displayed in the center of the analyzer 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.

Mode	Benefit - application	Limitation
Low pass Impulse	Highest resolution. Most useful for seeing small responses in devices that pass low frequencies, such as cables.	In both Low pass modes, frequencies down to DC and negative frequencies are extrapolated. Therefore, the Start frequency is adjusted when you click Set Freq.Low Pass Because this will affect calibration accuracy, be sure to calibrate AFTER completely setting up your time domain measurement.
Low pass Step	Easiest to identify inductive and capacitive discontinuities in devices that pass low frequencies, such as cables.	
Band pass Impulse	Easiest method - can be used with any frequency sweep. Most useful for measuring band limited devices such as filters and DC blocked cables.	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.

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.

IMPEDANCE	STEP RESPONSE	IMPULSE RESPONSE
OPEN	 Unity Reflection	 Unity Reflection
SHORT	 Unity Reflection = 180	 Unity Reflection = 180
RESISTOR $R > Z_0$		
RESISTOR $R < Z_0$		
INDUCTOR		
CAPACITOR		

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 analyzer 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.

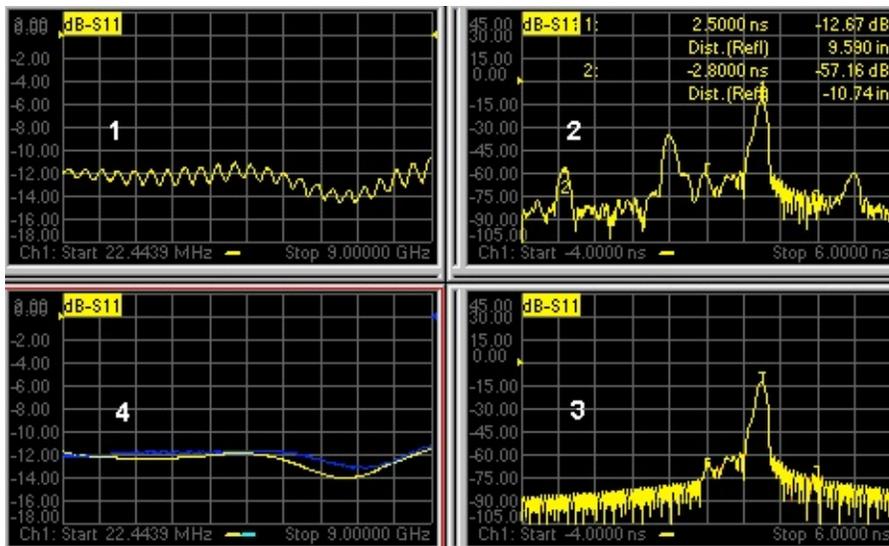
We will gate IN the attenuator response. All other responses will be gated OUT.

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.

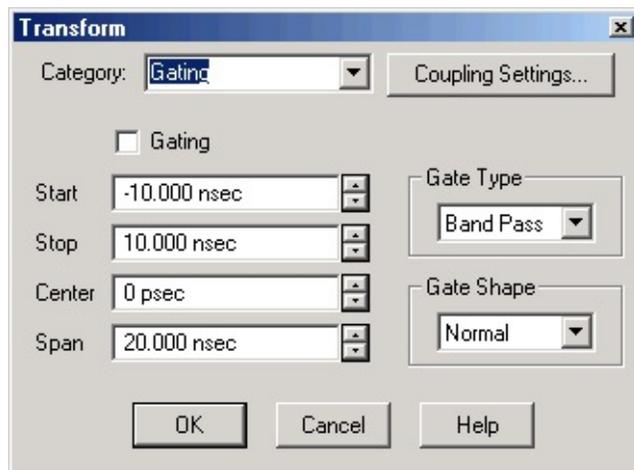
Window 4. On the original frequency measurement, turn Gating ON (Transform remains OFF). View the measurement without the effects of the two unwanted discontinuities. The blue trace is a measurement of the 6 dB attenuator with the unwanted discontinuities PHYSICALLY removed. The difference between the two traces in window 4 is the effect of "masking".



Learn how to launch the Transform dialog box



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.

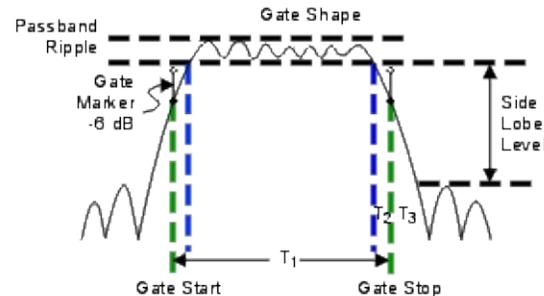
Center Specifies the value at the center of the area that is affected by the gating function. This value can be anywhere in the analyzer range.

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



Gate Shape	Passband Ripple	Sidelobe Levels	Cutoff Time	Minimum Gate Span
Minimum	± 0.1 dB	-48 dB	1.4/Freq Span	2.8/Freq Span
Normal	± 0.1 dB	-68 dB	2.8/Freq Span	5.6/Freq Span
Wide	± 0.1 dB	-57 dB	4.4/Freq Span	8.8/Freq Span
Maximum	± 0.01 dB	-70 dB	12.7/Freq Span	25.4/Freq Span

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.

- T_1 is the gate span, which is equal to the stop time minus the start time.
- T_2 is the time between the edge of the passband and the 6 dB point, representing the cutoff rate of the filter.
- T_3 is the time between the 6 dB point and the edge of the gate stopband.
- For all filter shapes T_2 is equal to T_3 , 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

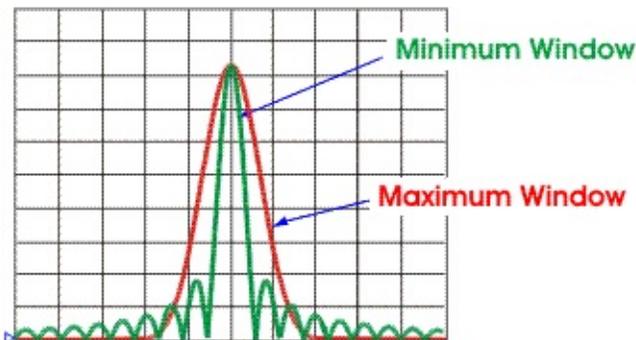
recommended gate span for proper operation. This is a consequence of the finite cutoff rate of the gate. If you specify a gate span that is smaller than the minimum span, the response will show the following effects:

- 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:

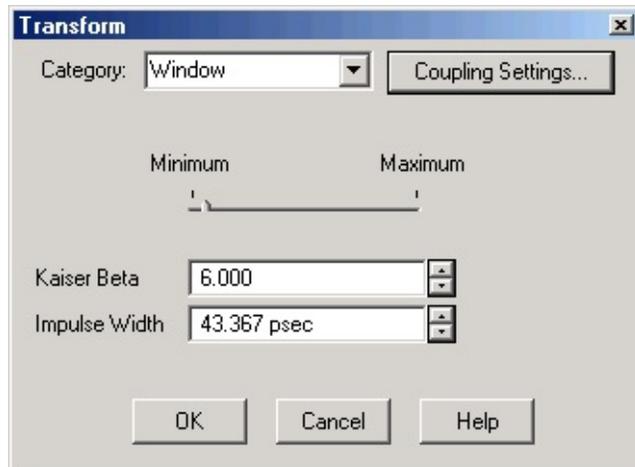
- **Minimum Window = Better Response Resolution** - the ability to resolve between two closely spaced responses.
- **Maximum Window = Dynamic Range** - the ability to measure low-level responses.



Learn how to launch the Transform dialog box



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 the 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

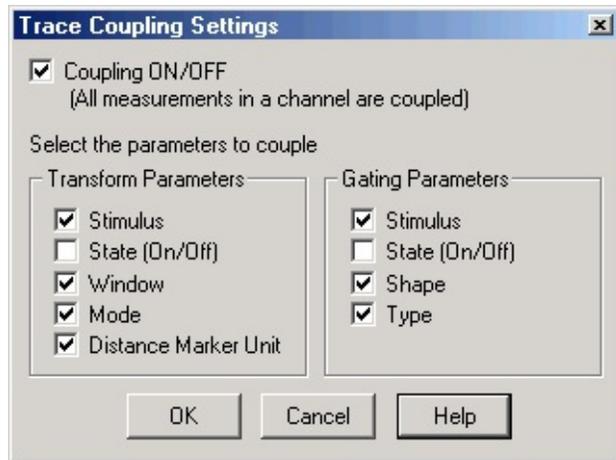
Learn more about [Windowing](#) (top)

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**

Trace Coupling Settings dialog box help



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.

Note: Trace coupling applies ONLY to the Y-axis scale/reference settings. There are no changes to your data as a result of trace coupling.

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

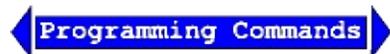
Stimulus Start, Stop, Center, and Span TIME settings.

State (On/Off) Gating ON and OFF

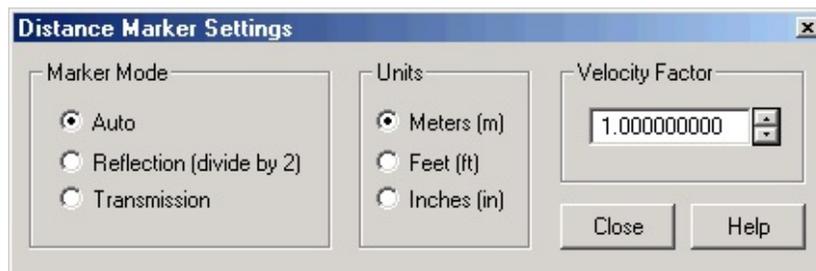
Shape Minimum, Normal, Wide, and Maximum

Type Bandpass and Notch

To launch the Distance marker dialog box, click **Dist. Marker Settings** on the **Transform** dialog box.



Distance Marker Settings dialog box help



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](#).

You can read out impedance versus time by creating a marker on a Time Domain trace, then changing the marker format to R+jX. [Learn how](#).

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.

Troubleshooting the PNA

By running a few checks, you can identify if the analyzer is at fault. Before calling Keysight 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 Keysight 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 2 of the PNA Service Guide, "System Tests, Verifications, and Adjustments," . You can download a copy of the Service Guide from our Web site: <http://www.Keysight.com>.
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, Keysight 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 Keysight.**

We respect the privacy of our customers. However, access to information that helps us improve the PNA is a benefit to both Keysight and you. Please take the time to contact us or email the saved information to **na_support@Keysight.com**.

The following procedure shows how to do this:

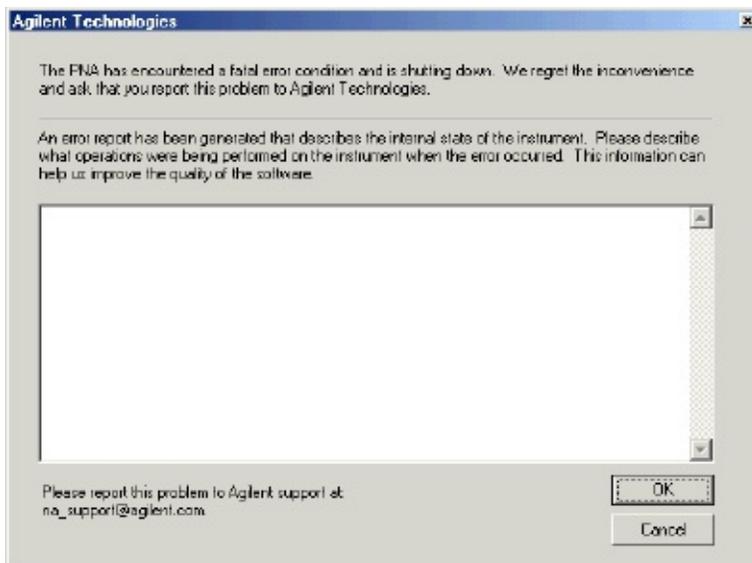
1. A message box immediately appears on the screen containing the location of a directory. Please record this message. If you miss the message, you can find the directory location using the Windows Event Log: On the PNA, click Start, Settings, Control Panel, Administrative Tools, Event Viewer. Double-click 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@Keysight.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

Check the PNA Service Guide for specific troubleshooting procedures to help identify problems. You can download a copy of the Service Guide from our Web site: <http://www.Keysight.com>.

Last modified:

10/16/06 Added phase lock lost

Analyzer 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

See Also: [About 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 programmatically 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: ecaldrvr.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 analyzer 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 analyzer.

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: <x> 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/Keysight/Network Analyzer/analyzerCalSets.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/Keysight/Network Analyzer/analyzerCalSets.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/Keysight/Network Analyzer/analyzerCalSets.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/Keysight/Network_Analyzer/analyzerCalSets.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/Keysight/Network Analyzer/analyzerCalSets.dat. Remove this file, then restart the application.

EventID: E8020286 (hex)

Message: 647

"Custom cal type not found."

Severity: Error

Further explanation:

Suggestions:

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 analyzer 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 analyzer 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 analyzer 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 it's 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 multipoint 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 analyzer 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 unlevelled."

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: 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 Keysight Support. Technical Information: DSP Type

Severity: Error

EventID: E8030368 (hex)

Message: 873

DSP Error. Please Contact Keysight Support. Technical Information: DSP Type 2

Severity: Error

EventID: E8030369 (hex)

Message: 874

DSP Error. Please Contact Keysight Support. Technical Information: DSP Type 3

Severity: Error

EventID: E803036A (hex)

Message: 875

DSP Error. Please Contact Keysight Support. Technical Information: DSP Type 4

Severity: Error

EventID: E803036B (hex)

Message: 876

DSP Error. Please Contact Keysight 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 analyzer 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 analyzer](#) 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 analyzer 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 analyzer](#) 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 unratiod 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 analyzer software may be in an unstable state and it is recommended that the analyzer 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 [analyzer 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 [analyzer 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 analyzer](#).

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 analyzer that doesn't have a source attenuator.

EventID: E8070754 (hex)

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: E8070755 (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: E8070756 (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: E8070757 (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: E8070758 (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: E8070759 (hex)

Message: 1882

The port number was outside the range of allowed port numbers.

Severity: Error

EventID: E807075A (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 [analyzer 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 allotted and is attempting recovery."

Severity: Warning

Further explanation: None

EventID: A808080C (hex)

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 analyzer 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:

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 Keysight Technologies GPIB driver can not be loaded or unloaded."

Severity: Error

Further explanation: None

Suggestions: If the problem persists, from the analyzer 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 analyzer.

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 analyzer 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 analyzer.

EventID: E808086D (hex)

Message 2158

"The Keysight 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 analyzer 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 analyzer 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: 2179

Failed to open gen.lic.

Severity: Error

EventID: E8080883 (hex)

About Error Messages

Analyzer errors and Operating System errors are displayed and logged in an error file. You can choose how to display errors, or choose to not display errors at all.

- [Error Display](#)
- [View Error Log](#)
- [List of PNA Errors](#)
- [SCPI Errors](#)

Other System topics

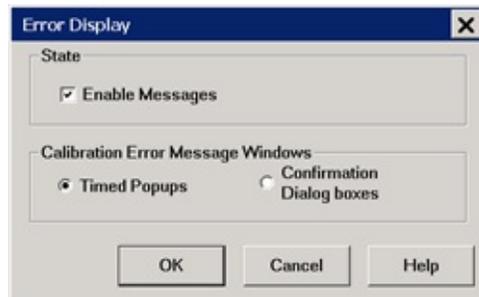
Error Display

By default, error messages appear on the 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	Using Menus
<ol style="list-style-type: none">1. Press SYSTEM2. then [Help]3. then [Error Messages]4. then [Error Display]	<ol style="list-style-type: none">1. Click Help2. then Error Messages3. then Error Display

 **Programming Commands** 

Error Display dialog box help



On Preset, these settings revert to their defaults (enabled, timed popups).

Enable Messages Check to display all error messages as they occur. Clear to suppress the display of 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.

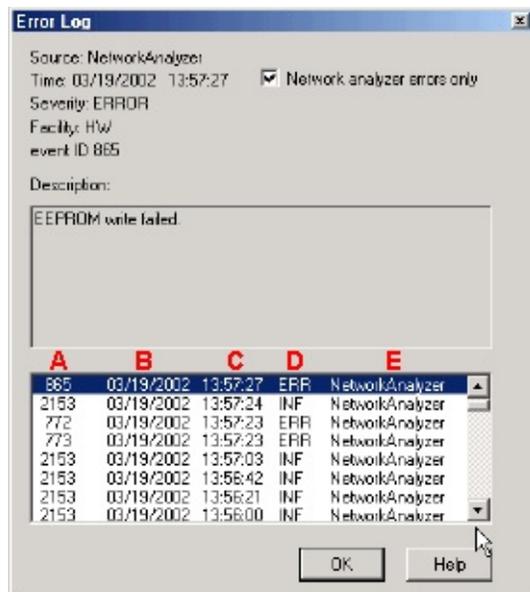
Confirmation Dialog boxes Displays error messages in a standard dialog box. You then choose **OK** or **Cancel** to close the dialog box, or press **Help** to get further information on the error message.

View Error Log

The analyzer Error Log is a list of all events that have occurred. (Events are used in programming the analyzer using COM.) Analyzer errors are a subset of events. Only events with severity codes of ERROR are displayed on the 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	Using Menus
<ol style="list-style-type: none"> 1. Press SYSTEM 2. then [Help] 3. then [Error Messages] 4. then [View Error Log] 	<ol style="list-style-type: none"> 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 analyzer errors. Clear to view all errors that occur on all applications of the computer.

Description Error message that appears on the analyzer 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
- INFormational - events that occur without impact on the measurement integrity
- WARning - 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**

Last Modified:

16-Sep-2010	Updated to PNA-X menu
24-Mar-2010	Updated to Error Display (A.09.20)
1-Jan-2007	MX Added UI

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 ESD Supplies](#)

Other Support topics

For product and order information:

- Visit www.Keysight.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.

Coax Mechanical Calibration Kits

Model	Connector Type	Frequency Upper Limit
85032B	Type-N (50 Ohm)	6 GHz
85032F	Type-N (50 Ohm)	9 GHz
85054B	Type-N (50 Ohm)	18 GHz
85036E	Type-N (75 Ohm)	3 GHz
85050B	7 mm	18 GHz
85033D	3.5 mm	6 GHz
85038A	7-16	7.5 GHz
85033E	3.5 mm	9 GHz
85052B	3.5 mm	26.5 GHz
85052C	3.5 mm TRL	26.5 GHz
85056K	2.92 mm	50 GHz
85056A	2.4 mm	50 GHz
85058B/E (data-based)	1.85 mm	67 GHz
85059A (data-based)	1.00 mm	110 GHz

Waveguide Mechanical Calibration Kits

Model	Connector Type	Frequency Range
-------	----------------	-----------------

X11644A	WR-90	8.2-12.4 GHz
P11644A	WR-62	12.4-18 GHz
K11644A	WR-42	18-26.5 GHz
R11644A	WR-28	26.5-40 GHz
Q11644A	WR-22	33-50 GHz
U11644A	WR-19	40-60 GHz
V11644A	WR-15	50-75 GHz

Electronic Calibration (ECal)

Model	Connector Type	Frequency Range
RF Two-Port		
85091C	7 mm (APC-7)	300 kHz-9 GHz
85092C	Type-N (50 ohm) Port B available with 3.5 mm or 7-16 ^a	300 kHz-9 GHz
85093C	3.5 mm Port B available with Type-N (50 ohm) or 7-16 ^a	300 kHz-9 GHz
85096C	Type-N (75 ohm)	300 kHz-3 GHz
85098C	7-16 ^a Port B available with Type-N (50 ohm) or 3.5 mm	300 kHz-7.5 GHz
85099C	Type-F	300 kHz-3 GHz
RF Four-Port		
N4431B	3.5mm (f) (four-port), Type-N (f)	9 kHz ^b -13.5 GHz

	(four-port), Mixed connector types	
N4432A Option 020	Type-N (f) (four-port)	300 kHz-18 GHz (available Feb. 2006)
N4432A Option 030	APC 7 (four-port)	300 kHz-18 GHz (available Feb. 2006)
N4433A Option 010	3.5mm (f) (four-port)	300 kHz-20 GHz (available Feb. 2006)
Microwave Two-Port		
N4690B	Type-N (50 ohm)	300 kHz-18 GHz
N4691B	3.5 mm	300 kHz-26.5 GHz
N4692A	2.92 mm	10 MHz-40 GHz
N4693A	2.4 mm	10 MHz-50 GHz
N4694A	1.85 mm	10 MHz-67 GHz
N4696BA	7 mm	300 kHz-18 GHz

^a Limits ECal module high frequency to 7.5 GHz.

^b 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

Model	Connector Type	Frequency Range
85055A	Type-N (50 Ohm)	300 kHz-9 GHz
85053B	3.5 mm	300 kHz-26.5 GHz
85057B	2.4 mm	.045-50 GHz
R11645A	WR-28	26.5-40 GHz

Q11645A

WR-22

33-50 GHz

Adapters and Accessory Kits

Model	Description
11878A	Type-N to 3.5 mm Adapter Kit
11525A	Type-N (m) to 7 mm (APC-7)
11853A	Type-N Accessory Kit
11900B	2.4 mm (f) to 2.4 mm (f)
11900C	2.4 mm (f) to 2.4 mm (m)
85130G	Test Port Adapter Set, 2.4 mm (f) to 2.4 mm (m,f)
11901B	2.4 mm (f) to 3.5 mm (f)
11901D	2.4 mm (f) to 3.5 mm (m)
85130F	Test Port Adapter Set, 2.4 mm (f) to 3.5 mm (m,f)
11902B	2.4 mm (f) to 7 mm (APC-7)
11920A	1 mm (m) to 1 mm (m)
11920B	1 mm (f) to 1 mm (f)
11920C	1 mm (m) to 1 mm (f)
11921A	1 mm (m) to 1.85 mm (m)
11921B	1 mm (f) to 1.85 mm (f)
11921C	1 mm (m) to 1.85 mm (f)
11921D	1 mm (f) to 1.85 mm (m)
11922A	1 mm (m) to 2.4 mm (m)
11922B	1 mm (f) to 2.4 mm (f)

11922C	1 mm (m) to 2.4 mm (f)
11922D	1 mm (f) to 2.4 mm (m)

Test Port Cables

Model	Description
N4697E	1.85 mm (f) to 1.85 mm (rugged f) flexible (single)
N4697F	1.85 mm (rugged f, f) to 1.85 mm (rugged m, rugged f) flexible (set)
N6315A	Type-N (m) to Type-N (f), 16 in. (single)
N6314A	Type-N (m) to Type-N (m), 24 in. (single)
85133D	2.4 mm (f) to 2.4 mm (m,f) semi-rigid (set)
85133F	2.4 mm (f) to 2.4 mm (m,f) flexible (set)
85134D	2.4 mm (f) to 3.5 mm (m,f) semi-rigid (set)
85134F	2.4 mm (f) to 3.5 mm (m,f) flexible (set)

USB Peripherals

Model	Description
N4688A	CD RW drive - with USB cable.
N4689A	USB 4-port hub - for connecting additional USB peripherals.
82357A	USB/GPIB Interface - for controlling GPIB devices through USB. Learn more about using the 82357A with the PNA

Connector and ESD Supplies

See [ESD topic](#)

See more [Connector Care supplies](#)

--	--

Part Number	Description
9300-1367	Adjustable antistatic wrist strap
9300-0980	Antistatic wrist strap grounding cord (5 foot)
9300-0797	Static control table mat (2 foot x 4 foot) with earth ground wire
9300-1126	ESD heel strap
1401-0248	ESD Safe End-Cap, Type-N (m)
1401-0247	ESD Safe End-Cap, Type-N (f)
1401-0214	Standard End-Cap, Type-N (m)
1401-0225	Standard End-Cap, Type-N (f)

Last Modified:

26-Feb-2013 Changed Type-N F end cap # and fixed supplies cat

9-Jan-2009 Added footnote for N4431B

1-Dec-2008 Added link to connector care

82357A USB / GPIB Interface

The Keysight 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

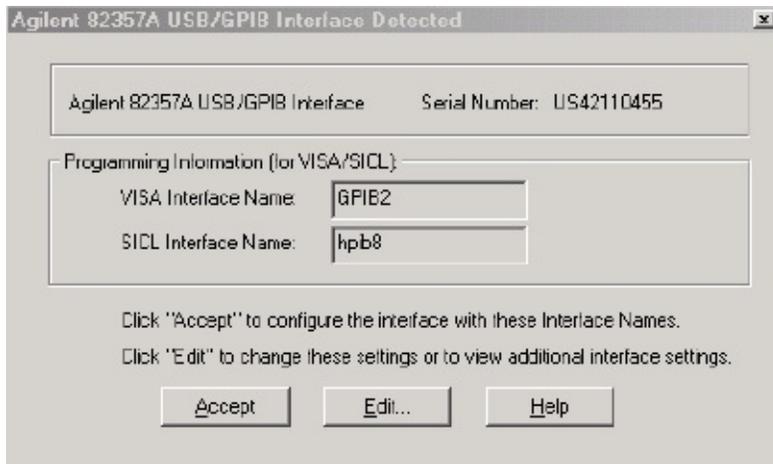
The 82357A can be used to connect a GPIB device using the PNA USB for any PNA application. In addition, the 82357A can be used to connect a power meter for a source power calibration.

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 Keysight IO libraries from the CDROM that was shipped with the 82357A. If not available, download them from www.Keysight.com (search for **82357A**)

Configure the 82357A USB/GPIB Interface

When the 82357A is connected to the PNA USB, the following dialog box appears:



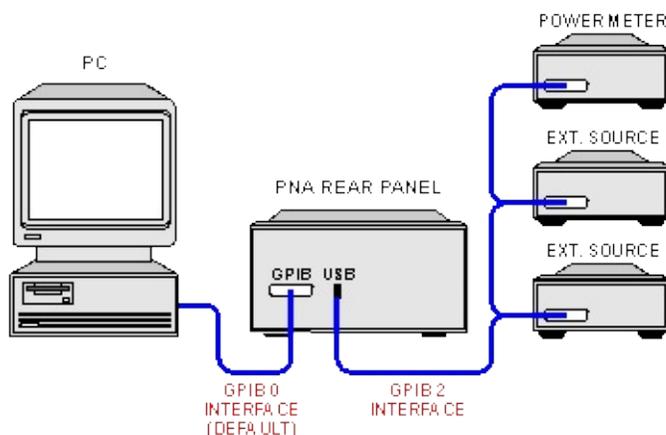
Normally, you do NOT need to edit these settings. The 82357A USB/GPIB Interface is configured automatically as the next unused VISA interface. This is usually **GPIB2** unless you have already configured it for another purpose.

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 GPIB Interface and USB/GPIB Interface should never be connected together.



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: Select **GPIB** at the Power Meter Settings dialog and specify the GPIB address of the power meter.
- To control other devices through your own program using the 82357A, you must include the new GPIB Interface number when addressing the devices.

Last modified:

23-Apr-2012 SPC update

Firmware Update

PNA firmware updates are available to you at no cost in a self-extracting Install Shield file. The update 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 update firmware. See http://na.support.keysight.com/pna/firmware/PNA_support_matrix.doc.

The following options are available for you to update 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**.

Tip: Use Move App to Back to cause the PNA application to move behind this application on the screen.

Note: After a firmware update...

- 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.

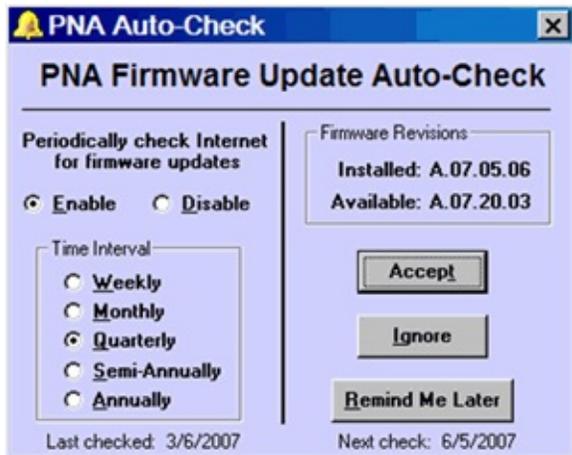
Auto-Check is run only when the PNA application is started. Once the PNA application is running, it will not check for updates again until it is restarted.

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, press **System**, then **Service**, then **AgileUpdate**. From within AgileUpdate, click **AutoCheck**. These preferences are stored in the PNA registry. Future firmware updates 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

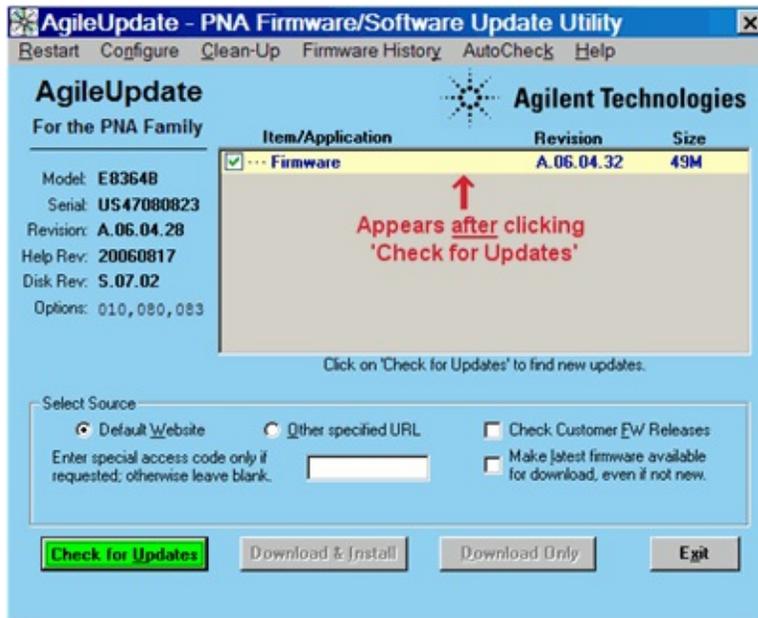
1. Press **SYSTEM**
2. then **[Service]**
3. then **[AgileUpdate]**

Using 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 Keysight. 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 Keysight site that contains update 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 Keysight 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 Keysight 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.

Keysight 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 Keysight website. You can then transfer the file from your PC to your analyzer over a LAN or other means.

1. Go to <http://na.support.keysight.com/pna/firmware>
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 update 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:

29-Aug-2013	Upgrade to update Move to back
12-Oct-2009	Added customer release (9.2)
13-Mar-2008	Added limitations note

M937x Configurations and Options

Included with each M937x Module is a mouse and keyboard.

- [Options](#)
- [Warranty Period](#)

See Also

- M937x Series Configuration Guide (requires an Internet connection)
- Click **Help** then **About Network Analyzer** to view the options that are installed on your analyzer.

Other Support Topics

M937x Series Modules

- **M9370A - 300 kHz to 4 GHz.**
- **M9371A - 300 kHz to 6.5GHz.**
- **M9372A - 300 kHz to 9 GHz.**
- **M9373A - 300 kHz to 14 GHz.**
- **M9374A - 300 kHz to 20 GHz.**
- **M9375A - 300 kHz to 26.5 GHz.**

Options

[Learn how to install options.](#)

Option	Required Model/Options	Description
103	None	Fast Sweep Mode. Learn more.
010	None	Time Domain
551	None	Calibrated Multiport Measurements



Option Enable Utility

The Option Enable utility allows you to perform the following activities on your analyzer.

- 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

[Configurations and Options](#)

Keywords

To add certain options, you need a keyword that is provided by Keysight. 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

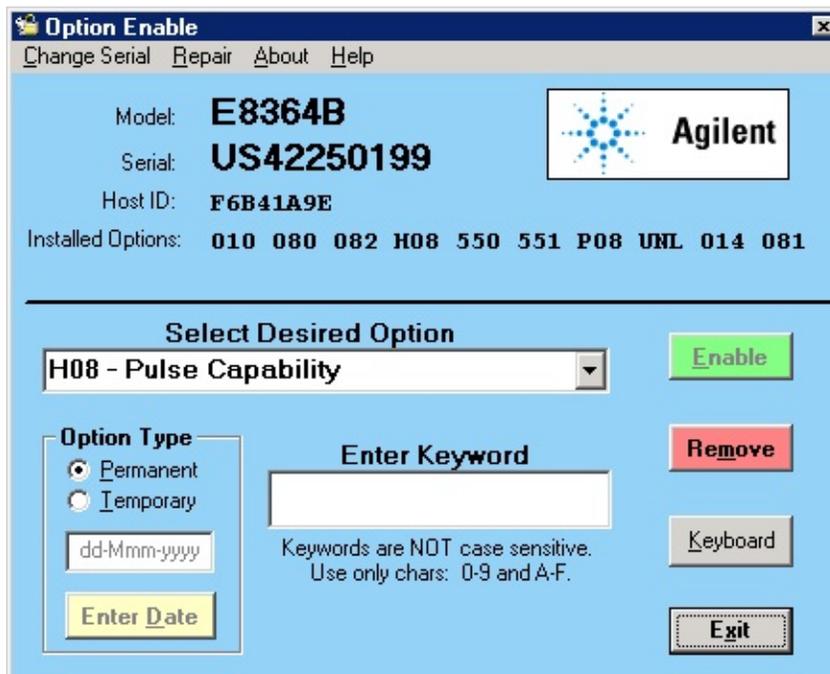
Any software option can also be installed on a temporary basis for a specified amount of time. This allows you to evaluate a specific feature or capability at no cost.

If the license key provided by Keysight 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, Keysight 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, press **System**, then **Service**, then **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:\users\public\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 Configurations and 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 Keysight
 - 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 Keysight 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 analyzer 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.

See Also: [Instrument Calibration Verification \(Sticker\)](#)

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 Keysight Technologies

service centers. See [Technical Support](#).

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 Keysight 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 Keysight Technologies service center:

Keysight 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.Keysight.com/find/calibration.

Other Resources

The following network analysis resources are also available.

Document Resources

[Application Notes](#)

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

[Analyzer 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 than 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, an 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 executing 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 placeholder 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

- 200 std_execGen 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.
- 201 std_invalidWhileInLocal Invalid while in local
- 202 std_settingsLost Settings lost due to rtl
- 203 std_commandProtected Command protected - Indicates that a legal password-protected program command or query could not be executed because the command was disabled.
- 210 std_trigger Trigger
- 211 std_triggerIgnored Trigger ignored
- 212 std_armIgnored Arm ignored
- 213 std_initIgnored Init ignored
- 214 std_triggerDeadlock Trigger deadlock
- 215 std_armDeadlock Arm deadlock
- 220 std_parm Parameter - Indicates that a program data element related error occurred.
- 221 std_settingsConflict Settings conflict - Indicates that a legal program data element was parsed but could not be executed due to the current device state.
- 222 std_dataOutOfRange 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
- 223 std_tooMuchData 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.
- 224 std_illegalParmValue Illegal parameter value - Indicates that the value selected was not part of the list of values given.

- 225 std_noMemoryForOp Out of memory - The device has insufficient memory to perform the requested operation.
- 226 std_listLength Lists not same length - Attempted to use LIST structure having individual LIST's of unequal lengths.
- 230 std_dataCorruptOrStale Data corrupt or stale - Indicates invalid data, a new reading started but not completed since the last access.
- 231 std_dataQuestionable 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, 10.7.6.3.
- 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 488.2, 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 could 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
- 293 std_nameAlreadyExists Referenced name already exists
- 294 std_incompatibleType Incompatible type

-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

.

Analyzer-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 calNotValidForConfidenceChe	"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." "A valid divisor buffer is required before

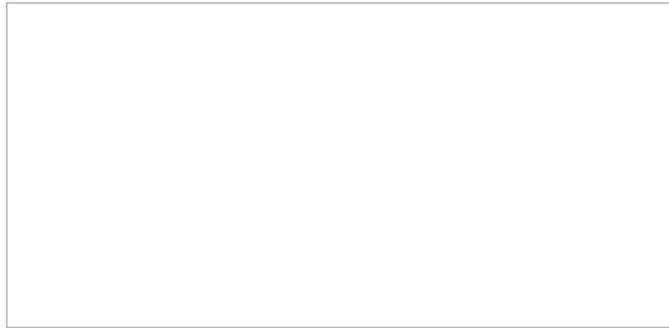
170	noDivisorBuffer	normalization can be turned on."
171	InvalidReceiverPowerCalParagraph	"Receiver power cal requires the measurement to be of unratioed power."
172	ecalCouldNotConfigure	"Could not configure the Electronic Calibration system. Check to see if the module is plugged into the proper connector."
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.Keysight.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

Luxemburg

(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

(alt) (+81) 426 56 7832

(fax) 0120 421 678

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

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 Keysight 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.

Procedures

Note: You must be logged onto the PNA as an Administrator to perform an adjustment. Learn more.

Click **Utilities**, then **System**, then **Service**, then **Adjustments**

At the Adjustments selection, click **10 MHz Frequency Adjustment**

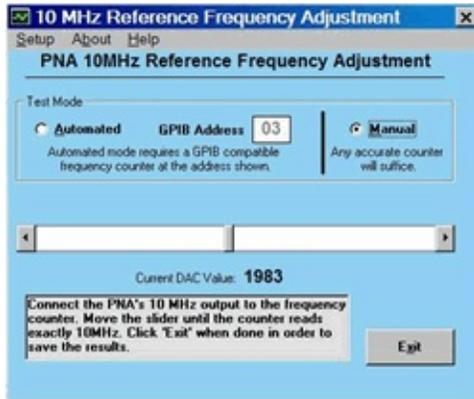
Procedure for GPIB Counters Only

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. On the PNA, press **System** menu, then **Service**, then **Adjustments**, then **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.

Data Storage

- The correction data is stored in the EEPROM on the 10MHz Ref board assembly.

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:

- | | |
|-------------|---------------------|
| 3-Jan-2014 | Updated per JV |
| 20-May-2013 | Updated per JV |
| 9-May-2008 | Updated with new UI |

Diagnostic Tools, Utilities, and Adjustments

The following Tools, Utilities, and Adjustments are available to help you keep your VNA at peak performance.

Diagnostic Tools

- Operators Check
- System Verification

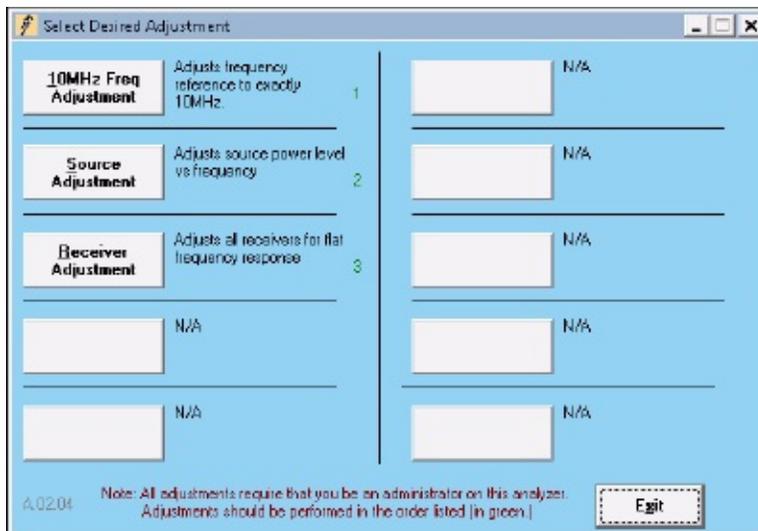
Utilities

- Receiver Display
- Restore ECal Memory
- Receiver Temperature

Adjustments

Not all of the adjustments listed below are valid for every VNA model. Only the adjustments needed for each particular VNA will be listed.

To access the service adjustments on the VNA, click **Utilities**, then **System**, then **Service**, then **Adjustments**



- 10 MHz Reference Frequency Adjustment

- Source Adjustment
- Receiver Adjustment

Operator's Check

- [Overview](#)
- [How to Run the Operator's Check](#)
- [Operators Check Dialog Box Help](#)

Tip: Use Move App to Back to cause the VNA application to move behind this application on the screen.

Overview

The Operator's Check should be performed when you first receive your VNA, and any time you wish to have confidence that the VNA is working properly.

Notes

- The Operator's Check does not verify performance to specifications. To verify VNA performance to specifications, run [System Verification](#).
- Allow the VNA to warm up for 30 minutes before considering a failed test to be valid.
- The Operator's Check can NOT be run with a Multiport test set enabled.

The Pass/Fail criteria used in the Operator's Check identifies **obvious failures** in the following portions of the VNA 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

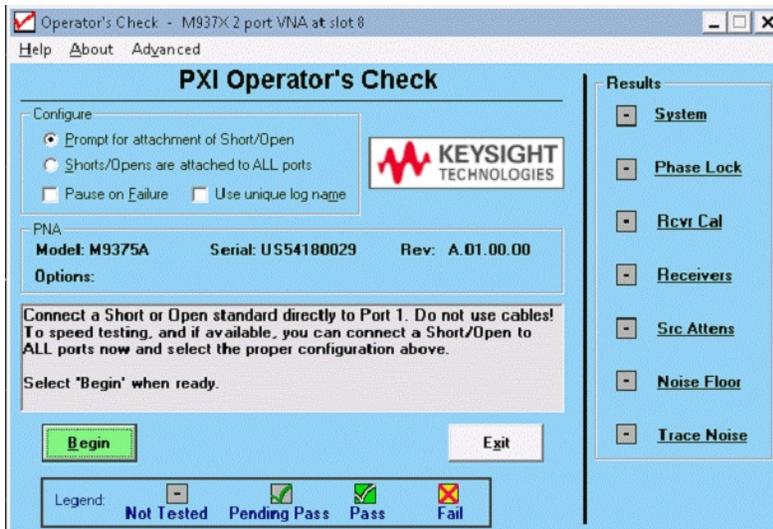
Using front-panel **HARDKEY** [softkey] buttons

Using Menus

1. Press **SYSTEM**
2. then **[Service]**
3. then **[Operator's Check]**

1. Click **Utility**
2. then **System**
3. then **Service**
4. then **Operator's Check**

1. Connect one or more standards (see **Configure**).
2. Click **Begin** and **Continue** (if necessary) until "Operator's Check is complete!" appears.



This dialog box will look slightly different, depending on VNA 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 unlevelled condition to appear during portions of the Operator's 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.

VNA Shows information about the VNA 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 VNA 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.

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 analyzer.

A network analyzer measurement "system" includes the analyzer, calibration kit, test cables, and any necessary adapters. The system verification software in the analyzer 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 analyzer performance tests are used to verify the analyzer's conformance to "instrument" specifications.

The system verification utility verifies the analyzer 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 Keysight. 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 analyzer.

- **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

VNA Models with 3.5 mm test ports

Equipment Type	3.5 mm	Type-N
Calibration kit or ECAL Module	85052B/C/D	85054B/D
	N4691A	N4690A
Verification kit	85053B	85055A
RF Cable(s)	Single: 85131C/E Pair: 85131D/F	Single: 85132C/E Pair: 85132D/F
Adapters	None	<u>Single:</u> 85130C and one 7mm-to-Type-N from 85054B cal kit <u>Pair:</u> Two 7mm-to-Type-N from 85054B cal kit

Cable Substitution

The test port cables specified for the analyzer 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-Keysight 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.

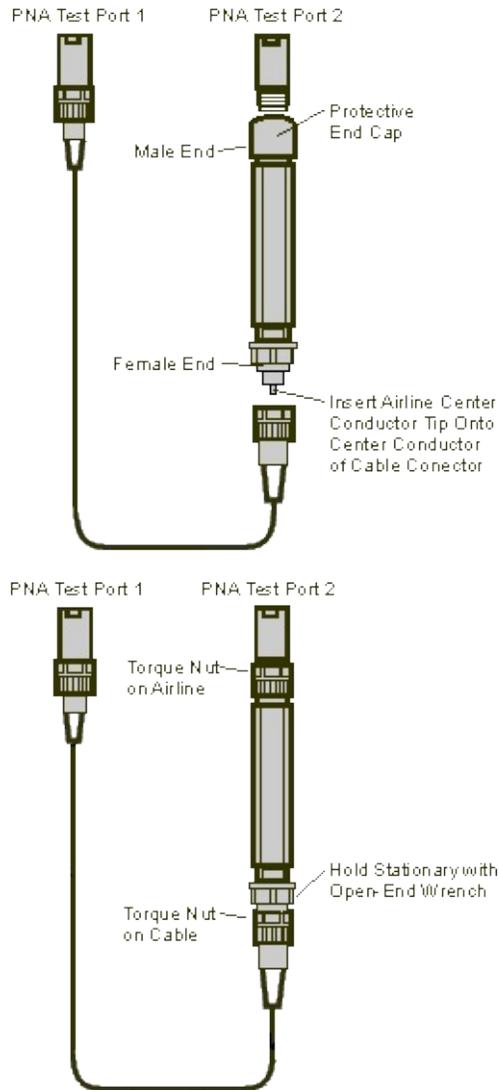
Calibration Kit Substitution

Non-Keysight calibration kits are not recommended nor supported.

Precautions for Handling Airlines

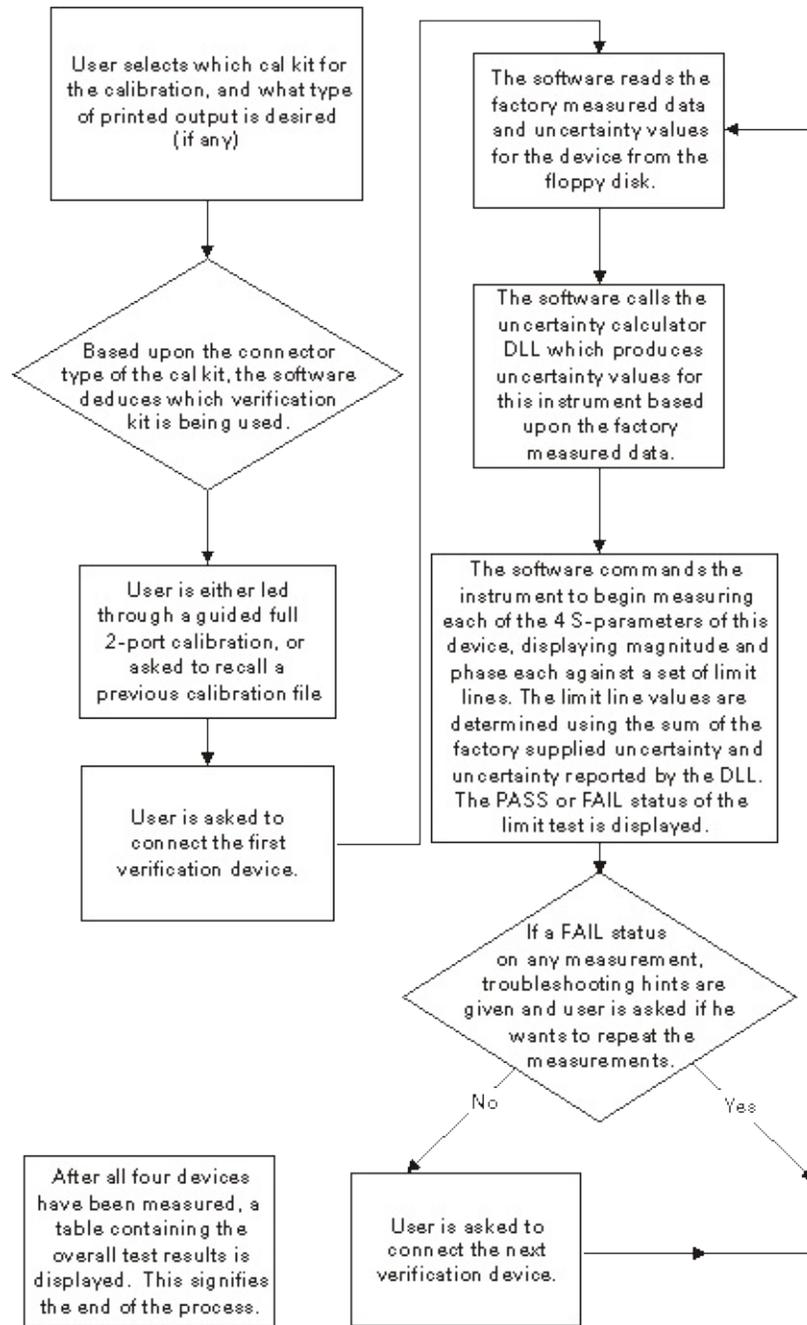
When you are using the airlines in the verification kit, observe the following practices to ensure good measurement techniques.

- 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.

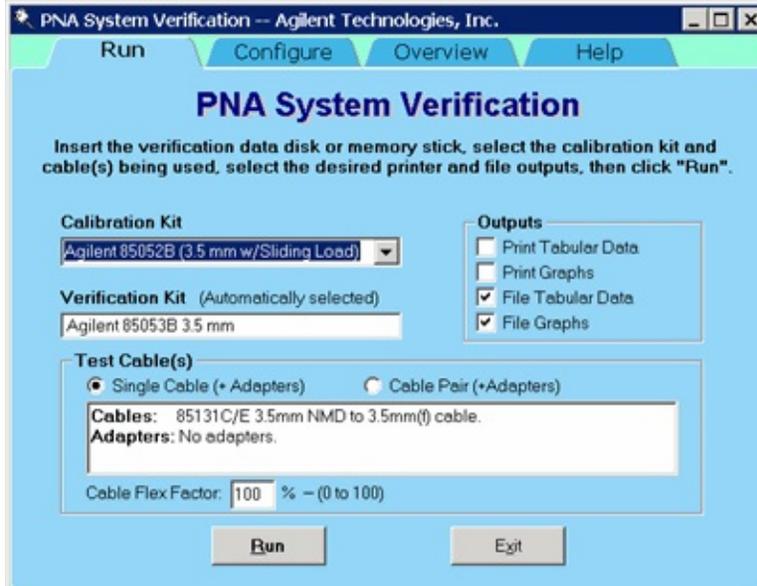


Procedure for System Verification

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 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.

System Verification Dialog



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:\users\public\network analyzer\documents directory.
 - **File Graphs:** Saves a screen image in .PNG format in the c:\users\public\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 analyzer:

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 analyzer:

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 analyzer.

5. The analyzer first sets up for magnitude measurements of all four S-parameters, each parameter in a separate window (lin mag for S_{11} and S_{22} , log mag for S_{21} and S_{12}). 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 analyzer uncertainties reported by the DLL. Upper and lower limits are displayed (factory measured data + uncertainty sum, factory measured data - uncertainty sum). The analyzer takes a sweep, limit test is turned on and PASS/FAIL 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/FAIL 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/FAIL results for all four parameters of each device.

If the System Fails the Verification Test

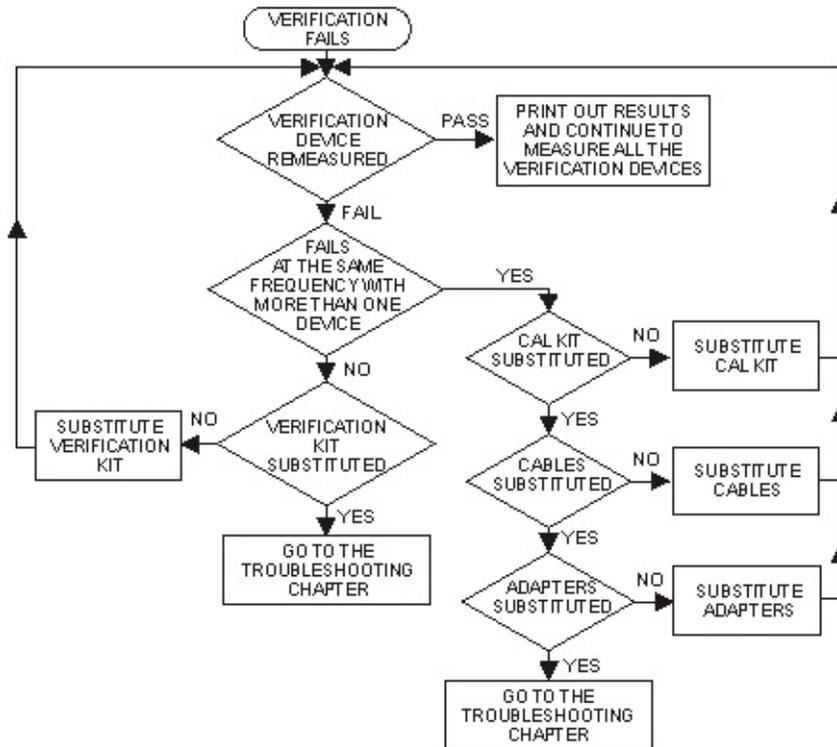
IMPORTANT: Inspect all connections. Do not remove the cable from the analyzer test port. This will invalidate the calibration that you have done earlier.

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.

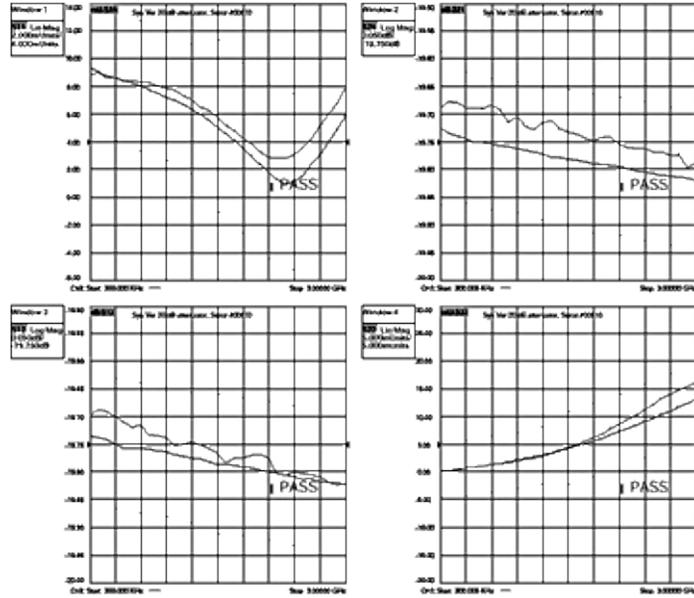
Printout of Tabular Verification Results

The image shows two overlapping printout pages of tabular verification results. Each page contains a table with the following columns: Frequency (MHz), Upper Limit (dB), Lower Limit (dB), Measurement (dB), and Status. The data points are listed for various frequencies, and the status for each measurement is indicated as either 'PASS' or 'FAIL'. The printouts are slightly offset to show multiple pages of data.

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

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 list of supported power meters and sensors.

[See PNA Accessories](#)

Procedure

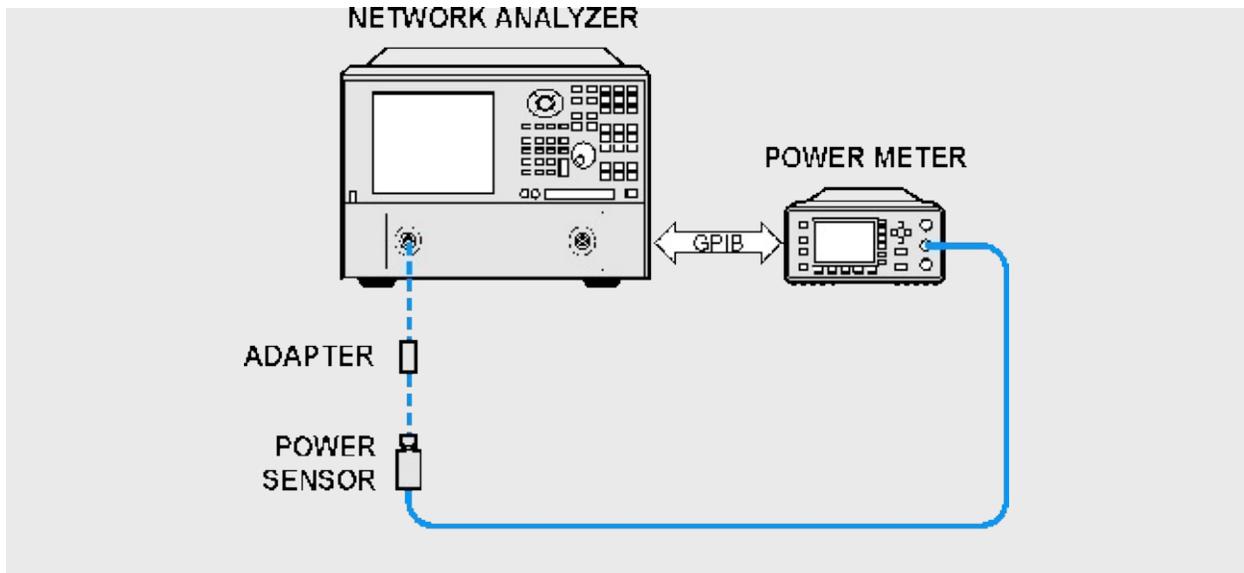
Note: You must be logged onto the PNA as an Administrator to perform an adjustment. [Learn more.](#)

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. Click **Utilities**, then **System**, then **Service**, then **Adjustments**
5. At the [Adjustments selection](#), click **Source Adjustment**
6. There are 3 different version 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.

7. 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.

Troubleshooting

In the event there is a problem with Source Adjustment, please refer to the "Troubleshooting" chapter in the PNA Service Guide.

Data Storage

- The correction data is stored in the flash memory on the Test Set Mother Board.

Last Modified:

22-May-2013 Updated info from JV

6-Apr-2009 Updated models

10-Mar-2009 Changed to Source Adjustment

Receiver Adjustment

This program 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

See list of supported power meters and sensors.

See [PNA Accessories](#)

Notes

- 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).
- In this adjustment, a power sensor with a specified lower frequency limit of 50 MHz may be used on all instrument that have a lower frequency limit of 10 MHz. Any added uncertainty is negligible.
- If using an older style sensor (without built-in correction factors), 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.
- You must be logged onto the PNA as an Administrator to perform an adjustment. [Learn more.](#)

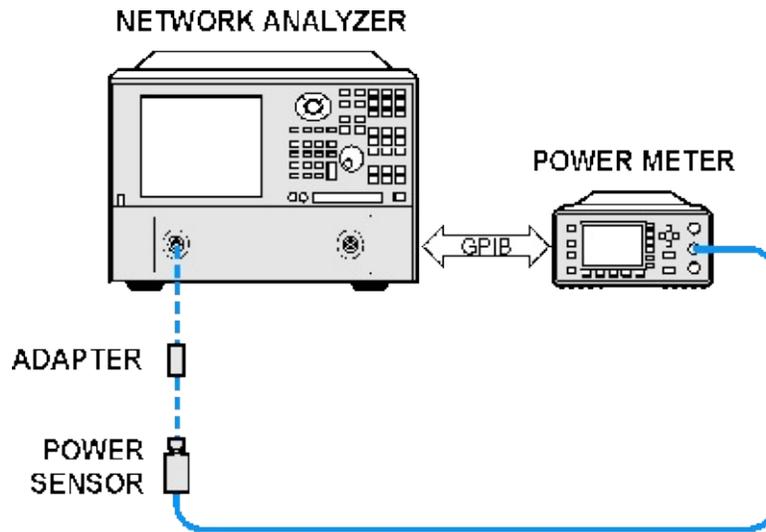
Procedure

1. Click **Utilities**, then **System**, then **Service**, then **Adjustments**
2. At the Adjustments selection, click **Receiver Adjustment**.
3. Connect a GPIB cable between the power meter and network analyzer.
4. Ensure the power sensor(s) are connected to the power meter.
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.

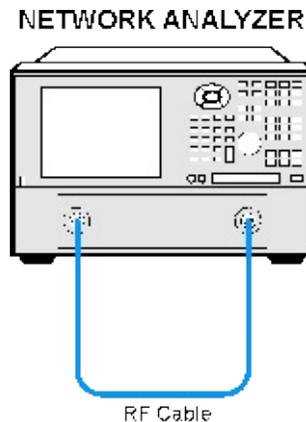
- b. Click **Calibrate** to begin the receiver calibration process. The software prompts you to connect the sensor(s), cable and adapter as needed (see the following graphics).

Connecting sensor(s) to the PNA



Connecting adapter and cable between sensor and PNA

Through connection using the specified cable



Additional Information

- Receiver Adjustment tests all PNA receivers, taking approximately 15 and 45 minutes. Length is dependent on frequency range and number of ports.
- Upon completion of the Receiver Adjustment, a transmission measurement of a good quality cable should appear to be smooth, with slightly increasing loss versus frequency. A reflection measurement of a short or open should appear to be a flat line across the entire frequency range with only a dB or two of variation/ripple. If instead, you see variations of 10-40dB, then the PNA may have a mixer problem. Typically, this means the uncorrected low end frequency phase relative to other receivers is different. This causes the correction algorithm to "blow up" and provide wildly incorrect data. This is almost always a hardware problem and typically one or more receivers must be replaced. The adjustment procedure has a quick test for this and it will show a warning message if excess phase shift is detected, however this test is not definitive and may not always catch every problem.

Troubleshooting

In the event there is a problem with Receiver Adjustment, please refer to the "Troubleshooting" chapter in your PNA Service Guide.

Data Storage

- The correction data is stored in the flash memory on the Test Set Mother Board.
-

Last Modified:

21-May-2013 Changed cal to adjustment.

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 the Service Guide for your VNA. You can download the Service Guide from our website: <http://na.support.keysight.com/pna/> or <http://na.support.keysight.com/pxi/>

How to Start the Receiver Display

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

1. Press **SYSTEM**
2. then **[Service]**
3. then **[Utilities]**
4. then **[Receiver Display]**

Using Menus

1. Click **Utilities**
2. then **System**
3. then **Service**
4. then **Utilities**
5. then **Receiver Display**

[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]**

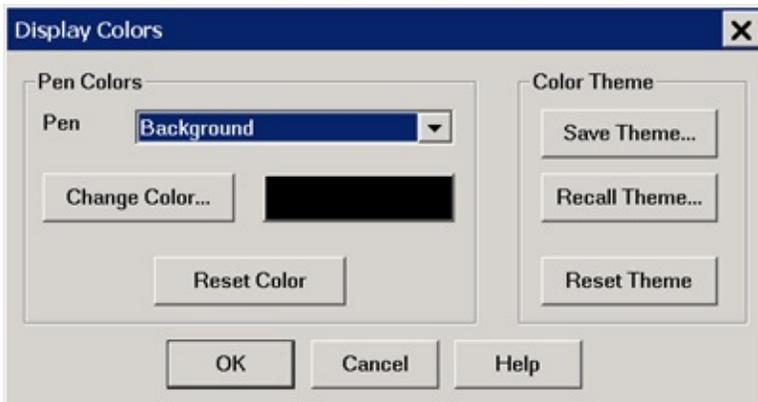
Using Menus

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**



Display and Print Colors dialog box help

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.

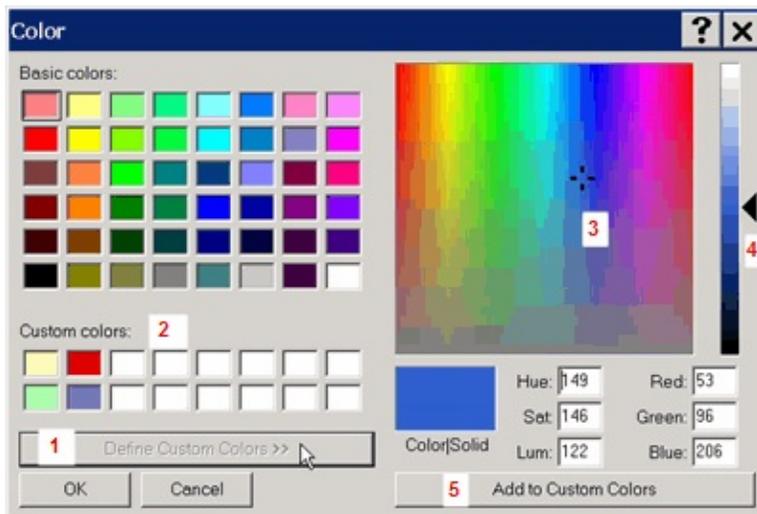
Reset Color Restores the default color for the selected pen.

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

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 test 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 analyzer 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.
- Power limiting does NOT clip power spikes that may occur during **frequency band crossings**.

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 **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 Menus

1. Click **Stimulus**
2. then **Power**
3. then **Power and Attenuators**
4. then **Offsets and Limits**

◀ Programming Commands ▶

Offsets and Limits dialog box help

	Power Limit		Power Offset: Channel 1			
	State	Limit	Source Power	Power Offset	Port Power	Source Cal
Port 1	On	0.00 dBm	-15.00 dBm	3.00 dB	-12.00 dBm	Off
Port 2	Off	100.00 dBm	-15.00 dBm	0.00 dB	-15.00 dBm	Off
Port 3	Off	100.00 dBm	-15.00 dBm	0.00 dB	-15.00 dBm	Off
Port 4	Off	100.00 dBm	-15.00 dBm	0.00 dB	-15.00 dBm	Off
Port 1 Src2	Off	100.00 dBm	-15.00 dBm	0.00 dB	-15.00 dBm	Off

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 test 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 source.

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.

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

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.

Receiver Temperature

This feature allows you to read the current temperature on the receiver microcircuit.

- To read temperature, press **System**, then **Service**, then **Utilities**, then **Receiver Temperature**.
 - The temperature reading is updated with every sweep.
 - Temperature is available in Celsius and Fahrenheit.
 - Temperature can also be read using remote commands.
 - SCPI: **SENSe:TEMPerature?**
-

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

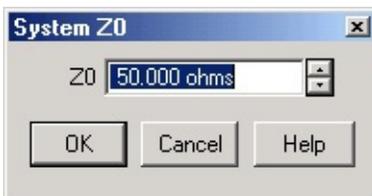
Using front-panel **HARDKEY** [softkey] buttons

1. Press **SYSTEM**
2. then **[Configure]**
3. then **[System Z0]**

Using Menus

1. Click **Utility**
2. then **System**
3. then **Configure**
4. then **System Z0**

◀ Programming Commands ▶



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

Beginning with A.09.50, when selecting a Cal Kit with an impedance other than 50 ohms (Waveguide = 1 ohm), it is **NO LONGER NECESSARY** to change the

System Impedance setting before performing a calibration. The impedance for the calibration is now derived from the Cal Kit '**Connector**' impedance setting.

Last modified:

- 10-Dec-2013 Added connector to Impedance setting
- 11-Sep-2012 Add Z0 note (BH)
- 14-Jul-2011 Edit WG restriction
- 3-Sep-2008 Removed legacy content
- 9/27/06 MX Added UI
- 9/12/06 Added link to programming commands

Application Notes

The following links require an **Internet connection**.

Note: Check out the multimedia VNA Demo presentations, including '[Network Analyzer Basics](#)'.

Calibrations

[AN1287-11 Specifying Calibration Standards and Kits for Keysight Vector Network Analyzers \(5989-4840EN\)](#)

[PN8510-8A TRL Calibration for Non-Coaxial Measurements \(5091-3645E\) Calibrating Standards for In-Fixture Device Characterization \(White Paper\) \(5989-3245EN\)](#)

[Electronic vs. Mechanical Calibration kits: Calibration methods and accuracy \(White Paper\) \(5988-9477EN\)](#)

[On-Wafer Calibration Using a 4-port, 20 GHz PNA-L Network Analyzer \(N5230A Option 240/245\) \(5989-2287EN\)](#)

ECal

[Keysight 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

[AN1408-7 Amplifier Linear and Gain Measurements \(5988-8644EN\)](#)

[AN1408-8 Amplifier Swept-Harmonic Measurements \(5988-9473EN\)](#)

[AN1408-9 Amplifier and CW Swept Intermodulation-Distortion Measurements \(5988-9474EN\)](#)

[AN1408-10 High-power measurements using the PNA \(5989-1349EN\)](#)

[AN1408-16 Power-Added Efficiency \(PAE\) 5989-7293EN](#)

AN1408-17 Making Accurate IMD Measurements with the PNA-X Network Analyzer (5989-7265EN)

AN1408-19 High Power Amplifier Measurements Using NVNA

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)

Antenna and RCS Configurations (White Paper) (5989-0220EN)

Radar Measurements (Application Note) (5989-7575EN)

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)

Backplane Differential Channel Microprobe Characterization in Time and Frequency Domains (White Paper) (5989-3248EN)

Mixer Measurements

AN1408-1 Mixer Transmission Measurements Using the Frequency Conversion Application (5988-8642EN)

AN1408-2 Mixer Conversion-Loss and Group Delay Measurement Techniques and Comparisons (5988-9619EN)

AN1408-3 Improving Measurement and Calibration Accuracy Using the Frequency Converter Application (5988-9642EN)

AN1408-18 Measuring Group Delay of Frequency Converters with Embedded Local Oscillators (5989-7385EN)

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 (5989-0219EN)

Pulsed Measurements

AN1408-11 Accurate Pulsed Measurements (5989-0563EN)

AN1408-12 Pulsed-RF S-Parameter Measurements Using Wideband and Narrowband Detection

AN1408-21 Active-Device Characterization in Pulsed Operation Using the PNA-x (5990-7781EN)

Pulsed Antenna Measurements Using PNA Network Analyzers (5989-0221EN)

Materials Measurements

Basics of Measuring the Dielectric Properties of Materials (5989-2589EN)

Split Post Dielectric Resonators for Dielectric Measurements of Substrates (5989-5384EN)

Other Measurements

AN1287-12 Time Domain Analysis Using a Network Analyzer (5989-5723EN)

AN1408-14 Using the PNA Series to Analyze Lightwave Components (5989-3385EN)

AN1408-15 Using the PNA for Banded Millimeter-Wave Measurements (5989-4098EN)

AN1408-19 High Power Amplifier Measurements Using NVNA (5990-5039EN)

AN1408-20 High-Accuracy Noise Figure Measurements Using the PNA-X MM-Wave Network Analyzers: Analysis of Cable Length on VNA System Performance (5989-1941EN)

Ultra-Low Impedance Measurements Using 2-Port Measurements (White Paper) (5989-5935EN)

Modeling

Utilizing TDR and VNA Data to Develop 4-port Frequency Dependent Models (White Paper) (5989-0638EN)

Advanced Measurements and Modeling of Differential Devices (White Paper)

(5989-4518EN)

Automation

AN 1408-13 [Introduction to Application Development using the PNA \(5980-2666EN\)](#)

[Connectivity Advances for Component Manufacturers \(5980-2782EN\)](#)

[The 'Need for Speed' in Component Manufacturing Test \(5980-2783EN\)](#)

Network Analyzer Basics

This self-paced two hour video discusses the basic concepts of Network Analysis.

From the Internet:

http://na.support.keysight.com/pna/NaBasics/network_analysis_basics.htm in streaming format.

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.support.keysight.com/pna/connectorcare/Connector_Care.htm

Preventing Test Port Connector Damage

Handling and Storing Connectors

Do

Keep connectors clean
Protect connectors with plastic end caps
Keep connector temperature same as analyzer

Do Not

Touch mating-plane surfaces
Set connectors contact-end down
Store connectors loose in box or drawer

Visual Inspection

Do

Inspect connectors with magnifying glass.
Look for metal debris, deep scratches or dents

Do Not

Use a connector with a bent or broken center conductor
Use a connector with deformed threads

Cleaning Connectors

Do

Clean surfaces first with clean, dry compressed air
 Use lint-free swab or brush
 Use minimum amount of alcohol
 Clean outer conductor mating surface and threads

Do Not

Use high pressure air (>60 psi)
 Use any abrasives
 Allow alcohol into connector support beads
 Apply lateral force to center conductor

Gaging Connectors**Do**

Inspect and clean gage, gage master and device tested
 Use correct torque wrench
 zero gage before use
 Use multiple measurements and keep record of readings

Do Not

Use an out of specification connector
 Hold connector gage by the dial

Making Connections**Do**

Align connectors first
 Rotate only the connector nut
 Use correct torque wrench

Do Not

Cross thread the connection
 Twist connector body to make connection
 Mate different connector types

Connector Care and Cleaning Supplies

Description	Web Site
Swabs	http://www.berkshire.com/swabs.shtml
Lint Free Cloths- Air dusters	http://www.ccrwebstore.com
Isopropyl	http://www.techspray.com
Nitrilite Gloves and Finger Cots	http://www.techni-tool.com

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](#)
- [Connector Grades and Performance](#)
- [Adapters as Connector Savers](#)
- [Connector Mating Plane Surfaces](#)

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.

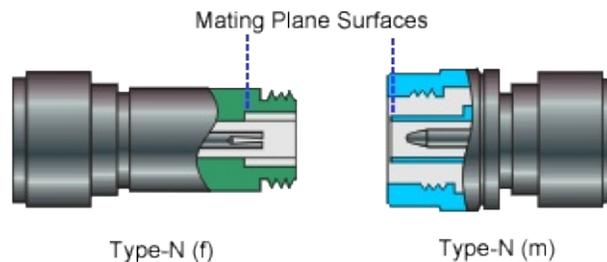
Adapters as Connector Savers

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).

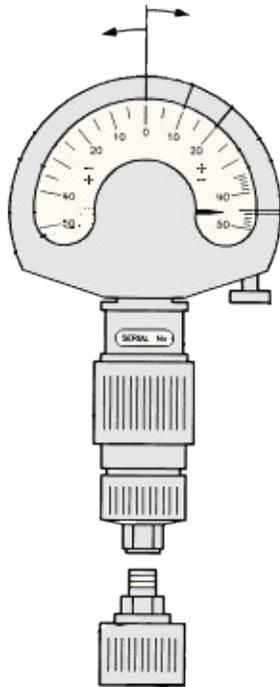


Gaging Fundamentals

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

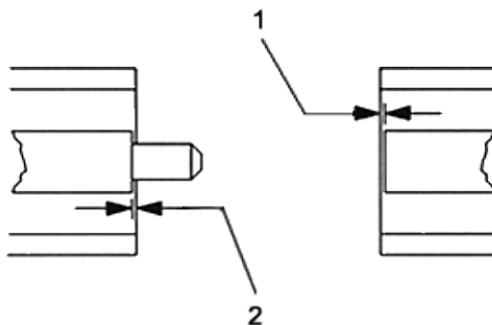
RECESSION PROTRUSION



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 Keysight 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 M Ω 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 M Ω series resistor).

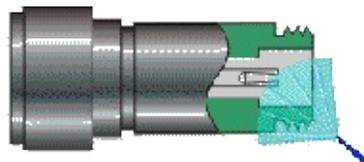
Use a magnifying glass ($\geq 10X$) and inspect connector for the following:

- 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

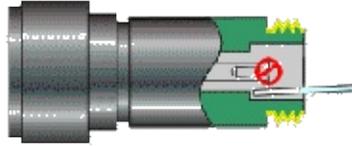
Caution: A damaged or out-of-specification device can destroy a good connector attached to it even on the first connection. Any connector with an obvious defect should be marked for disposal or sent out for repair.

To Clean Connectors

1. Wear a grounded wrist strap (having a 1 M Ω 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.



3. Moisten—do not saturate—a lint-free swab with isopropyl alcohol. See [Cleaning Supplies](#) for recommended type.
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.



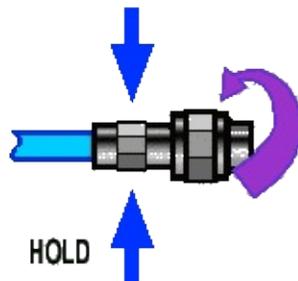
5. Let alcohol evaporate—then use compressed air to blow surfaces clean.
6. Inspect connector. Make sure no particles or residue remains.
7. If defects are still visible after cleaning, the connector itself may be damaged and should not be used. Determine the cause of damage before making further connections.

To Make Connections

1. Wear a grounded wrist strap (having a 1 M Ω 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.

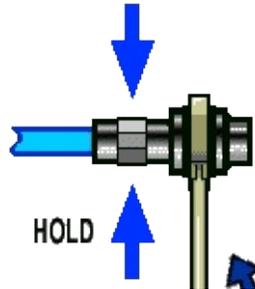


4. Carefully push the connectors straight together so they can engage smoothly. Rotate the connector nut (not the device itself) until finger-tight, being careful not to cross the threads.



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.

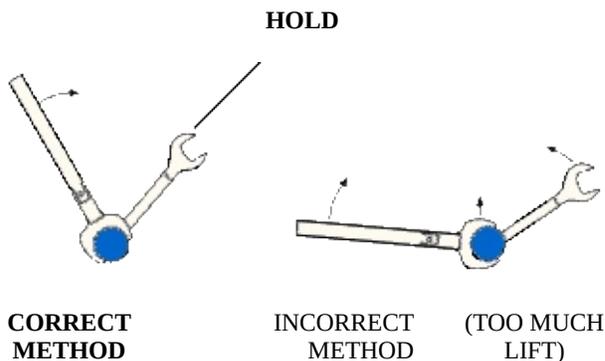


To Separate a Connection

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.

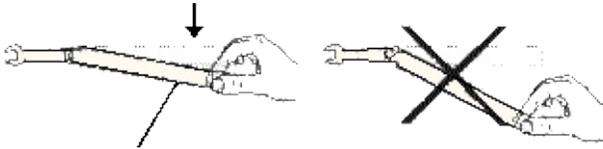
To Use a Torque Wrench

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 M Ω 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](#)

[See other Amplifier Parameters topics](#)

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.

- $0 \text{ dBm} = 1 \text{ mW}$
- $-10 \text{ dBm} = 100 \text{ }\mu\text{W}$
- $+10 \text{ dBm} = 10 \text{ mW}$

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.

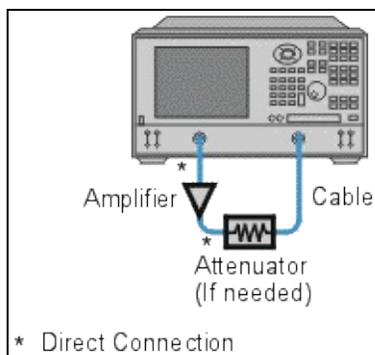
Attenuation of the amplifier's output power can be accomplished using either attenuators or couplers

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 unratiod power measurement (receiver B). [Learn how.](#)
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 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](#)

Other Tutorial topics

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:

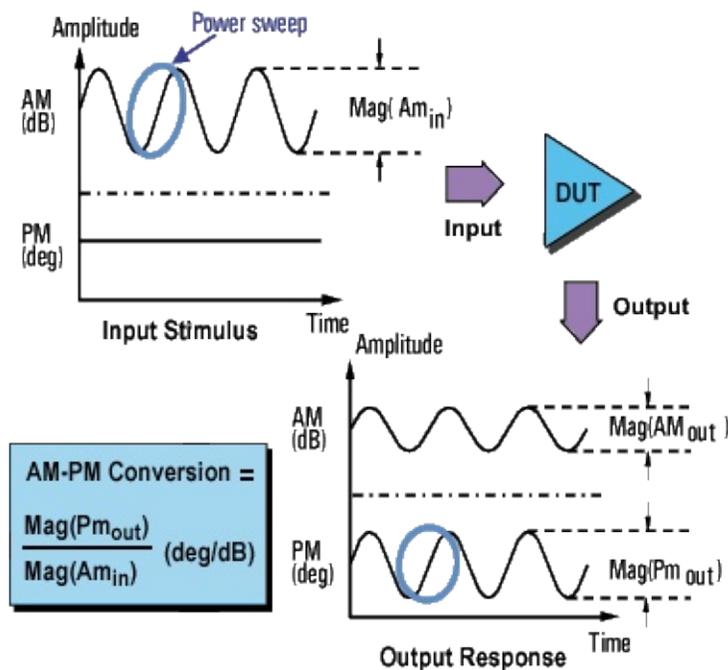
Unintentional amplitude variations (AM)

- 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 ($^{\circ}/\text{dB}$). An ideal amplifier would have no interaction between its phase response and the power level of the input signal.



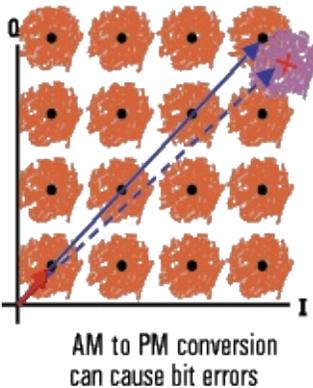
Why Measure AM-PM Conversion

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

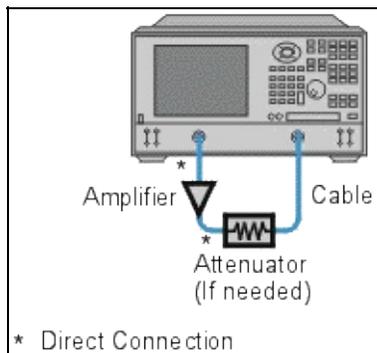
With this method of measuring AM-to-PM conversion, the modulation frequency is approximately the inverse of the sweep time. Even with the fastest power sweep available on most network analyzers, the modulation frequency ends up being fairly low (typically less than 10 Hz). This could cause a slight temperature 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 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.



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 S_{21} 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 \(SWR, \$\rho\$ \)](#)
- [Complex Impedance](#)
- [Gain Compression](#)
- [AM-to-PM Conversion](#)

See Also

- [High-Gain Amplifiers](#)

Gain

$$\tau = \frac{V_{\text{trans}}}{V_{\text{inc}}}$$
$$\text{Gain (dB)} = -20 \log_{10} |\tau|$$
$$\text{Gain (dB)} = P_{\text{out}} \text{ (dBm)} - P_{\text{in}} \text{ (dBm)}$$

The ratio of the amplifier's output power (delivered to a Z_0 load) to the input power (delivered from a Z_0 source). Z_0 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

$$\begin{aligned}\tau_g (\text{sec}) &= - \frac{\Delta \theta}{\Delta \omega} \\ &= - \frac{1}{360} * \frac{\Delta \theta}{\Delta f}\end{aligned}$$

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, ρ)

$$\begin{aligned}\Gamma &= \frac{V_{\text{refl}}}{V_{\text{inc}}} = \rho \angle \theta \\ \text{Reflection coefficient} &= \rho \\ \text{Return loss (dB)} &= -20 \log_{10} \rho \\ \text{SWR} &= \frac{1+\rho}{1-\rho}\end{aligned}$$

The measure of the reflection mismatch at the input or output of the amplifier relative to the system Z_0 characteristic impedance.

Complex Impedance

$$Z = \frac{1+\Gamma}{1-\Gamma} * Z_0$$
$$= R + jX$$

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

See Gain Compression Application.

AM-to-PM Conversion Coefficient

$$\text{AM/PM} = \frac{\Delta \theta}{\Delta P}$$

The amount of phase change generated in the output signal of an amplifier as a result of an amplitude change of the input signal.

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 a Keysight Vector Network Analyzer (VNA) 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 VNA is set to a start frequency.
2. As the antenna moves, the VNA responds to each external trigger signal by measuring an antenna port.
3. When all ports are measured, the VNA increments to the next frequency
4. Again the VNA 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 VNA decrements in frequency until all ports are measured at all frequencies and the VNA is set back to the original start frequency.

Once setup, only external trigger signals are sent to the VNA. After each trigger, measurement data is stored in internal VNA memory.

How to set up the VNA

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

1. On the **Trace** menu click **New Trace**
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)
-

Balanced Measurements

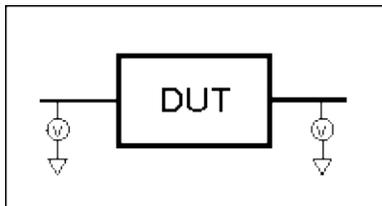
- What are Balanced Devices?
- Differential and Common Modes Model
- Measuring Mixed Mode (Balanced) S-Parameters
- Measuring Imbalance Parameters
- Measuring CMRR
- Port Mapping
- Calibrating Balanced Measurements
- How the analyzer makes Balanced Measurements

Other Measurement Setup Topics

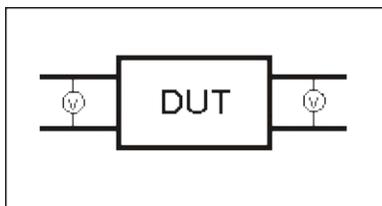
Check out the [Integrated True Mode Stimulus Application \(iTMSA\)](#).

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.



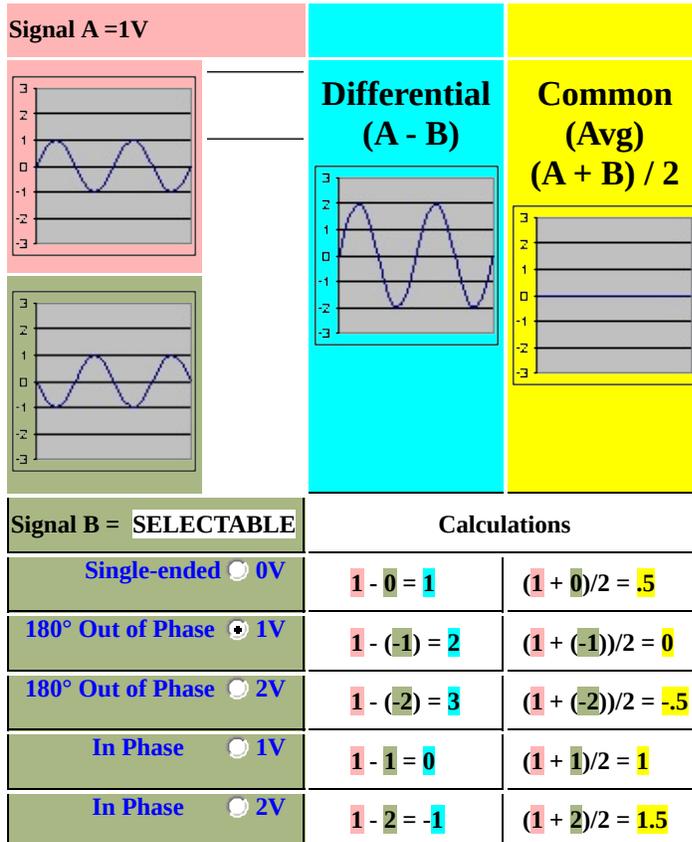
Differential and Common Modes Model

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 minus B**
- **Common** is calculated as the **AVERAGE of A and B**

Note: Click **Signal B** selections to see various Differential and Common signals.



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 analyzer, we would select an S_{dd21} measurement using the following balanced notation:

S_{abxy} -

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

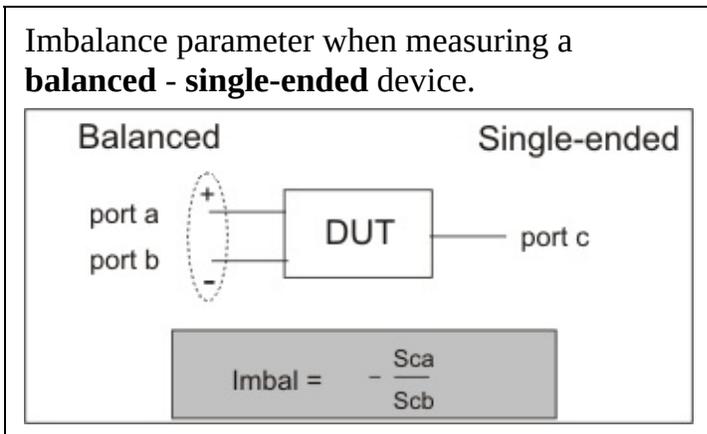
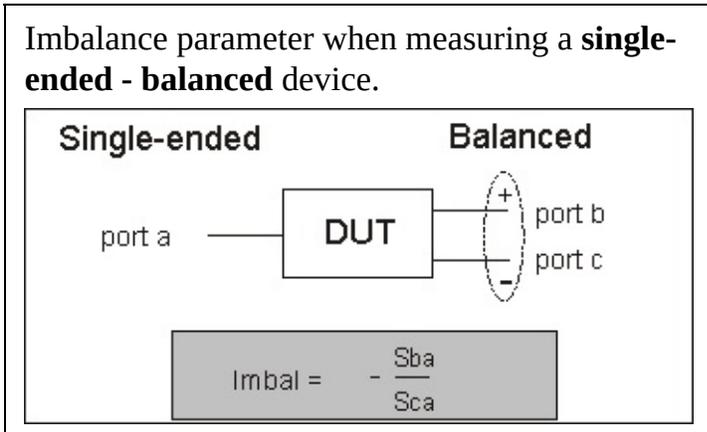
iTMSA

Measuring Imbalance Parameters

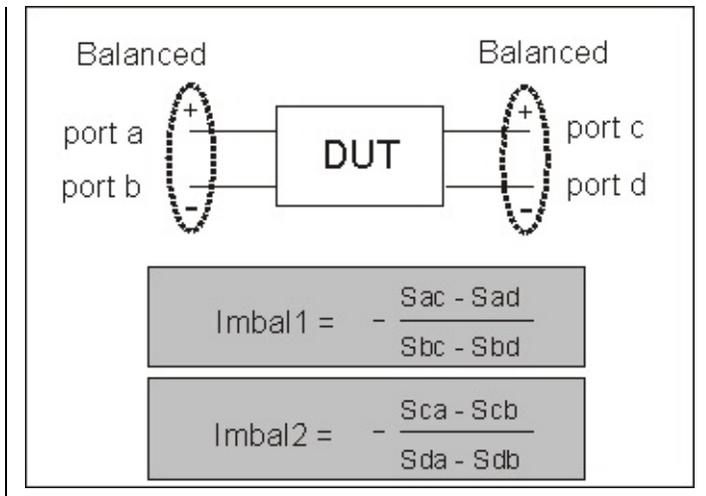
Imbalance is a measure of how well two physical ports that make up a balanced port are matched. With a perfectly balanced port, the same amount of energy flows to both ports and the magnitude of the ratio of these ports is 1.

The notation is similar to traditional S-parameters. In the following diagrams, the letters a, b, c, and d are used because any analyzer port can be assigned to any logical port using the **port mapping process**.

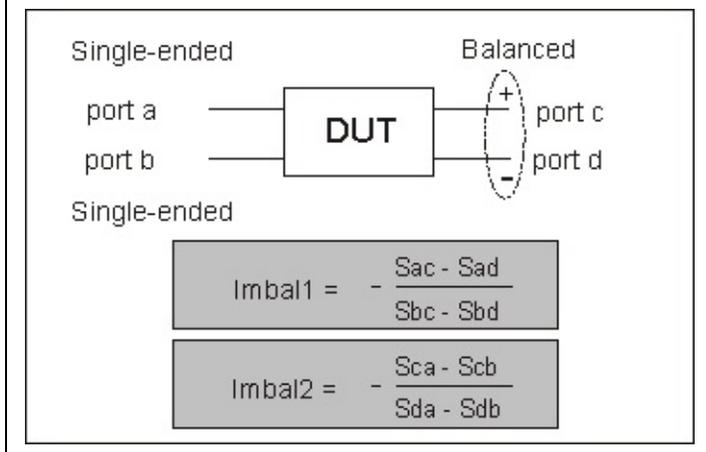
For example, in the following single-ended - balanced formula, **S_{ba}** indicates the device output port is logical port b and the input port is logical port a.



Imbalance1 and Imbalance2 parameters when measuring a **balanced - balanced** device.



Imbalance1 and Imbalance2 parameters when measuring a **single-ended - single-ended - balanced** device.



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.

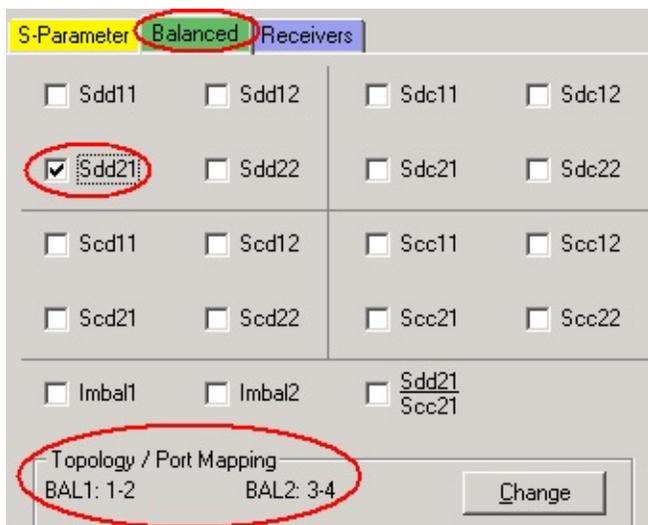
Single-ended - balanced device	Sds21	and	Ssd12
	-----		-----
	Scs21		Ssc12

Balanced - single-ended device	Ssd21 ----- Scs21	and	Sds12 ----- Scs12
Balanced - balanced device	Sdd21 ----- Scs21		
Single-ended - single-ended - balanced device	Sds31 ----- Scs31	and	Sds32 ----- Scs32

Device Topology and Port Mapping

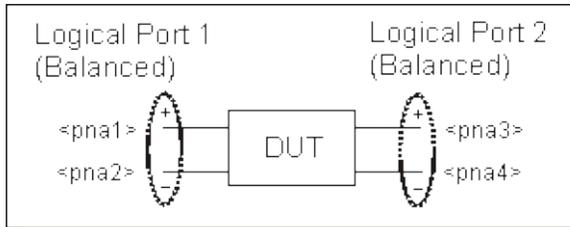
As we have seen on balanced inputs and outputs, the signal of interest is the difference or average of two BALANCED input or BALANCED output lines. It is also possible to have single-ended ports AND balanced ports on the same device. The two balanced input or output lines are referred to as a single "logical" port.

When configuring a balanced measurement on the analyzer, select a device 'topology'. Then map each test port to the DUT ports. The analyzer assigns "logical ports". [See how to set device topology.](#)

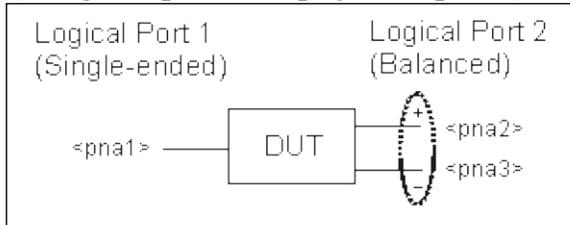


The following device topologies can be measured by a 4-port analyzer.

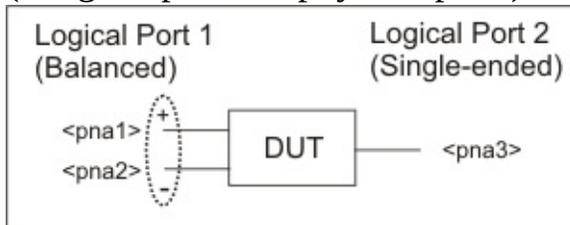
- **Balanced / Balanced**
(2 logical ports - 4 physical ports)



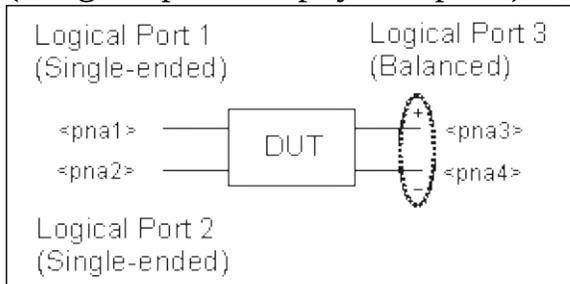
- **Single-ended / Balanced**
(2 logical ports - 3 physical ports)



- **Balanced / Single-ended**
(2 logical ports - 3 physical ports)



- **Single-ended - Single-ended / Balanced**
(3 logical ports - 4 physical ports)



These topologies can be used in the reverse (<==>) direction to measure:

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

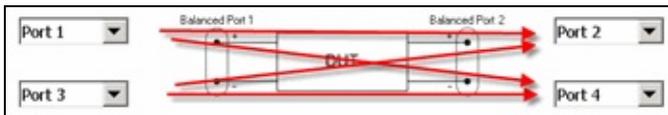
For example, to measure a **Balanced / Single-ended** topology, measure the S12 (reverse direction) of a **Single-ended / Balanced** topology.

Calibrating Balanced Measurements

Balanced measurements are calibrated in the same manner as single-ended (standard) measurements. However, for highest accuracy, you must choose Thru paths so that each **transmission path** of the balanced measurement is represented. For a Balanced/Balanced topology, this means that FOUR Thru connections should be made.

For example (see following image):

- Balanced Port 1 is ports 1 and 3
- Balanced Port 2 is ports 2 and 4
- Thru paths to be calibrated should be: 12, 14, 32, 34.
- Paths 13, and 24 are less important.



To select Thru paths:

1. From SmartCal, on the Select DUT Connectors and Cal Kits page, check **Modify Cal**.
2. Click **Next** to see the following Cal Wizard page:

Modify Cal				
	1st Port	2nd Port	Thru Cal Method	
Thru #1	1	2	Unknown Thru	Cal Type/Stds...
Thru #2	1	4	Unknown Thru	Cal Type/Stds...
Thru #3	3	4	Unknown Thru	Cal Type/Stds...
Thru #4	2	3	Unknown Thru	Cal Type/Stds...

How the analyzer makes Balanced Measurements

When using standard Balanced measurements, the analyzer does not provide true balanced measurements by stimulating both balanced inputs together and measuring both outputs relative to one another. Instead, the analyzer makes only Single-ended measurements. On a Balanced/ Balanced device, it stimulates each input and measures each output individually. From the output data, the analyzer calculates the Differential and Common outputs from the DUT using the same math formulas as the above model. However, all measurements and calculations 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.

Complex Impedance

When making an S_{11} or S_{22} 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 S_{11} or S_{22} 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 (Γ) 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 Γ corresponds uniquely to a complex device impedance (as a function of frequency), according to the equation:

$$Z_L = [(1 + \Gamma) / (1 - \Gamma)] \times Z_0$$

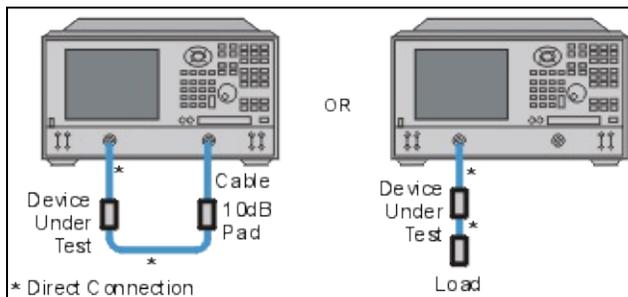
where Z_L is your test device impedance and Z_0 is the measuring system's characteristic impedance.

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:

$-d(\phi)/d(\omega)$ -- where **ϕ** is radian angle, and **ω** 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:

$-(1/360) * \Delta(\phi) / \Delta(f)$ -- where **ϕ** 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 Analyzer Delay Functions](#)

[See other Tutorials](#)

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?

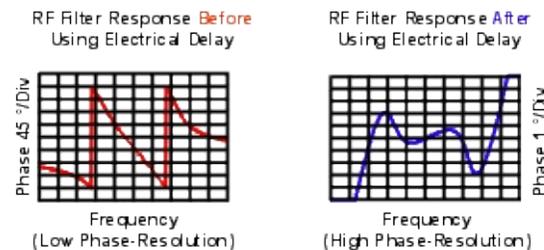
The deviation from linear phase measurement accomplishes the following:

- 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.



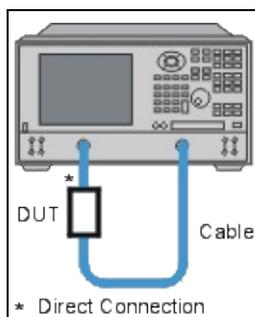
[Learn how to set Electrical Delay.](#)

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](#)

[See other Amplifier Parameter topics](#)

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

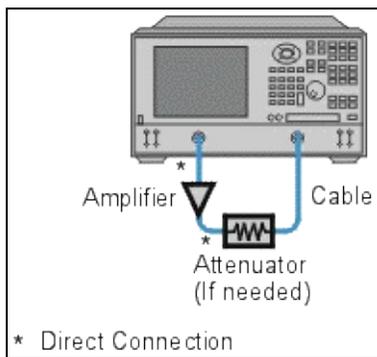
The frequency-response effects and mismatches of the attenuators and couplers must be accounted for during calibration since they are part of the test system. Proper error-correction techniques can reduce these effects.

- 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.

Gain Compression

Gain compression measures the level of input power applied to an amplifier that will cause a distorted output.

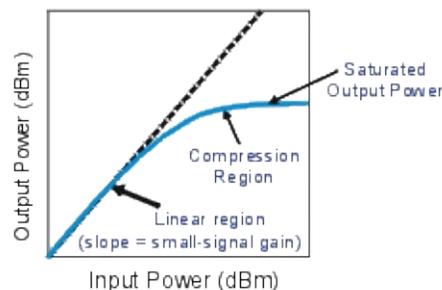
- [What Is Gain Compression?](#)
- [Why Measure Gain Compression?](#)
- [Accuracy Considerations](#)
- [How to Measure Gain Compression](#)

[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 signal.

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. 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

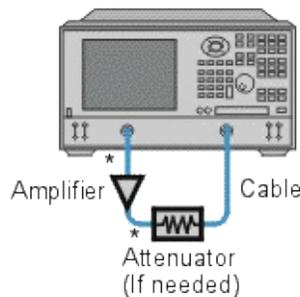
This procedure shows you how to make the following three measurements used to determine amplifier gain compression:

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 S_{21} 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. To reduce the effects of noise, you may want to specify a narrower IF bandwidth.



* Direct Connection

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 S_{21} 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 unratiod (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 Delay Functions](#).

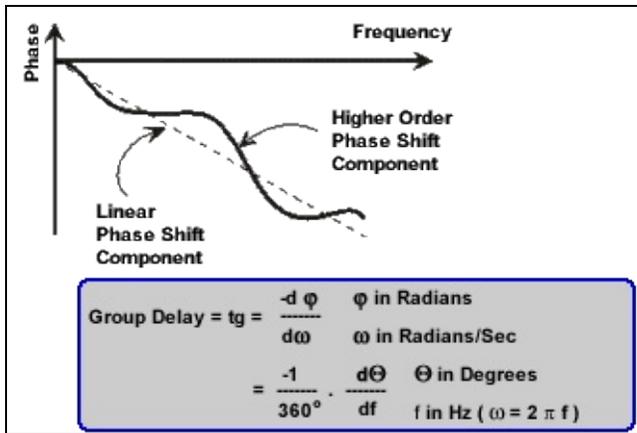
[See other Amplifier Parameter topics](#)

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.

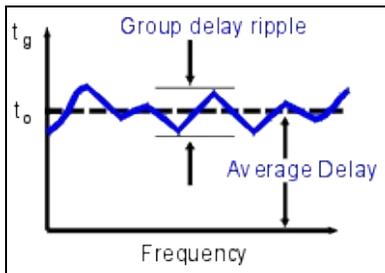
Refer to the graphic below for the following discussion:



The phase characteristic of a device typically consists of both linear and higher order (deviations from linear) phase-shift components.

Linear phase-shift component:	Higher-order phase-shift component:
Represents average signal transit time.	Represents variations in transit time for different frequencies.
Attributed to electrical length of test device.	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.

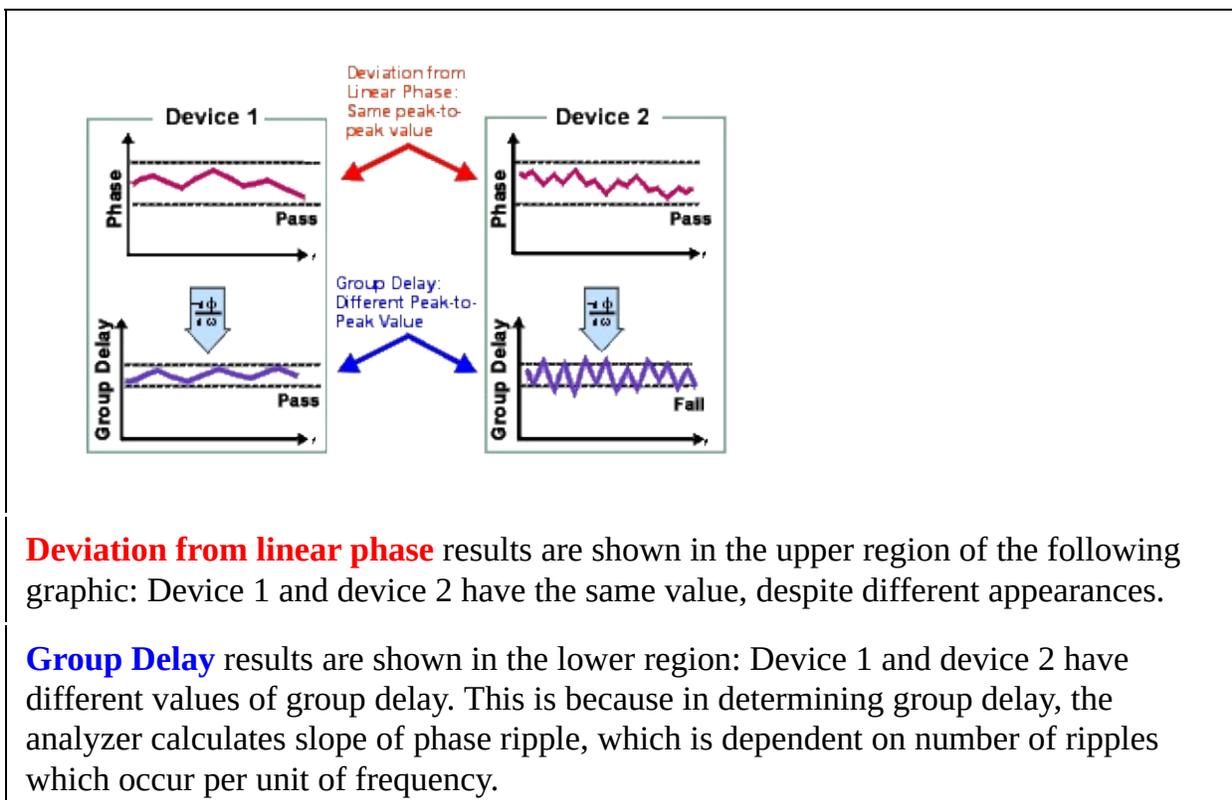
Refer to the following equation for this discussion on how group delay is calculated:

$$\begin{aligned} \text{Group Delay} = \text{tg} &= \frac{-d\phi}{d\omega} && \begin{array}{l} \phi \text{ in Radians} \\ \omega \text{ in Radians/Sec} \\ \Theta \text{ in Degrees} \\ f \text{ in Hz } (\omega = 2 \pi f) \end{array} \\ &= \frac{-1}{360^\circ} \cdot \frac{d\Theta}{df} \end{aligned}$$

- Phase data is used to find the phase change (-dφ).
- A specified frequency aperture is used to find the frequency change (dω).
- 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**.



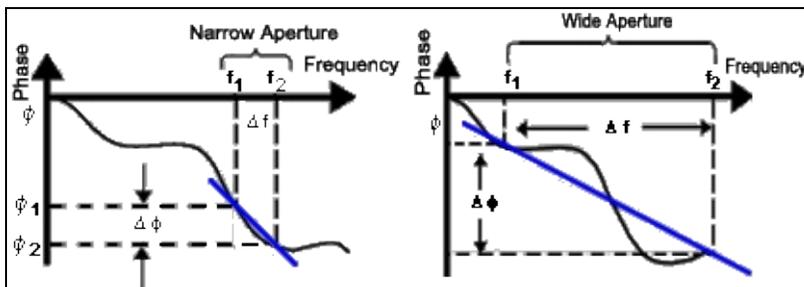
Deviation from linear phase results are shown in the upper region of the following graphic: Device 1 and device 2 have the same value, despite different appearances.

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, phase is measured 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 ($-\Delta\phi / \Delta f$) 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 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

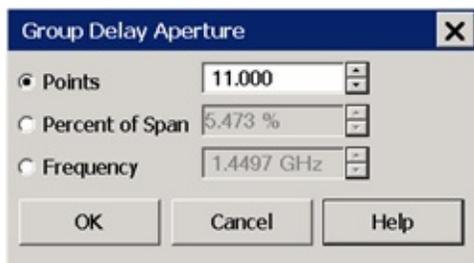
Using front-panel HARDKEY [softkey] buttons

1. Press **AVG**
2. then **[More]**
3. then **[Group Delay Aperture]**

Using Menus

1. Click **Response**
2. then **Avg**
3. then **Group Delay Aperture**

Programming Commands



Group Delay Aperture dialog box help

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 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 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.

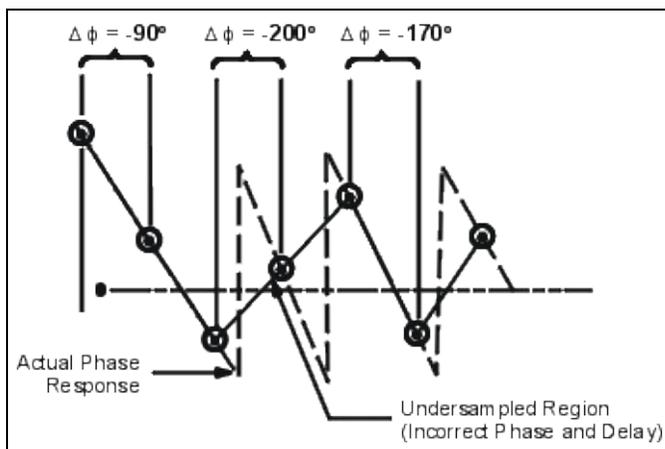
Cancel Closes the dialog. Setting changes are NOT applied.

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.

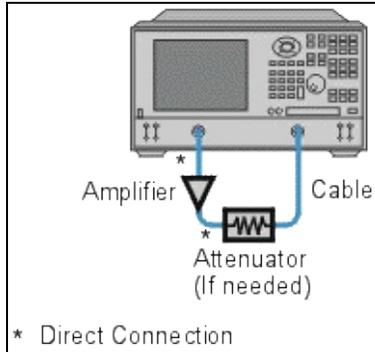
Particularly for an amplifier, the response may vary differently at various temperatures. The tests should be done when the amplifier is at the desired operating temperature.

How to Measure Group Delay

1. Preset the analyzer.
2. If your DUT is an amplifier, it may be necessary to adjust the 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 test port 2.

3. Connect the DUT as shown in the following graphic.



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.

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.

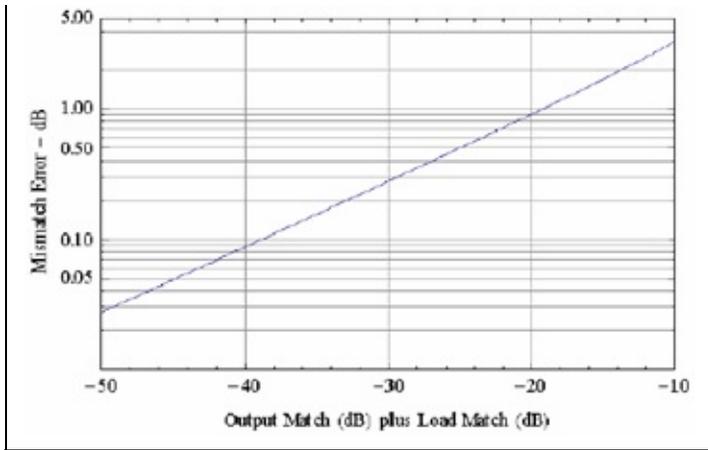
If you are using a large attenuator on port 2 (which improves output match), perform an Enhanced Response Calibration as follows. This corrects for the same errors as the full 2-port correction EXCEPT the interaction between the raw load match and the DUT output match.

1. There is NO need to Uncouple the port powers.
2. Set port powers to an acceptable level. Do NOT overpower the test port.
3. Perform Enhanced Response Cal. [Learn how](#). (Does not measure or correct for S12 or S22 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
--



Last Modified:

23-Sep-2009 MX New topic

Phase Measurements

Knowledge of both magnitude and phase characteristics is needed for successful higher-level component integration.

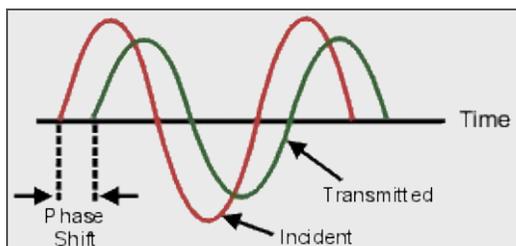
- [What are Phase Measurements?](#)
- [Why Measure Phase?](#)
- [Using the Analyzer's Phase Format](#)
- [Types of Phase Measurements](#)

[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.

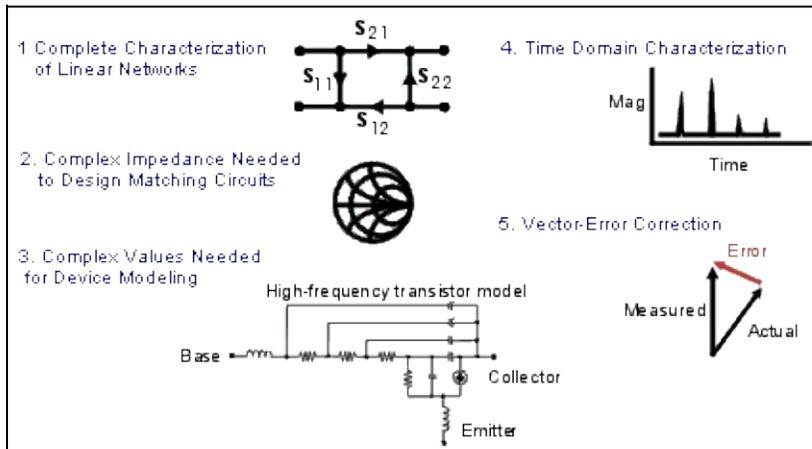
The following graphic shows the phase shift (in time or degrees) between an incident signal and a transmitted signal (as might be seen on an oscilloscope display).



Why Measure Phase?

Measuring phase is a critical element of network analysis. The following graphic

lists five reasons for measuring both magnitude and phase.



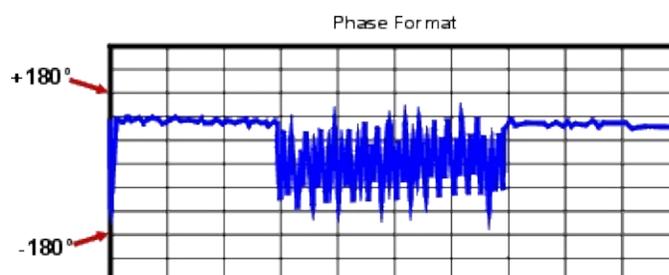
When used in communications systems to pass signals, components or circuits must not cause excessive signal distortion. This distortion can be:

- 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 ($^{\circ}/\text{dB}$).
- **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 Analyzer 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](#)

[See other Tutorials](#)

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 S_{12} .

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:

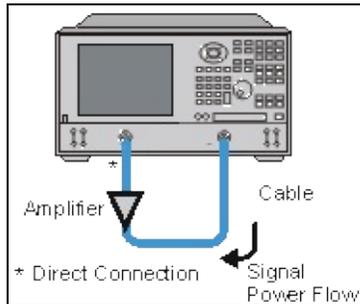
- Use or increase measurement averages.
- Reduce the IF bandwidth of the analyzer.

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.



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

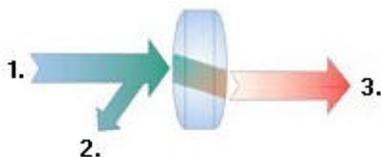
Reflection measurements are an important part of network analysis.

- [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

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

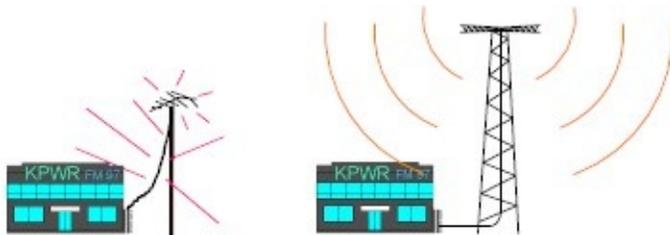
amplitude and phase information available at both the A and R channel. In S-parameter terminology, S11 is a reflection measurement of port1 of the device (the input port); S22 is a reflection measurement of the port 2 (the output port)

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

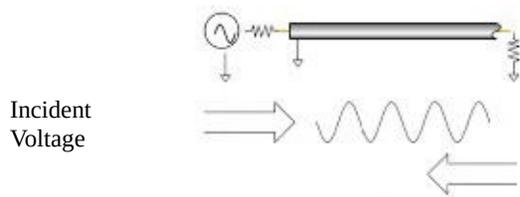
Another way of expressing reflection measurements is reflection coefficient gamma (Γ). Gamma includes both magnitude and phase.

The magnitude portion of gamma is called rho (ρ). Reflection coefficient is the ratio of the reflected signal voltage to the incident signal voltage. The range of possible values for ρ 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.

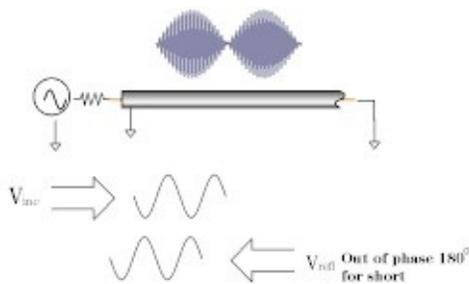


Z_0

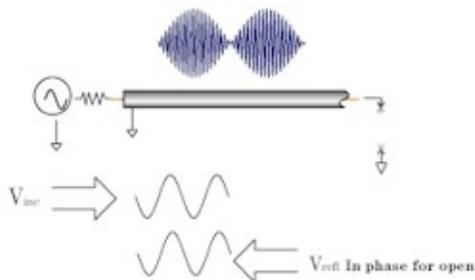
Reflected Voltage = 0

(All the incident power is absorbed in the load)

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.

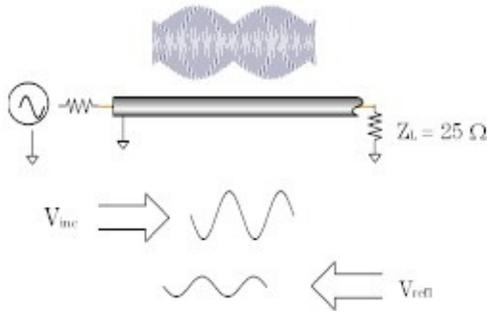


When a transmission line is terminated in an open circuit termination, all of the energy is reflected back to the source. The reflected wave is equal in magnitude to the incident wave ($\rho = 1$). However, no current can flow in an open circuit. Therefore, the voltage of the reflected wave will be in 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.

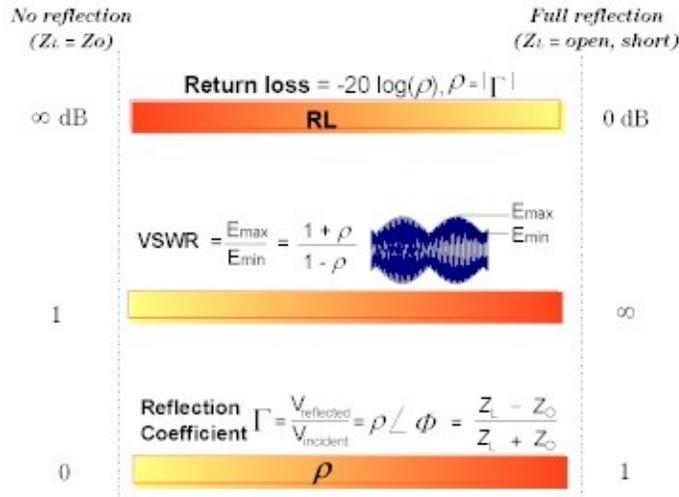


For more information, see [Phase Measurements](#).

Impedance

Impedance is another way of expressing reflection data. For more information on Impedance, see [Smith Charts](#).

Summary of the Expressions of Reflection Measurements:



Specifications - Select the Analyzer Model

Please select the Analyzer model of the specifications you would like to see.

PXI Specifications

The specifications are stored ONLY on the internet.

Doc Number	Model
M9370-90002	M9370A - 4 GHz
	M9371A - 6.5GHz
	M9372A - 9 GHz
	M9373A - 14 GHz
	M9374A - 20 GHz
	M9375A - 26.5 GHz

See the [equations that are used to generate uncertainty curves](#).

Glossary

A **B** **C** **D** **E** **F** **G** **H** **I** **J** **K** **L** **M**
N **O** **P** **Q** **R** **S** **T** **U** **V** **W** **X** **Y** **Z**

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: $10 \log_{10}$

(P_1/P_2) 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 $(-3\text{dB} + 10\text{dB} = +7\text{dB})$.

dBm Absolute unit of measure in decibels: $0\text{dBm} = 1 \text{ mW}$. 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.

E

ECal 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.

F

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.

G

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 Keysight 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: $f_1 \pm f_2$, $2xf_1 \pm f_2$, $2xf_2 \pm f_1$, $3xf_1 \pm 2x f_2$, 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 two 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.

J

K

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 Keysight 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.

O

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.

P

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

SA Spectrum Analyzer

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 10E6). For example, a stability of ± 5.0 ppm means that an analyzer will measure 1 MHz to an accuracy

of $\pm 5 \times 10^{-6} \times 10^6 \text{ Hz} = \pm 5 \text{ Hz}$.

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 $\pm 10 \text{ ppm}$ for $250 \text{ C} \pm 50 \text{ C}$.

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 $(1 + \pi) / (1 - \pi)$ where π 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 W), 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 (Keysight 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](#).

Quick Start

The following topics can help you become familiar with your analyzer:

- **Programming the M937xA**
- SFP Launcher
- Front Panel Tour
- Quick Start Measurements
- Front-panel Interface
- Traces, Channels, and Windows
- Basic Measurement Sequence
- Preferences
- Using Help

New | Delete | Meas Class | Manage Meas | Balanced Meas | Manage Channels

Hardware: External Test Set

Description	SCPI	
New		
Create S-Parameter Meas.	CALC:PAR:DEF:EXT	
Create Measurement	CALC:PAR:DEF:EXT	
Add Measurement	None	
List Measurements	CALC:PAR:CAT:EXT	
Delete		
Delete a measurement	CALC:PAR:DEL	
Delete ALL measurements	CALC:PAR:DEL:ALL	
Manage Measurements		
Get a handle to a Trace	None	
Select a Measurement	CALC:PAR:SEL	
Read Channel Number	SYST:ACTive:CHAN	
Read Channel Numbers in use.	SYST:CHAN:CAT?	
Read Number of Measurements	None	
Read Measurement Parameter	None	
Set / Read Measurement Name	SYST:ACTive:MEAS (read-only)	
Read Measurement Number	None	
Change Parameter	CALC:PAR:MOD:EXT	
Read the window number of the selected trace.	Calc:Par:WNUM	
Read the trace number of the selected trace.	Calc:Par:TNUM	
Maximize (Isolate) trace	DISPlay:TMAX	
Move a trace to another window	DISP:WIND:TRAC:MOVE	
Trace Hold	CALC:HOLD:TYPE	
Trace Hold Clear	CALC:HOLD:CLEAr	
All Measurement Classes / Applications		

Balanced Measurements and Fixturing		
Configure Topology		CALC:FSIM:BAL

Manage Channels

Add	None
Make Active	None
Read Channel Number	SYST:ACT:CHAN
Read UNUSED channel numbers	None
Read used channel numbers	None
Read number of Channels	None
Copy all Channel settings	SYST:MACR:COPY:CHAN
Copy ONLY mechanical switches and attenuator settings.	SENS:PATH:CONF:COPY
Delete a channel	SYSTem:CHANnels:DELet

External Testset Control (also for E5091A)

Load config file and Restart PNA .	SYST:CONFigure
Returns the test set ID number.	None
Switches an input to one of the valid outputs (E5091A only).	None
Reads a Cal Set for the Test Set model.	SENS:CORR:CSET:TSet:ALLPorts?
Reads a Cal Set for the Port Mapping.	SENS:CORR:CSET:TSET:TYPE?

Calibration Topics

Cal Basics

- [Calibration Wizard](#)
- [Select a Calibration](#)
- [Using Cal Sets](#)
- [Calibration Preferences](#)

To learn about calibrating Application channels, refer to the help topic for the Application.

Cal Types

- Using ECal
- Perform a 4-Port Cal with a 2-Port ECal Module
- TRL Cal
- Calibrate All Channels

Cal Concepts

- Calibration Overview
- About Calibration Standards
- Error Correction and Interpolation
- Calibration Thru Methods
- Accurate Calibrations
- Validity of a Calibration

Advanced Cal Topics

-
- [ECal User Characterization](#)
- [Port Extensions](#)
- [Fixture Simulator](#)
- [Swap Adapters Method](#)
- [Measurement Errors](#)
- [Modify Cal Kits](#)

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

[Measurements](#) | [Balanced Meas](#) | [Format](#) | [Scale](#) | [Elect Delay](#) | [Display](#) | [Avg](#) | [IFBW](#) | [Smoothing](#) | [Group Delay](#) | [Cal](#)
[Show](#) | [Hide: Status Bar](#) | [Toolbar](#) | [Tables](#) | [Meas Disp](#) | [Marker Disp](#) | [Windows](#) | [Colors](#)

Description	SCPI
Speed up Measurements !	
Measurement Trace On Off	DISP:WIND:TRAC
Display Update On Off	DISP:ENAB
Window Update On Off	DISP:WIND:ENABLE
Analyzer Visible On Off	DISP:VIS
Measurement display update	Calc:Par:Sel<name>,fast Calc:Par:Mnum <num>,fast

Balanced Measurements and Fixturing	
Configure Topology	CALC:FSIM:BAL
Configure Balanced Measurement	CALC:FSIM:BAL

Format	
Format trace	CALC:FORM

Scale	
AutoScale	DISP:WIND:TRAC:Y:AUTO
AutoScale All	DISP:WIND:Y:AUTO
Per Division	DISP:WIND:TRAC:Y:PDIV
Reference Level	DISP:WIND:TRAC:Y:RLEV
Reference Position	DISP:WIND:TRAC:Y:RPOS
Phase Offset	CALC:OFFS:PHAS
Magnitude Offset	CALC:OFFS:MAGN
Magnitude Slope Offset	CALC:OFFS:MAGN:SLOP

Scale Coupling	
Set method	DISP:WIND:TRAC:Y:COUP:METH
Enable window	DISP:WIND:TRAC:Y:COUP

Electrical Delay	
Electrical Delay	CALC:CORR:EDEL:TIME
Delay in distance	CALC:CORR:EDELay:DISTance
Set units for distance	CALC:CORR:EDELay:UNIT

Status Bar	
Status Bar On Off	DISP:ANN:STAT
Toolbars/ Title Bars	
Toolbars On Off	DISPlay:TOOLbar:ENTRy DISPlay:TOOLbar:MARKer DISPlay:TOOLbar:KEYS[:STATe] DISPlay:TOOLbar:EXT[:STATe] DISPlay:TOOLbar:TRAN[:STATe]
Title Bars On Off	None
Tables	
Tables On Off	DISP:WIND:TABLE
Measurement Display	
Trace Status buttons On Off	DISP:WIND:ANN:TRAC:STAT
X-axis values On Off	DISP:ANN:FREQ
Maximize (Isolate) trace	DISP:TMAX
Limit Test Results	CALC:LIM:STAT
Number of segments in a limit test	CALC:LIM:SEGM:COUN?
Limit Lines	CALC:LIM:DISP:STAT
Trace Title ON OFF	DISP:WIND:TRAC:TITL:STATe
Writes and reads trace title.	DISP:WIND:TRAC:TITL:DATA
Window Title On Off	DISP:WIND:TITL
Write Window Title	DISP:WIND:TITL:DATA
Set Grid to solid or dotted lines	DISP:WIND:TRAC:GRAT:GRID:LTYPE
System Clock On Off	SYST:CLOCK
Error Messages On Off (Help Menu)	DISP:ANN:MESS:STAT
Data/Memory Trace	

Memory Trace On Off	DISP:WIND:TRAC:MEM
Marker (readout) Display	
Marker Readout On Off	DISP:WIND:ANN:MARK:STAT
Marker Readout Size	DISP:WIND:ANN:MARK:SIZE
Readouts Per Trace	DISP:WIND:ANN:MARK:NUMB
Stimulus decimal places	DISP:WIND:ANN:MARK:RES:STIM
Response decimal places	DISP:WIND:ANN:MARK:RES:RESP
Readout position: X-axis	DISP:WIND:ANN:MARK:XPOS
Readout position: Y-axis	DISP:WIND:ANN:MARK:YPOS
Marker symbol	DISP:WIND:ANN:MARK:SYMB
Display marker symbols above trace or not	DISP:WIND:ANN:MARK:SYMB:ABOVe[STATe]

Windows	
Add a Window	DISP:WIND
Return Window Number(s)	DISP:CATalog?
Activate a Window	None
Arrange Measurement Windows	DISP:ARRange
Analyzer Window (Max Min Normal)	DISPlay:WIND:SIZE
Read the window number of the selected trace.	Calc:Par:WNUM

Display and Print Colors	
Active Window Background	DISP:COL:ABACkground
Set background color	DISP:COL:BACK
Set labels and grid frame colors	DISP:COL:GRAT1
Set inner lines of all grids in all windows colors	DISP:COL:GRAT2
Set Inactive window label colors	DISP:COL:ILAB
Set limit line colors	DISP:COL:LIM1
Set trace data and Limit Line colors	DISP:COL:TRAC:DATA
Set data trace marker colors	DISP:COL:TRAC:MARK

Set memory trace colors	DISP:COL:TRAC:MEM
Set memory trace marker colors	DISP:COL:TRAC:MMAR
Load a color theme	DISP:COL:LOAD
Saves the current color theme.	DISP:COL:STOR
Resets to the default PNA colors.	DISP:COL:RES

Averaging	
Average ON OFF	SENS:AVER
Average Factor	SENS:AVER:COUN
Return the Average Count	None
Enable Point Averaging	SENS:AVER:MODE
Average Restart	SENS:AVER:CLE

IF Bandwidth	
IF Bandwidth	SENS:BWID
Previous IF Bandwidth	None
Next IFBandwidth	None
Reduce IF BW	SENS:BWID:TRACk

Smoothing	
Smoothing ON OFF	CALC:SMO
Smoothing Aperture	CALC:SMO:APER
Smoothing Points	CALC:SMO:POIN

Group Delay Aperture	
Set aperture using frequency	CALC:GDELay:FREQ
Set aperture using percent of span	CALC:GDELay:PERCent
Set aperture using fixed number of points	CALC:GDELay:POINts
Set Preference to 2 points	SYST:PREF:ITEM:GDEL:TWOP

Set Up Measurements

Set up your measurement using the following information:

Basic Settings

- [Preset the Analyzer](#)
- [Select a Measurement Parameter](#)
- [Set Frequency Range](#)
- [Set Power Level](#)
- [Set Sweep Type](#)
- [Set Number of Points](#)
- [Set Triggering](#)
- [Set Data Format](#)
- [Set Scale](#)
- [Pre-configured Measurement Setups](#)
- [Customize the Screen](#)
- [Undo/Redo Settings](#)

Advanced Settings

- [Copy Channels](#)

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

Optimize Measurements

A measurement has many interdependent settings. You can modify the settings to achieve the goals of your measurement application: faster throughput or greater measurement accuracy.

Increase Measurement Throughput

- [Achieve Fastest Sweep](#)
- [Switch Between Multiple Measurements](#)
- [Data Transfer Speed](#)
- [Using Macros](#)
- [Speed up PXIe Measurements](#)

Improve Measurement Accuracy

- [Increase Dynamic Range](#)
- [Reduce Noise](#)
 - [Averaging](#)
 - [IFBW](#)
 - [Smoothing](#)
- [Group Delay \(Separate Topic\)](#)
- [Improve Phase Measurement Accuracy](#)
- [Improve Accuracy for Electrically Long Devices](#)
- [Improve Reflection Accuracy on Low-Loss 2-Port Devices](#)
- [Increase Measurement Stability](#)
- [Decrease Receiver Crosstalk](#)
- [Reduce the Effects of Test Accessories](#)

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

Analyze Data

- Markers
- Math / Memory Operations
- Equation Editor
 - Import Functions
- Limit Lines

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

[Save /Recall](#) | [Manage Files](#) | [Manage Folders](#) | [Print](#) | [Read Clock Preset \(User\)](#) | [Security](#) | [Configure](#)

[Hdwr & Capabilities](#) | [Macros](#) | [Status](#) | [GPIB Pass-through](#) | [VISA Pass-through](#) | [Preferences](#)
[LXI](#) | [Error Messages](#)

Preset / User	SCPI
Preset	SYST:PRES *RST
Preset plus delete window	SYST:FPRreset
Security	
Frequency Blanking	SEC:LEVel
Configure	
Local Lockout	Local Lockout
Set and return GPIB address	None
Set PNA to GPIB system controller or talker/listener	None
Set and return SICL address	None
Control the PNA via SICL	None
Return Full computer name	None
System Impedance	SENS:CORR:IMP:INP:MAGN
Load Test Set Config file and Restart PNA.	SYST:CONFigure
Hardware	Trace/Chan Menu
IO Configuration	None



--

Hardware and Capabilities	
DSP Revision	SYST:CONF:REVision:DSP?
DSP FPGA	SYST:CONF:REVision:DSPFpga?
CPU Speed	SYST:CONF:REVision:CPU?
Hostname	SYST:COMM:LAN:HOSTName?
Many queries regarding the capability of a specific VNA	SYSTem:CAPability
Macros	
Execute Macro	SYSTem:SHORtcut:EXECute
Delete Macro	SYSTem:SHORtcut:DELeTe
Write macro path, argument, and title	SYSTem:SHORtcut:PATH SYSTem:SHORtcut:ARGuMents
Read macro path, argument, and title	SYSTem:SHORtcut:TITLe
Status Commands	
Status Registers	GP-IB/Status
*OPC;*WAI	GP-IB/Common_Commands

GPIB Pass Through	
Open a GPIB pass-through session	SYST:COMM:GPIB:RDEV:OPEN
Write string data to the GPIB pass-through device.	SYST:COMM:GPIB:RDEV:WRIT

Write data to the GPIB pass-through device - with header.	SYST:COMM:GPIB:RDEV:WBLock
Write data to the GPIB pass-through device - without header.	SYST:COMM:GPIB:RDEV:WBINary
Reads string data from the GPIB pass-through device.	SYST:COMM:GPIB:RDEV:READ?
Closes a GPIB pass-through session	SYST:COMM:GPIB:RDEV:CLOS
Closes ALL GPIB pass-through sessions	SYST:COMM:GPIB:RDEV:RES

VISA Pass Through

Returns list of visa address strings or aliases.	SYST:COMM:VISA:RDEV:FIND?
Sets timeout value for VISA pass-through commands.	SYST:COMM:VISA:RDEV:TIMeout

Preferences

Reset Preference default settings	SYST:PREF:DEF
Touchscreen ON Off	SYST:TOUC
Selected trace is wider.	None
Selected trace changes width briefly.	None
Cal: Auto-save User Cal Set	SENS:CORR:PREF:CSET:SAVE
Cal: For Guided Cal, set external trigger.	SENS:CORR:PREF:TRIG:FREE
Cal: For Unguided Cal, set external trigger.	SENS:CORR:PREF:TRIG:FREE
Cal: Simulated Cal Behavior	SENS:CORR:PREF:SIMCal
Cal: Use Primary FOM (for mmWave)	SENS:CORR:PREF:CAL:RANG

Memory: Sets and reads the state of the memory data interpolation default preference	SYST:PREF:ITEM:MINterpolate
Memory: Data Math 8510 Mode	None
Meas: Mathematical offset for receiver attenuation	SYST:PREF:ITEM:OFFS:RCV
Meas: Mathematical offset for source attenuation	SYST:PREF:ITEM:OFFS:SRC
Meas: RF power On during frequency sweep retrace	SYST:PREF:ITEM:RETR:POW
Meas: Power Sweep Retrace	SYST:PREF:ITEM:PSRT
Meas: External Trigger OUT is Global	TRIG:PREF:AIGL
Meas: Port 1 Noise Tuner Switch state	SYST:PREF:ITEM:SWIT:DEF
Meas: Draw failed trace segments in red	SYST:PREF:ITEM:RTOF
Meas: Draw limit lines in red	SYST:PREF:ITEM:REDL
Marker: Programming treats Mkr 10 as Reference	SYST:PREF:ITEM:REFM
Markers: On Preset, Coupled Markers is ON	SYST:PREF:ITEM:MCPreSet
Markers: On Preset, Coupling Method is Channel	SYST:PREF:ITEM:MCMMethod
Markers: Coupling controls on off state of markers.	SYST:PREF:ITEM:MCControl
Report source unlevelled events as errors	SYST:ERR:REP:SUNL
Ext Device: de-activate on PRESET and recall.	SYST:PREF:ITEM:EDEV:DPOL
On PRESET set two-point group delay aperture	SYST:PREF:ITEM:GDEL:TWOP

On PRESET always turn power ON	SYST:PREF:ITEM:PRES:POWer
On PRESET show Quick Start dialog	SYST:PREF:ITEM:QStart
Report when receiver is overloaded	SYST:PREF:ITEM:REC:CERR
Turn source power OFF when receiver is overloaded	SYST:PREF:ITEM:REC:OVER:POW
More buttons	
Define Data Saves	See Files
User Preset	See Preset
User Key	None
Printer Page Setup	Hardcopy
Millimeter settings	None
Power Limit	See Power Limits
Display and Print Colors	See Display

LXI	
Returns Structured status of the PNA networking configuration.	None
Returns string status of the PNA networking configuration.	None
Resets the PNA LAN configuration.	None
Modifies settings of the PNA computer networking configuration.	None
Displays the LAN Status dialog with LAN Status Indicator showing IDENTIFY.	LXI:IDEN

Error Messages

Enable the display of Error Messages

DISP:ANN:MESS:STAT

Timed vs Dialog messages

None

**Markers: Basics | Ref | Advanced | Function | Search: Bandwidth | Compression | PSAT | PNOP
Math | Limits | PassFail | Statistics | Gating | Transform | Eq Editor**

Description	SCPI	
Basics		
ON OFF	CALC:MARK	
Delete All Markers	CALC:MARK:AOFF	
Delete Marker	CALC:MARK	
Delta Marker	CALC:MARK:DELT	
Viewing Marker readouts	Display	
Get a handle to Ref marker	N/A	
Reference Marker		
Reference Marker On Off	CALC:MARK:REF	
Set and read X-axis location	CALC:MARK:REF:X	
Read Y-axis location	CALC:MARK:REF:Y?	
Advanced Settings		
Interpolate All Markers (Discrete)	None	
Interpolate Individ. Marker (Discrete)	CALC:MARK:DISC	
Type (Normal Fixed)	CALC:MARK:TYPE	
Format All Markers	None	
Format Individ. Marker	CALC:MARK:FORM	
Coupled Markers	CALC:MARK:COUP	
Coupled Markers Method	CALC:MARK:COUP:METH	
Read/Set Data Point number	CALC:MARK:BUCKet	
Read/Set X-axis value	CALC:MARK:X	
Read/Set Y-axis value	CALC:MARK:Y	
Function		
Marker=> Span	CALC:MARK:SET	
Marker=> Center (Freq)	CALC:MARK:SET	
Marker=> CW Freq and change sweep type	CALC:MARK:SET	
Marker=> Start (Freq)	CALC:MARK:SET	
Marker=> Stop (Freq)	CALC:MARK:SET	

Marker=> Elect. Delay	CALC:MARK:SET
Marker=> Ref. Level	CALC:MARK:SET
Marker=> CW Freq - No sweep type change	CALC:MARK:SET

Marker Search	
Execute Search	CALC:MARK:FUNC:EXEC
Select Search Function	CALC:MARK:FUNC
Maximum	CALC:MARK:FUNC
Minimum	CALC:MARK:FUNC
Target (Value)	CALC:MARK:TARG
Excursion Value	CALC:MARK:FUNC:APE:EXC
Threshold Value	CALC:MARK:FUNC:APE:THR
Assign Marker to Domain	CALC:MARK:FUNC:DOM:USER
Domain Range Start	CALC:MARK:FUNC:DOM:USER:START
Domain Range Stop	CALC:MARK:FUNC:DOM:USER:STOP
Tracking	CALC:MARK:FUNC:TRAC

Bandwidth Marker Search	
Bandwidth (Target)	CALC:MARK:BWID
Search Filter Bandwidth	CALC:MARK:BWID
Read Filter BandWidth	CALC:MARK:BWID
Read Filter Center Freq	CALC:MARK:BWID
Read Filter Loss	CALC:MARK:BWID
Read Filter Q	CALC:MARK:BWID

Compression Marker Search	
Compression Marker level found.	CALC:MARKer:COMP:LEVel
Read Compression Marker Input power	CALC:MARKer:COMP:PIN
Read Compression Marker Output power	CALC:MARKer:COMP:POUT
Search function	CALC:MARKer:FUNC:SEL
Execute function	CALC:MARKer:FUNC:EXEC

PSAT Marker Search

Initiate a PSAT search	CALC:MARK:PSAT:BACKoff
Set and read PSAT backoff	CALC:MARK:PSAT:BACKoff
Read PSat Out	CALC:MARK:PSAT:POUT?
Read PSat In	CALC:MARK:PSAT:PIN?
Read PMax Out	CALC:MARK:PSAT:POUT:MAXimum?
Read PMax In	CALC:MARK:PSAT:PIN:MAXimum?
Read Gain Sat	CALC:MARK:PSAT:GAIN?
Read Gain Max	CALC:MARK:PSAT:GAIN:MAXimum?
Read Gain Linear	CALC:MARK:PSAT:GAIN:LINear?
Read Comp Sat	CALC:MARK:PSAT:COMP:SAT?
Read Comp Max	CALC:MARK:PSAT:COMP:MAX?

PNOP Marker Search

Initiate a PNOP search	CALC:MARK:PNOP:BACKoff
PNOP backoff	CALC:MARK:PNOP:BACKoff
PNOP Power Offset	CALC:MARK:PNOP:POFFset
Read Pnop Out	CALC:MARK:PNOP:POUT?
Read Pnop in	CALC:MARK:PNOP:PIN?
Read Pnop Gain	CALC:MARK:PNOP:GAIN?
Read Pnop Comp	CALC:MARK:PNOP:COMPression?
Read PMax Out	CALC:MARK:PNOP:POUT:MAXimum?
Read PMax In	CALC:MARK:PNOP:PIN:MAXimum?
Read Gain Max	CALC:MARK:PNOP:GAIN:MAXimum?
Read Comp Max	CALC:MARK:PNOP:COMPression:MAXimum?
Read PBO Out	CALC:MARK:PNOP:BACKoff:POUT?
Read PBO In	CALC:MARK:PNOP:BACKoff:PIN?
Read PBO Gain	CALC:MARK:PNOP:BACKoff:GAIN?

Math	
Data Trace ON OFF	DISP:WIND:TRAC
Memory Trace ON OFF	DISP:WIND:TRAC:MEM
Data =>Memory	CALC:MATH:MEM
Trace Math (Add Sub Mult Div)	CALC:MATH:FUNC
Sets and reads the state of the memory data interpolation	CALC:MATH:INTerpolate

Limit Lines	
Display Lines ON OFF	CALC:LIM:DISP:STAT
Fail Sound ON OFF	CALC:LIM:SOUN
Testing ON OFF	CALC:LIM:STAT
Limit Test Failed	CALC:LIM:FAILed?
Count Limit Lines	None
Read Test Results	GP-IB_Command_Finder/Status
Set / Read entire Limit Line	CALC:LIM:DATA
Limit Line Type (Max Min)	CALC:LIM:SEGM:TYPE
Begin Stimulus	CALC:LIM:SEGM:STIM:STAR
End Stimulus	CALC:LIM:SEGM:AMPL:STOP
Begin Response	CALC:LIM:SEGM1:AMPL:STAR
End Response	CALC:LIM:SEGM1:AMPL:STOP
Show Limit table	DISP:WIND:TABLE
Sets and returns the X-axis position of the Limit Line Pass/Fail indicator	DISP:WIND:ANN:LIM:XPOSITION
Sets and returns the Y-axis position of the Limit Line Pass/Fail indicator	DISP:WIND:ANN:LIM:YPOSITION
Global Pass/Fail	
Show / hide the pass/fail dialog.	DISP:FSIG

Statistics	
Statistics ON OFF	CALC:FUNC:STAT
Statistics Range	CALC:FUNC:DOM:USER
Domain Range Start	CALC:FUNC:DOM:USER:STAR
Domain Range Stop	CALC:FUNC:DOM:USER:STOP

Set Type (Pk-Pk StdDev Mean)	CALC:FUNC:TYPE
Get All Statistics Data	CALC:FUNC:DATA?
Get Standard Deviation	CALC:FUNC:DATA?
Get Mean	CALC:FUNC:DATA?
Get Peak to Peak	CALC:FUNC:DATA?

Transform	
Transform ON OFF	CALC:TRAN:TIME:STAT
Mode (LowPass, BandPass)	CALC:TRAN:TIME
Start Time	CALC:TRAN:TIME:STAR
Stop Time	CALC:TRAN:TIME:STOP
Center	CALC:TRAN:TIME:CENT
Span	CALC:TRAN:TIME:SPAN
Step Rise Time	CALC:TRAN:TIME:STAR
Set Low Pass Frequency	CALC:TRAN:TIME:LPFR
Coupling	
Enable trace coupling	SENS:COUP:PAR
Set transform coupling parameters	CALC:TRAN:COUP:PAR
Distance Markers	
Specify measurement type for distance markers	CALC:TRAN:TIME:MARK:MODE
Specify units for distance markers	CALC:TRAN:TIME:MARK:UNIT
Set and return marker distance value	CALC:MARK:DIST

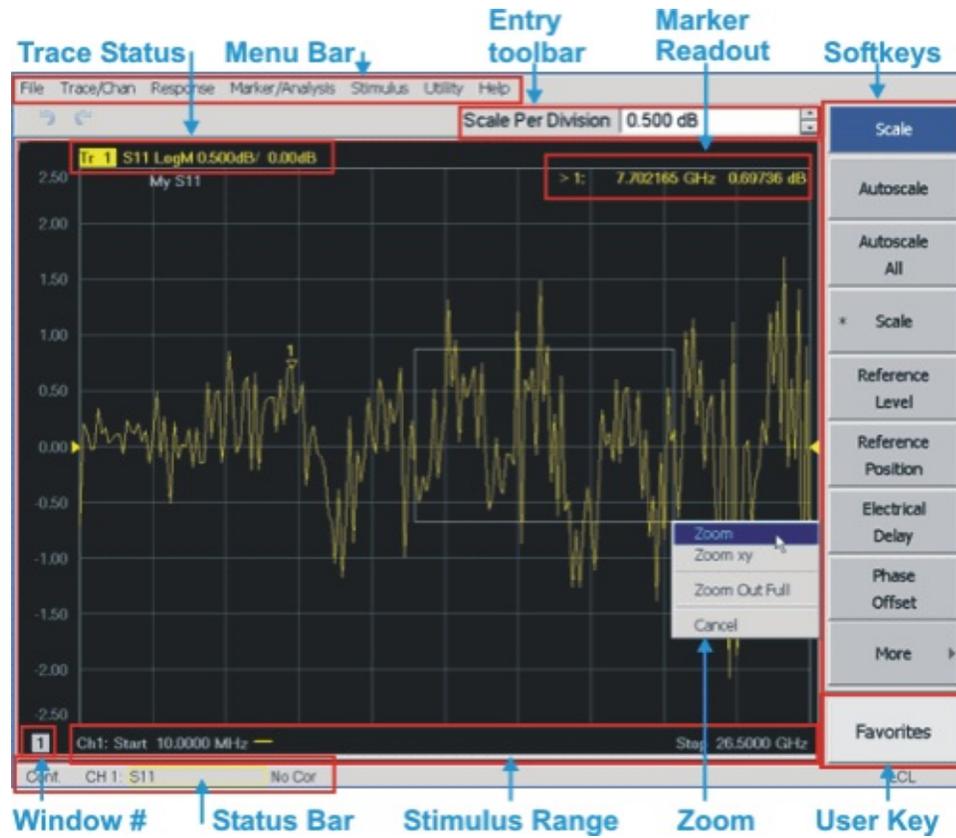
Gating	
ON OFF	CALC:FILT:TIME:STAT
Type (BandPass, Notch)	CALC:FILT:TIME
Shape	CALC:FILT:GATE:TIME:SHAP
Start	CALC:FILT:TIME:STAR
Stop	CALC:FILT:GATE:TIME:STOP
Center	CALC:FILT:GATE:TIME:CENT
Span	CALC:FILT:GATE:TIME:SPAN

Set gate coupling parameters	CALC:FILT:COUP:PAR
Window	
Kaiser Beta	CALC:TRAN:TIME:KBES
Impulse Width	CALC:TRAN:TIME:IMP:WIDT

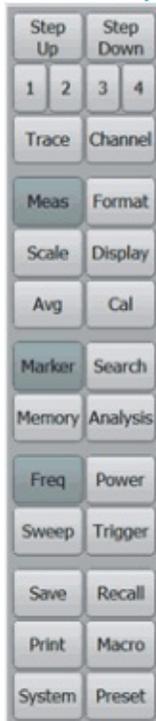
Equation Editor	
Turn ON / OFF equation	CALC:EQUation:STATe
Set equation	CALC:EQUation:TEXT
Return validity of equation	CALC:EQUation:VALid?
Returns the functions in DLL.	CALC:EQU:LIBR:FUNC
Imports the functions in DLL	CALC:EQU:LIBR:IMPorT
Is DLL Imported?	CALC:EQU:LIBR:IMPorT?
Remove a DLL	CALC:EQU:LIBR:REM

Screen Tour

Click on image areas to learn more.



Hardkeys



See Also

- [Marker Drag](#)
- [Expanded Mouse capabilities](#)
- [Learn how to Customize the Screen](#)

Using front-panel HARDKEY [softkey] buttons

Using a mouse with PNA Menus

1. Press **SYSTEM**
2. then [**Touchscreen**]
3. then [**Calibrate Touchscreen**]
or
[**Touchscreen ON | off**]

1. Click **Utility**
2. then **System**
3. then **Touchscreen**
4. then **Calibrate Touchscreen**
or
Touchscreen ON | off

Trace Status

Provides details of each trace in the window. Highlighted trace indicates the active trace. [Learn more.](#)

Menu Bar

The mouse-driven menu closely follows the menu structure of the front-panel hardkey/softkey menus. See [Hardkeys](#) to learn more about the logic of both menu structures.

Undo / Redo Icons

Allows you to quickly undo and redo selected operations. [Learn more](#)

Entry Toolbar

Along with the softkeys, allows numeric values to be entered for settings. Learn about all [toolbars](#).

Marker Readout

Provides stimulus and response information for markers. Learn about [customizing the marker readout area](#). See also [Marker Drag](#).

Window Number

Provides window identification which is useful for remote programmers. only.

Status Bar

Provides detail about all aspects of the status of the analyzer. [Learn more.](#)

Stimulus Range

Displays the start and stop values of the sweep range.

Zoom

Allows you to easily change the start and stop frequencies or start and stop power levels in a power sweep. [Learn more.](#)

See Also: [Search Within](#)

Marker Drag

Drag a displayed marker across the trace using a mouse. [Learn more.](#)

Expanded Mouse Capabilities

- Cursor changes to a “hand” when hovering over a clickable object.
- Right-click on the Entry toolbar to launch a mouse-compatible numeric pad.

Windows

- Right-click on a window area to make selections pertaining to that window.
- Double-click on a window area to maximize the window. To return to original window configuration, right-click on window area, then click **Tile**.
- Left-click on **X-axis** annotation to select the active channel/trace.
- Right-click on **X-axis** annotation to change stimulus properties. Applications are not fully supported.
- Quickly change Scale, Reference Level, and Position. [Learn how.](#)
- Right-click on **Y-axis** annotation to change Scale.
- Drag a trace from one window to another. Click or touch either the trace or the [Trace Status](#). Drag the trace to another window, then release the mouse or lift your finger.

Traces

- Left-click a trace or [Trace Status](#) to make it the selected trace.
- Double-click on a trace or Trace Status to maximize the trace. Double-click again to return to the original trace configuration. [See programming commands.](#)
- Set a [preference](#) to **always** widen the active trace.

- Set a **preference** to **briefly** widen the active trace.
- Drag a trace from one window to another. Click or touch either the trace or the **Trace Status**. Drag the trace to another window, then release the mouse or lift your finger.

Markers

- Right-click on a trace or Trace Status to add a marker.
- Right-click a marker to make selections pertaining to that marker, such as Marker Search or Function.

Simulated Hardkeys

- Click **Utility**, then **System**, then Keys toolbar to show and hide the simulated hardkeys.
- These keys would simulate the front panel hardkeys of a standard instrument.
- The combination of hard keys and softkeys allow easy access to ALL analyzer features.

Softkeys

- One additional softkey is reserved as a **User Key** which is always visible. [Learn more.](#)
- To Hide the softkeys to make more room for measurement windows, right-click any softkey.
- To Show the softkeys, press any front-panel hardkey that launches a softkey menu.
- The Hide/Show choice is reset to the default (Show) only on analyzer startup (NOT on Preset).

Softkey Annotations

Item	Description
	Selection launches a dialog box.

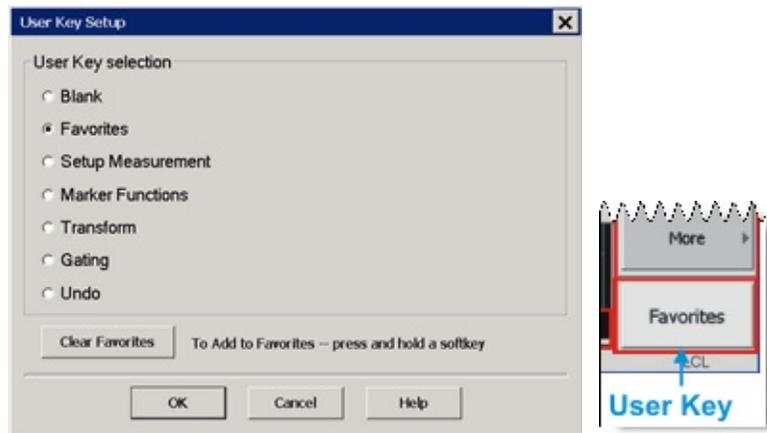
Menu ▶	Selection launches another level of softkeys.
Item ■	Indicates the item (marker, trace, window) is ON. Any number of objects can be ON.
Item □	Indicates the item (marker, trace, window) is OFF. Click to turn item ON.
Item ●	Indicates the item IS selected.
Item ○	Indicates the item is NOT selected. Click to select. Only one item in the collection can be ON.
* Item	Enter value in Entry toolbar.
Item on OFF	Capitalization indicates the current setting.
Return	Return to the previous softkeys.
More	Launches more of the same level softkeys.

User Key

Choose from several popular menus to assign to the User key. Menu structures for existing and future applications can also be selected as the User Key. The most flexible of the User Key selections is **Favorites**.

How to setup the User Key	
Using front-panel HARDKEY [softkey] buttons	Using Menus
<ol style="list-style-type: none"> 1. Press SYSTEM 2. then [User Key...] 	<ol style="list-style-type: none"> 1. Click Utility 2. then System 3. then User Key

User Key dialog box help



Assigns the selected function to the **User (soft) Key** on the main display. The User Key is always visible on the display.

Blank Leave the User Key Blank - no assignment.

Favorites

Allows you to select and display any eight softkeys from any level. Your Favorite menu selections are always two button presses away.

Note: Applications softkeys that are created as Favorites do NOT survive a Preset.

- To **Add** a Favorite, press and hold any softkey for three seconds . A message appears notifying you that the Favorite was added successfully.
- To **Delete** a Favorite, all of the Favorites must be cleared. Press **Clear Favorites**, then start adding new Favorites.

Setup Measurement When assigned to the User Key, press the Setup Measurement softkey to populate the softkeys with measurement settings that are relevant for the active trace. For example, when a Gain Compression measurement is active, press Setup Measurement to quickly access the GCA setup dialog.

Marker Functions, Transform, Gating When any of these functions are assigned to the User Key, one press of the User Key and the softkeys are populated with the most common settings for the corresponding function.

Undo Quickly access the Undo/Redo feature. [Learn more.](#)

Frequency | **Power** | **Source Ports** | **# Points** | **Trigger** :Ext
Sweep settings:...**Time** | **Setup** | **Segment** | **Power** |

New See **Remotely Specifying a Source Port** .

Description	SCPI	
Frequency		
Start Freq	SENS:FREQ:STAR	
Stop Freq	SENS:FREQ:STOP	
Center Freq	SENS:FREQ:CENT	
Span	SENS:FREQ:SPAN	
CW Frequency	SENS:FREQ:CW	
Number of Points	SENS:SWE:POIN	
Step size	SENS:SWEp:STEP	

Power Settings See Remotely Specifying a Source Port .		
Power ON OFF	OUTP	
Source Power (Auto ON OFF)	SOUR:POW:MODE	
Power Value	SOUR:POW1	
Port Selection	SENS:SWE:SRCP	
Couple Ports OFF ON	SOUR:POW:COUP	
Attenuator Mode Auto Manual	SOUR:POW:ATT:Auto	
Attenuation Value	SOUR:POW:ATT	
Power Slope ON OFF	SOUR:POW:SLOP:STAT	
Power Slope Value	SOUR:POW:SLOP	

Power Limit and Offsets

Set power limit	SYST:POWer:LIMit
Power limit ON/OFF	SYST:POWer:LIMit:STATe
Power limit UI lock	SYST:POWer:LIMit:LOCK

Specifying Source Ports See [Remotely Specifying a Source Port.](#)

Returns the number of source ports.	None
Returns the string names of source ports.	SOURce:CAT?
Returns the source port number of the specified string port name.	None

IF Bandwidth

IF Bandwidth	SENS:BWID
Previous IF Bandwidth	None
Next IF Bandwidth	None
Reduce IF BW	SENS:BWID:TRACk

Sweep

Sweep Time Value	SENS:SWE:TIME:AUTO
Number of Points	SENS:SWE:POIN

Sweep Setup

Sweep Type (Lin Pwr CW Seg Phase)	SENS:SWE:TYPE
Sweep Generation (Stepped Analog)	SENS:SWE:GEN
Dwell Time Value	SENS:SWE:DWEL
Dwell Time Auto set the minimum dwell time	SENS:SWE:DWEL:AUTO
Sweep Delay	SENS:SWEep:DWELl:SDELay
Alternate Sweeps	SENS:COUP
External ALC	SOUR:POW:DET
Enable Point Sweep	SENS:SWE:GEN:POINtsweep
Fast Sweep	SENS:SWE:SPEEd
Fast CW	SENS:SWE:TYPE:FACW
Power Sweep	
Start Power	SOUR:POW:STAR
Stop Power	SOUR:POW:STOP
Center	SOUR:POW:CENT
Span	SOUR:POW:SPAN
Segment Sweep	
ON OFF	SENS:SEGM
Add a segment	SENS:SEGM:ADD
Delete a segment	SENS:SEGM:DEL
Delete all segments	SENS:SEGM:DEL:ALL
Count the segments	SENS:SEGM:COUN
Read the segment number	None
Segment Center Frequency	SENS:SEGM:FREQ:CENT

Segment Frequency Span	SENS:SEGM:FREQ:SPAN
Segment Start Frequency	SENS:SEGM:FREQ:STAR
Segment Stop Frequency	SENS:SEGM:FREQ:STOP
Number of Points	SENS:SEGM:SWE:POIN
IF Bandwidth	SENS:SEGM:BWID
IF Bandwidth Option	SENS:SEGM:BWID:CONT
Source Power	SENS:SEGM:POW
Source Power Option	SENS:SEGM:POW:CONT
X-Axis Point Spacing	SENS:SEGM:X:SPAC
Allow Arbitrary Segments	SENS:SEGM:ARB
Upload a segment table	Sens:Segm:List
Download a segment table	Sens:Segm:List

Trigger	
Source (where trigger comes from)	
Source (Int Ext Manual)	TRIG:SOUR
Internal Manual	INIT:CONT
Trigger! (for Manual Source)	INIT
Scope (what is triggered)	
Scope (Global Channel)	TRIG:SCOP
Channel Settings (how the channel responds to triggers)	
Continuous	SENS:SWE:MODE CONTinuous
Read Continuous Mode	None

Number of Groups	SENS:SWE:GRO:COUN
Read Groups	None
Hold	SENS:SWE:MODE HOLD
Hold Mode (read-only)	None
All channels in Hold	SYST:CHAN:HOLD
All channels Resume	SYST:CHAN:RESume
Single	SENS:SWE:MODE SINGLE
Trigger Mode (Channel Point Sweep)	SENS:SWE:TRIG:Mode
Restart	INIT
Abort	ABOR

External Meas Trigger Input	
Scope (Global/Chan)	TRIG:SCOP
Trigger Delay (Global)	TRIG:DEL
Trigger Delay (Channel)	SENS:SWE:TRIG:DEL
MeasTrigIn/ Hand I/O	TRIG:ROUTE:INP
Level or Edge	TRIG:TYPE
Neg/Low or Pos/High	TRIG:SLOPE
Ready for Trigger Indicator (Out)	
MeasTrig Rdy/ Hand I/O	TRIG:ROUTE:READy
High / Low	TRIGger:READy:POLarity

Auxiliary Triggering (PNA-X and N522x models)	

Which AuxTrig connector pair being used.	N/A
How many Aux connector pairs.	TRIGger:AUX:COUNt?
AUX TRIG OUT	
Enable	TRIG:CHAN:AUX
Global or Channel Pref.	TRIG:PREF:AIGL
Polarity (Pos/Neg)	TRIG:CHAN:AUX:OPOL
Position (Before/After acq)	TRIG:CHAN:AUX:POS
OUT Pulse width	TRIG:CHAN:AUX:DUR
Point or Sweep.	TRIG:CHAN:AUX:INT
AUX TRIG (Ready) IN	
Enable Handshake	TRIG:CHAN:AUX:HAND
Edge or Level Level NOT in UI.	TRIG:CHAN:AUX:TYPE
Polarity High/leading or Low/trailing.	TRIG:CHAN:AUX:IPOL
Delay	TRIG:CHAN:AUX:DEL

SCPI Example Programs

Setup Measurements	Environment
Catalog Measurements	Visual Basic
Create an S-parameter Measurement	VBScript
Create a Balanced Measurement	VBScript
Channels, Windows, and Measurements	VBScript
Setup Sweep Parameters	Visual Basic
Setup the Display	Visual Basic
Setup a Measurement	Visual C++
Triggering the Analyzer	VBScript
Setup Markers	VBScript
Setup PNOP and PSAT Markers	VBScript
Calibrations See Calibrating the PNA Using SCPI	
Guided 2-Port or 4-Port Cal	VBScript
Guided 2-Port Comprehensive Cal	VBScript
Guided ECal	VBScript
Guided Mechanical	VBScript
Guided 1-port Mechanical Cal on Port 2	VBScript
Guided TRL	VBScript
Guided Unknown Thru or TRL Cal (apply Delta Match Cal)	VBScript
Perform a Guided QSOLT Cal	VBScript
Unguided ECal	VBScript
Unguided 2-port Mechanical Cal	VBScript
Unguided 1-port Mechanical Cal on Port 2	VBScript

Unguided 2-port Cal on a 4-Port PNA	Visual Basic
Unguided Thru Response Cals	VBScript
Perform a CalAllChannels Calibration	VBScript
Perform Unguided Cal on Multiple Channels	VBScript
Perform an ECal User Characterization	VBScript
Perform an ECAL Confidence Check	Visual Basic
Perform a Sliding Load Cal	Visual Basic
Load Error Terms during a Cal Sequence	None
Create a New Cal Kit	RMB
Modify a Calibration Kit	Visual Basic

Miscellaneous

Getting and Putting Data	Visual Basic
Getting and Putting Data (Definite Block Transfers)	RMB
Transfer Data using MMEM:TRANSfer	RMB
Establish a VISA Session	Visual Basic
Status Reporting	Visual Basic
GPIB Pass-Through	VBScript
PNA as Controller and Talker/Listener	Visual Basic
Upload and Download a Segment List	VBScript
Set ECal States	VBScript

See more programming information and examples
at:<http://na.support.keysight.com/pna/programming/>

[Unguided](#) | [SmartCal\(Guided\)](#) | [ECAL](#) | [Cal All](#) | [Save-Recall](#) | [Cal Sets](#) | [Cal Types](#) | [Prefer's Correction](#) | [Port Ext](#) | [Fixturing](#) | [CPM](#) | [Cal Kits](#) | [Standards](#) | [Source Pwr Cal](#) | [Power Sensors](#) | [Receiver Cal](#) | [Cal Data](#) | [CalPod](#) | [Custom Cal Window](#)

Description	SCPI
Perform an Unguided Calibration	
Launch Cal Wizard	SYST:CORR:WIZard
Simultaneous 2-Port Calibration	SENS:CORR:TSTandards
Acquisition Direction	SENS:CORR:SFORward
Measure a Standard	SENS:CORR:COLLect
Calculate Errors	SENS:CORR:COLL:SAVE
Do Isolation	SENS:CORR:COLLect
Perform and apply Response (Normalization) cal	SENS:CORR:COLL:METHod
Perform a Guided Cal	
Initiate a Guided Cal	SENS:CORR:COLL:GUID:INIT
List valid Connector Types for a Port	SENS:CORR:COLL:GUID:CONN:CAT?
List valid Cal Kits for a Connector type.	SENSe:CORR:COLL:GUID:CKIT:CAT?
Select a Connector Type	SENS:CORR:COLL:GUID:CONN:PORT
Select a Cal Kit	SENS:CORR:COLL:GUID:CKIT:PORT
Set cal method for each port pair.	SENS:CORR:COLL:GUID:PATH:CMET
Set Thru Method for each port pair.	SENS:CORR:COLL:GUID:PATH:TMET
Set Thru Port Pairs	SENS:CORR:COLL:GUID:THRU:PORT
Return Number of Steps in a Cal	SENS:CORR:COLL:GUID:STEPs?
Return a Description of a Cal Step	SENS:CORR:COLL:GUID:DESC?
Measure a Cal Standard in a Guided Cal	SENS:CORR:COLL:GUID:ACQuire
Save Cal	SENS:CORR:COLL:GUID:SAVE
Adapter settings for Unknown Thru or Adapter Removal	
Sets use of a THRU adapter	SENS:CORR:COLL:GUID:ADAP:CREate
Set adapter delay	SENS:CORR:COLL:GUID:ADAP:DELay
Set adapter description	SENS:CORR:COLL:GUID:ADAP:DESC
Set port pairs for adapter	SENS:CORR:COLL:GUID:ADAP:PATH
Clear the settings	SENS:CORR:COLL:GUID:ADAP:COUNt:ZERO

Return number of adapters	SENS:CORR:COLL:GUID:ADAP:COUNt?
Optional Guided Cal commands	
Auto-Orient ECal	SENS:CORR:PREF:ECAL:ORI
Manual orient ECAL	SENS:CORR:PREF:ECAL:PMAP
Read orientation	SENS:CORR:CKIT:ECAL:ORI?
Calculate Error Terms from a Guided Cal	SENS:CORR:COLL:GUID:SAVE
Computes the error correction terms, turns Correction ON, and saves the calibration to an existing, specified Cal Set	SENS:CORR:COLL:GUID:ETER:COMPUte
Save Cal to an existing Cal Set GUID	SENS:CORR:COLL:GUID:SAVE:CSET
Load Eterms during a cal	SENS:CORR:COLL:GUID:ETERms:LOAD
Perform Isolation	SENS:CORR:COLL:GUID:ISOL:PATHs
Increment Avg for Isolation	SENS:CORR:COLL:GUID:ISOL:AVER:INCR
Abort Guided cal	SENS:CORR:COLL:GUID:ABORt
Perform Enhanced Response Cal	
Set guided Cal method	SENS:CORR:COLL:GUID:PATH:CMET
Set guided Thru method	SENS:CORR:COLL:GUID:PATH:TMET
Perform Sliding Load Acquisition	
Set preference to not prompt	SENS:CORR:COLL:GUID:PREF:SLID
Read iteration step	SENS:CORR:COLL:GUID:ITER:COUNt?
Read minimum iterations	SENS:CORR:COLL:GUID:ITER:MIN?
Reset iterations	SENS:CORR:COLL:GUID:ITER:RESet
Perform an ECAL	
Specify Module and Characterization	SENS:CORR:COLL:ACQ
Do ECAL 1-Port	SENS:CORR:COLL:CKIT:99
Do ECAL 2-Port	SENS:CORR:COLL:CKIT:99
Get ECAL Module Info	SENS:CORR:COLL:CKIT:INF? SENS:CORR:CKIT:ECAL:INF?
Get list of ECal Modules attached to PNA	SENS:CORR:CKIT:ECAL:LIST?
Get list of characterizations in ECal module	SENS:CORR:CKIT:ECAL:CLISt?
Perform Module Orientation during calibration	SENS:CORR:PREF:ECAL:ORI

Maps ECAL Module to PNA Ports	SENS:CORR:PREF:ECAL:PMAP
Reads ECal orientation	SENS:CORR:CKIT:ECAL:ORI?
Perform ECal Isolation	SENS:CORR:COLL:ISOL:ECAL
Increment Avg for ECal Isolation	SENS:CORR:COLL:ISOL:AVER:INCR

Perform ECal User Characterizations

Perform User ECal Characterization	All SCPI commands
------------------------------------	-------------------

Manage PNA Disk Memory Characterizations

Delete disk memory characterizations.	SENSe:CORR:CKIT:ECAL:DMEMory:CLEar
Saves a disk memory characterization to an archive file.	SENS:CORR:CKIT:ECAL:EXPort
Imports the ECal characterization from the specified archive file.	SENSe:CORR:CKIT:ECAL:DMEMory:IMPort
Reads the user-characterization info from ECal module or PNA disk memory.	SENS:CORR:CKIT:ECAL:KNAM:INF?

ECal Confidence Check

Confidence Check Parameter	SENS:CORR:CCH:PAR
Confidence Check Acquire	SENS:CORR:CCHeck
Confidence Check Done	SENS:CORR:CCH:DONE

Set/Read ECal State

Sets the state of an ECAL module	CONT:ECAL:MOD:PATH:STATe
Read ECal state data	SENS:CORR:CKIT:ECAL:PATH:DATA?
Read number of ECal states for specified path	CONT:ECAL:MOD:PATH:COUNt? SENS:CORR:CKIT:ECAL:PATH:COUNt?

Calibrate All Channels

Select the channels to be calibrated.	SYST:CAL:ALL:CHAN:SEL
Set the IFBW	SYST:CAL:ALL:IFBW
Set the power level	SYST:CAL:ALL:PORT:SOUR:POW
Set the power offset	SYST:CAL:ALL:PORT:SOUR:POW:OFFS
Set the receiver atten	SYST:CAL:ALL:PORT:SOUR:POW:REC:ATT
Set the source atten	SYST:CAL:ALL:PORT:SOUR:POW:ATT
Set the User Calset Prefix	SYST:CAL:ALL:CSET:PREFix

Read unique Cal properties	SYST:CAL:ALL:MCL:PROP:NAME:CAT?
Read unique property values	SYST:CAL:ALL:MCL:PROP:VAL:CAT?
Set property name/value	SYST:CAL:ALL:MCL:PROP:VAL
Read Cal channel	SYST:CAL:ALL:GUID:CHAN?
Get GuidedCal handle	N/A
For each channel, sets the ports to be calibrated.	SYST:CAL:ALL:CHAN:PORT
Returns a final list of ports to be calibrated.	SYST:CAL:ALL:GUID:PORT?
Read generated Cal Sets	SYST:CAL:ALL:CSET:CATalog?

Recall / Save / Apply a Calibration or Error Term	
Recall a Calibration	SENS:CORR:CSET
Apply a Calibration to a measurement	SENS:CORR:CSET
Save a Calibration	SENS:CORR:CSET:SAVE
Save or Recall an Error Term	CALC:DATA Scorr
Read/ Write Cal Set data	SENS:CORR:CSET:DATA
Apply an Error Term after Uploading	SENS:CORR:COLLect:APPLy
Cal Sets	
Create a Cal Set	SENS:CORR:CSET:CREate
Delete a Cal Set	SENS:CORR:CSET:DEL
List Cal Sets	SENS:CORR:CSET:CAT?
List Cal Sets in PNA	None
Get Cal Set Information	None
List Cal Set Error Terms	SENS:CORR:CSET:ETERM:CAT?
Return if a Cal Set exists	CSET:EXISts?
Select a Cal Set by GUID	SENS:CORR:CSET:GUID
Apply a Cal Set to a channel	SENS:CORR:CSET:ACTivate
Copy a Cal Set	SENSe:CORR:CSET:COPY
Save a Cal Set	SENS:CORR:CSET:SAVE
Save Cal Sets	None
Automatically save to User Cal Set	SENS:CORR:PREF:CSET:SAVE
Change the Description of a Cal Set	SENS:CORR:CSET:DESC

Change the Name of a Cal Set	SENS:CORR:CSET:NAME
Recall a Cal File	SENS:CORR:CSET
Save 'in-memory' Cal Set to disk.	SENS:CORR:CSET:FLATten
Create Cal Set with De-embedded fixture removed.	CSET:FIXTure:DEEMbed
Create Cal Set with Matching Network included.	CSET:FIXTure:EMBed
Adds stimulus data to a specific buffer.	None
Returns the stimulus values over which the specific error term was acquired.	None
Returns FOM stimulus values from a Calset.	SENS:CORR:CSET:STIM?
Returns the Cal Types from the calset.	None
Returns the properties of the calset.	None
Returns the numbers of the channels using the calset.	None
Unselect Cal Set	SENS:CORR:CSET:DEAC

Apply Cal Types	
Catalog ALL Cal Types for the PNA	SENS:CORR:TYPE:CAT?
Catalog Cal Types in the Cal Set	SENS:CORR:CSET:TYPE:CAT?
Is a specific Cal Type contained in the Cal Set?	None
Set and return the measurement Cal Type	CALC:CORR:TYPE?
Set port to measure QSOLT reflection standards.	None

Correction Settings	
Turn Correction ON OFF for a channel	SENS:CORR
Turn Correction ON OFF for a measurement	CALC:CORR
Interpolation ON OFF	SENS:CORR:INT
Returns the error correction state for the measurement	CALC:CORR:INDicator?

Preferences	
Set default Cal Set Save behavior	SENS:CORR:PREF:CSET:SAVE
Sets behavior for simulated cal	SENS:CORR:PREF:SIMCal
External or internal trigger during cal	SENS:CORR:PREF:TRIG:FREE
Set ECal Auto-orient	SENS:CORR:PREF:ECAL:ORI

Set ECal Port Map	SENS:CORR:PREF:ECAL:PMAP
Set default Cal Type	None

Port Extensions	
Extensions ON OFF	SENS:CORR:EXT
Port 1 Extensions Value	SENS:CORR:EXT:PORT
Port 2 Extensions Value	SENS:CORR:EXT:PORT
Set Freq 1 2	SENS:CORR:EXT:PORT:FREQ
Set Loss 1 2	SENS:CORR:EXT:PORT:LOSS
Use 1 2	SENS:CORR:EXT:PORT:INCL
Set Loss at DC	SENS:CORR:EXT:PORT:LDC
Relative Velocity	SENS:CORR:RVEL:COAX
Port Ext in distance	SENS:CORR:EXT:PORT:DIST
Set distance units	SENS:CORR:EXT:PORT:UNIT
Set Media per port	SENS:CORR:EXT:PORT:MEDIUm
Set waveguide cutoff freq per port	SENS:CORR:EXT:PORT:WGCutoff
Set Velocity Factor per port	SENS:CORR:EXT:PORT:VELF
Couple to system Velocity Factor	SENS:CORR:EXT:PORT:SYSV
Couple to system Media type	SENS:CORR:EXT:PORT:SYSM

Auto Port Extensions	
Measure OPEN or SHORT for Auto Port Ext.	SENS:CORR:EXT:AUTO:MEASure
Sets the frequencies used for Auto Port Ext. calculation.	SENS:CORR:EXT:AUTO:CONFIg
Include loss correction in Auto Port Ext.?	SENS:CORR:EXT:AUTO:LOSS
Include DC Offset in Auto Port Ext.?	SENS:CORR:EXT:AUTO:DCOF
Enable specified port for Auto Port Ext..	SENS:CORR:EXT:AUTO:PORT<n>
Clears old port extension delay and loss data.	SENS:CORR:EXT:AUTO:RESet
Set user span start frequency for Auto Port Ext.	SENS:CORR:EXT:AUTO:STARt
Set user span stop frequency for Auto Port Ext.	SENS:CORR:EXT:AUTO:STOP

Fixturing Commands

Turn fixturing ON and OFF	CALC:FSIM:STAT
Change order of operations	CALC:FSIM:SEND:OORD
2and 4-port Extrapolate	CALC:FSIM:SNP:EXTR
2-Port Fixturing	
Port matching ON and OFF	CALC:FSIM:SEND:PMC:STAT
Reverse ports	CALC:FSIM:SEND:DEEM:PORT<n>:SNP:REV
Sets Port Matching circuit model.	CALC:FSIM:SEND:PMC:PORT:TYPE
Sets Port Matching 'S2P' file name.	CALC:FSIM:SEND:PMC:PORT:USER:FIL
Sets Capacitance 'C' value.	CALC:FSIM:SEND:PMC:PORT:PAR:C
Sets Conductance 'G' value.	CALC:FSIM:SEND:PMC:PORT:PAR:G
Sets Inductance 'L' value.	CALC:FSIM:SEND:PMC:PORT:PAR:L
Sets Resistance 'R' value.	CALC:FSIM:SEND:PMC:PORT:PAR:R
De-embed ON and OFF	CALC:FSIM:SEND:DEEM:STAT
Sets De-embedding circuit model.	CALC:FSIM:SEND:DEEM:PORT
Sets De-embedding 'S2P' file name.	CALC:FSIM:SEND:DEEM:PORT:USER:FIL
Port Impedance ON and OFF	CALC:FSIM:SEND:ZCON:STAT
Port Z Real	CALC:FSIM:SEND:ZCON:PORT:REAL
Port Z Imag	CALC:FSIM:SEND:ZCON:PORT:IMAG
Port Z Real and Imag	CALC:FSIM:SEND:ZCON:PORT:Z0
4-Port Network Embed/De-embed commands	
Specifies the PNA / DUT topology	CALC:FSIM:EMB:TYPE
Specifies the 4-port touchstone file	CALC:FSIM:EMB:NETW:FIL
Embed De-embed?	CALC:FSIM:EMB:NETW:TYPE
Specify PNA port connections	CALC:FSIM:EMB:TOP:A:PORT CALC:FSIM:EMB:TOP:B:PORT CALC:FSIM:EMB:TOP:C:PORT
4-port remap	CALC:FSIM:EMB:NETW<n>:PMAP
Turn ON or OFF	CALC:FSIM:EMB:STAT
Differential Port Arbitrary Impedance	
Sets the impedance value	CALC:FSIM:BAL:DZC:BPOR:Z0
Sets real part of impedance	CALC:FSIM:BAL:DZC:BPOR:REAL
Sets imaginary part of impedance	CALC:FSIM:BAL:DZC:BPOR:IMAG

Turn ON or OFF	CALC:FSIM:BAL:DZC:STAT
Common Mode Port Arbitrary Impedance	
Sets the impedance value	CALC:FSIM:BAL:CZC:BPOR:Z0
Sets real part of impedance	CALC:FSIM:BAL:CZC:BPOR:REAL
Sets imaginary part of impedance	CALC:FSIM:BAL:CZC:BPOR:IMAG
Turn ON or OFF	CALC:FSIM:BAL:CZC:STAT
Differential Port Matching	
Sets type of circuit to embed.	CALC:FSIM:BAL:DMC:BPOR
Specifies the 2-port touchstone file	CALC:FSIM:BAL:DMC:BPOR:USER:FIL
Sets Capacitance value	CALC:FSIM:BAL:DMC:BPOR:PAR:C
Sets Conductance value	CALC:FSIM:BAL:DMC:BPOR:PAR:G
Sets Inductance value	CALC:FSIM:BAL:DMC:BPOR:PAR:L
Sets Resistance value	CALC:FSIM:BAL:DMC:BPOR:PAR:R
Turns ON/OFF	CALC:FSIM:BAL:DMC:STAT
Power Compensation	
Compensate source power	CALC:FSIM:SEND:POW:PORT:COMP
Remote ONLY	
Create Cal Set with De-embedded fixture removed.	CSET:FIXTure:DEEMbed
Create Cal Set with Matching network included.	CSET:FIXTure:EMBed

Cal Plane Manager	SCPI
Characterize a fixture	CSET:FIXTure:CHARacterize
Creates a single S2P file from two existing files.	CSET:FIXTure:CASCade

Manage and Modify Cal Kits	
Set a Cal Kit Active	SENS:CORR:COLL:CKIT
Clear all Cal Kits from PNA	SENS:CORR:CKIT:CLE
Get a Handle to the Active Cal Kit	
Save All Cal Kits after Modifying	

Load collection of Kits	SENS:CORR:CKIT:LOAD
Load (Recall) All Cal Kits	
Import a specified kit.	SENS:CORR:CKIT:IMPorT
Restore Cal Kit Default	SENS:CORR:COLL:CKIT:RESet
Restore ALL Cal Kits Default	SENS:CORR:CKIT:INITialize
Build a Hybrid Cal Kit	
Set the Name of a Cal Kit	SENS:CORR:COLL:CKIT:NAME
Set a description of a Cal Kit	SENS:CORR:COLL:CKIT:DESC
Get the amount of installed kits	SENS:CORR:CKIT:COUNt?
Set the Port Label of a Cal Kit	
Saves a Cal Kit to a file.	SENS:CORR:CKIT:EXP

Modify TRL Cal Kit	
Set reference plane	SENS:CORR:COLL:CKIT:TRL:RPLane
Set impedance standard	SENS:CORR:COLL:CKIT:TRL:IMPedance
Set LRL auto-characterization	SENS:CORR:COLL:CKIT:TRL:LRLChar

Modify Cal Standards	
Select a Cal Standard	SENS:CORR:COLL:CKIT:STAN
Delete a standard	SENS:CORR:COLL:CKIT:STAN:REM
Change description of a standard	SENS:CORR:COLL:CKIT:STAN:SDES
Assign a Class to a Standard	SENS:CORR:COLL:CKIT:ORD1
Set Standard Type	SENS:CORR:COLL:CKIT:STAN:TYPE
Add connector family name	SENS:CORR:COLL:CKIT:CONN:ADD
Delete connector family name	SENS:CORR:COLL:CKIT:CONN:DEL
List connector family names used in a Cal Kit	SENS:CORR:COLL:CKIT:CONN:CAT?
Replace connector family name.	SENS:CORR:COLL:CKIT:CONN:FNAME
Assign connector family name to a standard	SENS:CORR:COLL:CKIT:CONN:SNAME
Set Delay	SENS:CORR:COLL:CKIT:STAN:DEL
Set Loss	SENS:CORR:COLL:CKIT:STAN:LOSS
Set Impedance	SENS:CORR:COLL:CKIT:STAN:IMP

Set Max Frequency	SENS:CORR:COLL:CKIT:STAN:FMAX
Set Min Frequency	SENS:CORR:COLL:CKIT:STAN:FMIN
Set Label	SENS:CORR:COLL:CKIT:STAN:LAB
Set Medium (coax waveguide)	SENS:CORR:COLL:CKIT:STAN:CHAR
Set Capacitance (C0 to C3)	SENS:CORR:COLL:CKIT:STAN:C0
Set Inductance (L0 to L3)	SENS:CORR:COLL:CKIT:STAN:L0
Set Arbitrary Impedance (TZReal, TZImag)	SENS:CORR:COLL:CKIT:STAN:TZReal

Modify TRL Cal Kit	
Set reference plane	SENS:CORR:COLL:CKIT:TRL:RPLane
Set impedance standard	SENS:CORR:COLL:CKIT:TRL:IMPedance
Set LRL auto-characterization	SENS:CORR:COLL:CKIT:TRL:LRLChar

Source Power Calibration	
Copy Source Power cal to another channel	SYST:MACR:COPY:CHAN
GPIB Power Meter Address	SYST:COMM:GPIB:PMET:ADDR
Turn correction ON OFF	SOUR:POW:CORR

Power Meter/Sensor settings	
See commands to configure a Power Meter as Receiver (PMAR)	
See commands to configure multiple power sensors for guided Power Cal	
Specifies the type of power sensor to be used	SYST:COMM:PSEN
Specifies the location of the power sensor to be used.	SYST:COMM:PSEN
Returns the ID string of connected USB power meters / sensors.	SYST:COMM:USB:PMET:CAT?
Read/Write cal data	SOUR:POW:CORR:DATA
Receiver Cal	
Set offset from test port power	SENS:CORR:RPOWER:OFFSet[:AMPLitude]
Set cal method to receiver cal	SENS:CORR:COLL:METH RPOWER
Take measurement	SENS:CORR:COLL[:ACQ] Power
Turn receiver cal ON OFF	SENS:CORR[:STATe] ON OFF
Do interpolation	SENS:CORR:INT[:STATe] ON OFF

CalPod	
Command used to send other commands as arguments	CONTRol:CALPod:COMMand
Start the CalPod software	Calpod:LAUNCh
Assign Calpod serial number to a port.	Calpod:ENABLE
Unassign Calpod serial number from a port.	Calpod:Disable
Initialize the selected channel	Calpod:INITialize:ACTive
Initialize ALL channels	Calpod:INITialize:ALL
Recorrect the selected channel	Calpod:Recorrect:ACTive
Recorrect ALL channels	Calpod:Recorrect:ALL
Show refresh dialog	Calpod:SHOW
Hide refresh dialog	Calpod:HIDE
Sets impedance state	Calpod:STATe
Read Calpod temperature	Calpod:TEMP?

Custom Cal Window	
Turn ON OFF Custom Cal window.	SENS:CORR:COLL:DISP:WIND
Show NO Custom Cal windows.	SENS:CORR:COLL:DISP:WIND:AOff
Specify channel to sweep before Cal acquisition.	SENS:CORR:COLL:SWE:CHAN
Sweep NO channel before Cal acquisition.	SENS:CORR:COLL:SWE:CHAN:AOff
Preview sweep before remote Cal acquisition.	SENS:CORR:COLL:GUID:PACQuire

Retrieve and Put Calibration Data	
Retrieve Cal Data from the PNA	SENS:CORR:CSET:DATA
Put Cal Data in the PNA	SENS:CORR:CSET:DATA

Calculate:FSimulator:SEnDed Commands

Specifies settings for embedding and de-embedding **Single-Ended** (2-port) fixturing circuits.

```
CALCulate:FSIMulator:SEnDed:  
DEEMbed  
PORT  
SNP:REVerse  
[TYPE]  
USER:FILeName  
STATe  
OORder  
PMCircuit  
PORT  
PARameters  
C  
G  
L  
R  
[TYPE]  
USER:FILeName  
STATe  
POWER:PORT:COMPensation  
ZCONversion  
PORT  
IMAG  
REAL  
Z0[:R]  
STATe
```

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Fixturing](#)

- [Set Port Extensions with SCPI](#)
- [Synchronizing the Analyzer and Controller](#)
- [SCPI Command Tree](#)

Note: CALC:FSIM commands affect ALL measurements on the specified channel.

CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>:SENSE:DEEMbed:PORT<n>:SNP:REV<bool>

(Read-Write) Set and read whether or not to reverse ports on a 2-port fixture or adapter to de-embedded. [Learn more.](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<port> PNA port number to which SNP file is to be de-embedded.

<bool> Choose from:
ON or **1** - Reverse ports
OFF or **0** - Do NOT Reverse ports

Examples

```
CALC:FSIM:SEND:DEEM:PORT1:SNP:REV 1
calculate2:fsimulator:sended:deembedded:port2:snp:reverse 0
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>:SENSE:DEEMbed:PORT<n>:SNP:REV

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>[:TYPE] <char>

(Read-Write) Select whether or not to load a 2-port De-embedding circuit model for the specified port number. Circuit model USER is valid when an associated .s2p file is specified with **CALC:FSIM:SEND:DEEM:PORT1:USER:FILENAME**.

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.
- <port> Port number to receive circuit model.
- <char> Choose from:
 - NONE** - Port does not have a circuit model.
 - USER** - Circuit model for the port will be loaded from PNA drive.

Examples

```
CALC:FSIM:SEND:DEEM:PORT1 USER  
calculate2:fsimulator:sended:deembed:port2:type none
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>[:TYPE]?

Return Type Character

Default None

CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>:U<string>

(Read-Write) Specifies the filename of the circuit model to be used for de-embedding. Circuit model is applied when both **CALC:FSIM:SEND:DEEM:PORT1 USER** is selected and the filename is specified with this command.

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.

<port> Port number to receive circuit model.

<string> File name and extension (.s2P) of the de-embedding circuit.

Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents

To recall from a different folder, specify the full path name.

Examples

```
CALC:FSIM:SEND:DEEM:PORT1:USER:FILE 'myFile.s2P'  
calculate2:fsimulator:deembed:port2:user:file "C:/Program  
Files/Keysight/Network Analyzer/Documents/myFile.s2P"
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:PORT<n>:USER:FI

Return Type String

Default Not Applicable

CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:STATE<bool>

(Read-Write) Turns de-embedding ON or OFF for all ports on the specified channel.

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 de-embedding ON

OFF or **0** - Turns de-embedding OFF

Examples

```
CALC:FSIM:SEND:DEEM:STAT 1  
calculate2:fsimulator:sended:deembed:state 0
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:DEEMbed:STATE?

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:SENDED:OORDER <a,b,c,d>

(Read-Write) Sets and returns the order in which Single-ended Fixture Operations occur. [Learn more about these operations.](#)

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.
- <a,b,c,d> Order of operations, where:
- 0** - Port Extension operation
 - 1** - 2-Port DeEmbedding operation
 - 2** - Port Matching operation
 - 3** - Arbitrary Impedance operation

Examples

```
CALC:FSIM:SEND:OORD 1,2,3,0  
calculate2:fsimulator:sended:order 1,2,3,0
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDED:OORDER?

Return Type Comma-separated values

Default 0,1,2,3

CALCulate<cnum>:FSIMulator:SENDED:PMCircuit:PORT<n>:I<value>

(Read-Write) Sets and returns the Capacitance, 'C' value for the specified port number.

Note: This command affects ALL measurements on the specified channel.

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.

<port> Port number to receive value.

<value> Capacitance value in farads. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:SEND:PMC:PORT1:PAR:C 0.00002  
calculate2:fsimulator:sended:pmcircuit:port2:parameters:c 0.00003
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>:PARameter

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>:I<value>

(Read-Write) Sets and returns the Conductance, 'G' value for the specified port number.

Note: This command affects ALL measurements on the specified channel.

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.

<port> Port number to receive value.

<value> Conductance value in siemens. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:SEND:PMC:PORT1:PAR:G 0.00002  
calculate2:fsimulator:sended:pmcircuit:port2:parameters:g 0.00003
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>:PARameter

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:SENDED:PMCircuit:PORT<n>:I <value>

(Read-Write) Sets and returns the Inductance, 'L' value for the specified port number.

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.

<port> Port number to receive value.

<value> Inductance value in henries. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:SEND:PMC:PORT1:PAR:L 0.00002  
calculate2:fsimulator:sended:pmcircuit:port2:parameters:l 0.00003
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDED:PMCircuit:PORT<n>:PARAMeters

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:SENDED:PMCircuit:PORT<n>:I <value>

(Read-Write) Sets and returns the Resistance, 'R' value for the specified port number.

Note: This command affects ALL measurements on the specified channel.

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement

that channel. If unspecified, <num> is set to 1.

<port> Port number to receive value.

<value> Resistance value in ohms. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:SEND:PMC:PORT1:PAR:R 0.00002  
calculate2:fsimulator:sended:pmcircuit:port2:parameters:r 0.00003
```

Query Syntax CALCulate<num>:FSIMulator:SENDEd:PMCircuit:PORT<n>:PARameter

Return Type Numeric

Default 0

CALCulate<num>:FSIMulator:SENDEd:PMCircuit:PORT<n>[:TYPE] <char>

(Read-Write) Select whether of not to load a 2 port matching circuit model for the specified port number. Circuit model USER is valid when an associated .s2p file is specified with **CALC:FSIM:SEND:PMC:PORT1:USER:FILE**

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<port> Port number to receive circuit model.

<char> Circuit model. Choose from

NONE No circuit model

SLPC Series L - Parallel C

PCSL Parallel C - Series L

PLSC Parallel L - Series C

SCPL Series C - Parallel L

PLPC Parallel L - Parallel C

USER Load S2P file

Examples

```
CALC:FSIM:SEND:PMC:PORT1:USER  
calculate2:fsimulator:sended:pmcircuit:port2:type slpc
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>
[:TYPE]?

Return Type Character

Default None - No circuit model

CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>:U <string>

(Read-Write) Specifies the filename of the circuit model to be used for port matching. Circuit model is applied when both **CALC:FSIM:SEND:PMC:PORT1 USER** is selected and the file name is specified with this command.

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.

<port> Port number to receive circuit model.

<string> File name and extension (.s2P) of the de-embedding circuit.

Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents"

To recall from a different folder, specify the full path name.

Examples

```
CALC:FSIM:SEND:PMC:PORT1:USER:FILE 'myFile.s2P'  
calculate2:fsimulator:pmcircuit:port2:user:file "C:/Program  
Files/Keysight/Network Analyzer/Documents/myFile.s2P"
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:PORT<n>:USER:FILE

Return Type String

Default Not Applicable

CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:STATE<bool>

(Read-Write) Turns Port Matching ON or OFF for all ports on the specified channel.

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 Port Matching ON
OFF or 0 - Turns Port Matching OFF

Examples

```
CALC:FSIM:SEND:PMC:STAT 1  
calculate2:fsimulator:sended:pmcircuit:state 0
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:PMCircuit:STATE?

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:SENDEd:POWER:PORT<n>:COI<bool>

(Read-Write) Adjusts the source power at the specified port by the combined amount of l 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.

Parameters

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.
- <port> Port number to receive power compensation.
- <bool> Choose from:
ON or 1 - Compensate source power
OFF or 0 - Do NOT compensate source power

Examples

```
CALC:FSIM:SEND:POW:PORT1:COMP 1  
calculate2:fsimulator:power:port2:compensation off
```

Query Syntax CALCulate<num>:FSIMulator:SENDED:POWER:PORT<n>:COMPensate

Return Type Boolean

Default OFF

CALCulate<num>:FSIMulator:SENDED:ZCONversion:PORT<n> <value>

(Read-Write) Sets and returns the Imaginary portion of the impedance value for the specified single-ended port. Use **CALC:FSIM:SEND:ZCON:PORT:REAL** to set the real value. Or use **CALC:FSIM:SEND:ZCON:PORT:ZO** to set both values together.

Note: This command affects ALL measurements on the specified channel.

Parameters

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.
- <n> Port number to receive value.
- <value> Imaginary impedance value. Choose a value between -1E18 and 1E18

Examples

```
CALC:FSIM:SEND:ZCON:PORT1:IMAG 75
```

```
calculate2:fsimulator:sended:zconversion:port2:imag 150
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:ZCONversion:PORT<n>:IMAG

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:SENDEd:ZCONversion:PORT<n> <value>

(Read-Write) Sets and returns the Real portion of the impedance value for the specified single-ended port. Use **CALC:FSIM:SEND:ZCON:PORT:IMAG** to set the imaginary value. Or use **CALC:FSIM:SEND:ZCON:PORT:Z0** to set both values together.

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.

<n> Port number to receive value.

<value> Real Impedance value in ohms. Choose a value between 0 to 1E7

Examples

```
CALC:FSIM:SEND:ZCON:PORT1:REAL 51  
calculate2:fsimulator:sended:zconversion:port2:real 75
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:ZCONversion:PORT<n>:REAL

Return Type Numeric

Default 50

CALCulate<cnum>:FSIMulator:SENDEd:ZCONversion:PORT<n>

<value>

(Read-Write) 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 **CALC:FSIM:SEND:ZCON:PORT:REAL** and **CALC:FSIM:SEND:ZCON:PORT:IMAG**.

Note: This command affects ALL measurements on the specified channel.

Parameters

- <cnm> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnm> is set to 1.
- <n> Port number to receive value.
- <value> Port Impedance value in ohms. Choose a value between 0 to 1E7

Examples

```
CALC:FSIM:SEND:ZCON:PORT1:Z0 51  
calculate2:fsimulator:sended:zconversion:port2:z0:r 75
```

Query Syntax CALCulate<cnm>:FSIMulator:SENDEd:ZCONversion:PORT<n>:Z0[:R

Return Type Numeric

Default 50

CALCulate<cnm>:FSIMulator:SENDEd:ZCONversion:STATE <bool>

(Read-Write) Turns Port Impedance ON or OFF for all ports on the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

- <cnm> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnm> is set to 1.

<bool> Choose from:
ON or **1** - Turns Port Impedance ON
OFF or **0** - Turns Port Impedance OFF

Examples

```
CALC:FSIM:SEND:ZCON:STAT 1  
calculate2:fsimulator:sended:zconversion:state 0
```

Query Syntax CALCulate<cnum>:FSIMulator:SENDEd:ZCONversion:STATe?

Return Type Boolean

Default OFF

Last Modified:

3-Feb-2014	Fixed tree
20-May-2011	Fixed SNP syntax
12-Nov-2010	Added SNP reverse
2-Feb-2009	Added OORder

Calculate:FSimulator:GLOop Commands

Specifies settings and fixturing for ground loop de-embedding / embedding.

CALCulate:FSIMulator:GLOop:

DEEMBed:

| **C**
| **L**
| **R**
| **STATe**
| **TYPE**
| **USER**

EMBed:

| **C**
| **L**
| **R**
| **STATe**
| **TYPE**
| **USER**

See Also

- [SCPI Command Tree](#)

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using [Calc:Par:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>:FSIMulator:GLOop:DEEMbed:C <value>

(Read-Write) Sets and returns the Capacitance, 'C' value for the ground loop de-embedding of the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<value> Capacitance value in farads. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:DEEM:C 0.00002  
calculate2:fsimulator:gloop:deembed:c 0.00003
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:DEEMbed:C?

Return Type Numeric

Default 0

CALCulate<num>:FSIMulator:GLOop:DEEMbed:L <value>

(Read-Write) Sets and returns the Inductance, 'L' value for the ground loop de-embedding of the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<value> Inductance value in henries. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:DEEM:L 0.00002  
calculate2:fsimulator:gloop:deembed:l 0.00003
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:DEEMbed:L?

Return Type Numeric

Default 0

CALCulate<num>:FSIMulator:GLOop:DEEMbed:R <value>

(Read-Write) Sets and returns the Resistance, 'R' value for the ground loop de-embedding of the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<value> Resistance value in ohms. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:DEEM:R 0.00002  
calculate2:fsimulator:gloop:deembed:R 0.00003
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:DEEMbed:R?

Return Type Numeric

Default 0

CALCulate<num>:FSIMulator:GLOop:DEEMbed:STATe <bool>

(Read-Write) Turns ON or OFF De-embedding for the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<bool> Choose from:

ON or 1 - Turns De-embedding ON

OFF or 0 - Turns De-embedding OFF

Examples

```
CALC:FSIM:GLO:DEEM:STAT 1  
calculate2:fsimulator:gloop:deembed:state 0
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:DEEMbed:STATe?

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:GLOop:DEEMbed:TYPE <char>

(Read-Write) Specifies the circuit model type for ground loop de-embedding.

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:

USER - Loads s1p file specified using the **CALC:FSIM:GLO:DEEM:USER** command.

RL - Selects Shunt L circuit model.

RC - Selects Shunt C circuit mode.

Examples

```
CALC:FSIM:GLO:DEEM:TYPE RL  
calculate2:fsimulator:gloop:deembed:type RC
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:DEEMbed:TYPE?

Return Type Character

Default RL

CALCulate<cnum>:FSIMulator:GLOop:DEEMbed:USER <filename>

(Read-Write) Specifies the filename of the s1p file to load for ground loop de-embedding.

Note: This command affects ALL measurements on the specified channel.

Parameters

- <num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.
- <filename> File name and extension (.s1P) of the de-embedding circuit.
Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents"
To recall from a different folder, specify the full path name.

Examples

```
CALC:FSIM:GLO:DEEM:USER 'myFile.s1P'  
calculate2:fsimulator:gloop:deembed:user "C:/Program  
Files/Keysight/Network Analyzer/Documents/myFile.s1P"
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:DEEMbed:USER?

Return Type String

Default Not Applicable

CALCulate<num>:FSIMulator:GLOop:EMBed:C <value>

(Read-Write) Sets and returns the Capacitance, 'C' value for the ground loop embedding of the specified channel.

Note: This command affects ALL measurements on the specified channel.

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<value> Capacitance value in farads. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:EMB:C 0.00002  
calculate2:fsimulator:gloop:embed:c 0.00003
```

Query Syntax CALCulate<num>:FSIMulator:GLOop:EMBed:C?

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:GLOop:EMBed:L <value>

(Read-Write) Sets and returns the Inductance, 'L' value for the ground loop embedding of the specified channel.

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.

<value> Inductance value in henries. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:EMB:L 0.00002  
calculate2:fsimulator:gloop:embed:l 0.00003
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:EMBed:L?

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:GLOop:EMBed:R <value>

(Read-Write) Sets and returns the Resistance, 'R' value for the ground loop embedding of the specified channel.

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.

<value> Resistance value in ohms. Choose a value between -1E18 to 1E18

Examples

```
CALC:FSIM:GLO:EMB:R 0.00002  
calculate2:fsimulator:gloop:embed:R 0.00003
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:EMBed:R?

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:GLOop:EMBed:STATe <bool>

(Read-Write) Turns ON or OFF Embedding for the specified channel.

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 Embedding ON
OFF or 0 - Turns Embedding OFF

Examples

```
CALC:FSIM:GLO:EMB:STAT 1  
calculate2:fsimulator:gloop:embed:state 0
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:EMBed:STATe?

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:GLOop:EMBed:TYPE <char>

(Read-Write) Specifies the circuit model type for ground loop embedding. Learn more.

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:
USER - Loads s1p file specified using the

CALC:FSIM:GLO:EMB:USER command.

RL - Selects Shunt L circuit model.

RC - Selects Shunt C circuit mode.

Examples

```
CALC:FSIM:GLO:EMB:TYPE RL  
calculate2:fsimulator:gloop:embed:type RC
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:EMBed:TYPE?

Return Type Character

Default RL

CALCulate<cnum>:FSIMulator:GLOop:EMBed:USER <filename>

(Read-Write) Specifies the filename of the s1p file to load for ground loop embedding.

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.

<filename> File name and extension (.s1P) of the embedding circuit.

Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents"

To recall from a different folder, specify the full path name.

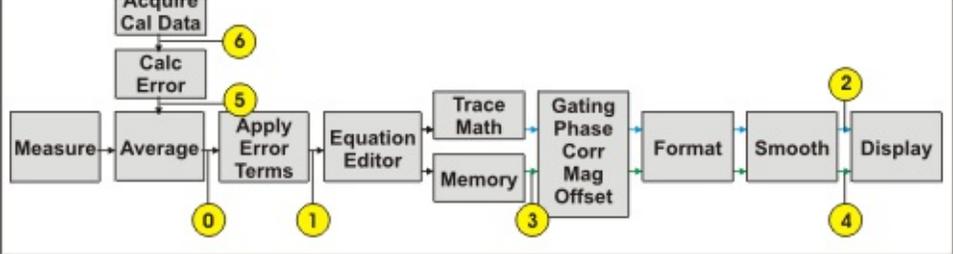
Examples

```
CALC:FSIM:GLO:EMB:USER 'myFile.s1P'  
calculate2:fsimulator:gloop:embed:user "C:/Program  
Files/Keysight/Network Analyzer/Documents/myFile.s1P"
```

Query Syntax CALCulate<cnum>:FSIMulator:GLOop:EMBed:USER?

Return Type String

Default Not Applicable



Equations Used to Generate Uncertainty Curves

This topic contains the measurement uncertainty equations used to generate the uncertainty curves in **Specifications** documents.

It also contains general information about determining system measurement uncertainties.

Note: RSS Computations are included along with worst case computations.

Learn about the following subjects in this topic:

- [Measurement Uncertainty Equations](#)
- [Forward Reflection Uncertainty](#)
- [Forward Transmission Uncertainty](#)
- [Reverse Transmission Uncertainty](#)
- [Reverse Reflection Uncertainty](#)
- [Sources of Systematic Errors](#)
- [Sources of Random Errors](#)
- [Determining Expected System Performance](#)
- [Determining Cable Stability Terms \(\$C_{R1}\$, \$C_{R2}\$, \$C_{TM1}\$, \$C_{TM2}\$, \$C_{TP1}\$, \$C_{TP2}\$ \)](#)

See Also

- [Measurement Errors](#)
 - [What is Measurement Calibration?](#)
 - [Why is Calibration Necessary?](#)
-

Measurement Uncertainty Equations

Any measurement result is the vector sum of the actual test device response plus all error terms. The precise effect of each error term depends on its magnitude and phase relationship to the actual test device response. When the phase of an error response is not known, phase is assumed to be worst case (-180° to +180°).

View the [abbreviations for residual systematic errors](#) used in the equations.

View the [abbreviations for random errors](#) used in the error models and equations.

Forward Reflection Uncertainty

Equation 1: Forward Reflection Magnitude Uncertainty (Worst Case Computation)

$$\Delta S_{1(\text{mag})} = \sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}$$

where:

$$\text{Systematic} = E_{LF} + E_{RF}S_{11} + E_{SF}S_{11}^2 + S_{21}S_{12} (E_{LF} + 2E_{SF}E_{LF}S_{11} + E_{LF}^2S_{22}) + A_M S_{11}$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = C_{M1}^2 (1 + S_{11}^4) + 4C_{M1}^2 S_{11}^2 + C_{M2}^2 S_{21}^2 S_{12}^2$$

$$R^2 = (R_{R1} (1 + S_{11}^2) + 2R_{T1} S_{11})^2 + (R_{R2} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_F S_{11})^2 + N_F^2$$

Equation 2: Forward Reflection Phase Uncertainty (Worst Case Computation)

$$\Delta S_{1(\text{phase})} = \sin^{-1} \left(\frac{\sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}}{S_{11}} \right) + 2C_{P1}$$

where:

$$\text{Systematic} = E_{LF} + E_{RF}E_{11} + E_{SF}E_{11}^2 + S_{21}S_{12} (E_{LF} + 2E_{SF}E_{LF}S_{11} + E_{LF}^2S_{22}) + \sin(A_F) S_{11}$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = C_{M1}^2 (1 + S_{11}^4) + 4C_{M1}^2 S_{11}^2 + C_{M2}^2 S_{21}^2 S_{12}^2$$

$$R^2 = (R_{R1} (1 + S_{11}^2) + 2R_{T1} S_{11})^2 + (R_{R2} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_F S_{11})^2 + N_F^2$$

Equation 3: Forward Reflection Magnitude Uncertainty (RSS Computation)

$$\Delta S_{11(\text{mag})} = \sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}$$

where:

$$\text{Systematic}^2 = E_{ZF}^2 + E_{ZF}^2 S_{11}^2 + E_{ZF}^4 S_{11}^4 + S_{21}^2 S_{12}^2 (E_{ZF}^2 + 4E_{ZF}^2 E_{ZF}^2 S_{11}^2 + E_{ZF}^4 S_{22}^2) + A_M^2 S_{11}^2$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = C_{M1}^2 (1 + S_{11}^4) + 4C_{M1}^2 S_{11}^2 + C_{M2}^2 S_{21}^2 S_{12}^2$$

$$R^2 = R_{21}^2 (1 + S_{11}^4) + 4R_{21}^2 S_{11}^2 + R_{22}^2 S_{21}^2 S_{12}^2$$

$$\text{Noise}^2 = (N_F S_{11})^2 + N_F^2$$

Equation 4: Forward Reflection Phase Uncertainty (RSS Computation)

$$\Delta S_{11(\text{phase})} = \sqrt{\left(\sin^{-1} \left(\frac{\sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}}{S_{11}} \right) \right)^2 + 4C_{P1}^2}$$

where:

$$\text{Systematic}^2 = E_{ZF}^2 + E_{ZF}^2 S_{11}^2 + E_{ZF}^4 S_{11}^4 + S_{21}^2 S_{12}^2 (E_{ZF}^2 + 4E_{ZF}^2 E_{ZF}^2 S_{11}^2 + E_{ZF}^4 S_{22}^2) + \sin^2(A_P) S_{11}^2$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = C_{M1}^2 (1 + S_{11}^4) + 4C_{M1}^2 S_{11}^2 + C_{M2}^2 S_{21}^2 S_{12}^2$$

$$R^2 = R_{21}^2 (1 + S_{11}^4) + 4R_{21}^2 S_{11}^2 + R_{22}^2 S_{21}^2 S_{12}^2$$

$$\text{Noise}^2 = (N_F S_{11})^2 + N_F^2$$

Forward Transmission Uncertainty

Equation 5: Forward Transmission Magnitude Uncertainty (Worst Case Computation)

$$\Delta S_{21(\text{mag})} = \sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}$$

where:

$$\text{Systematic} = E_{ZF} + S_{21} (E_{ZF} + E_{ZF} S_{11} + E_{ZF} S_{11} + E_{ZF} E_{ZF} (S_{21} S_{12} + S_{11} S_{22}) + A_M)$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = S_{21}^2 (C_{M1}^2 + C_{M2}^2 + C_{M1}^2 S_{11}^2 + C_{M2}^2 S_{22}^2)$$

$$R^2 = S_{21}^2 (R_{T1} + R_{21} S_{11})^2 + (R_{T2} + R_{22} S_{22})^2$$

$$\text{Noise}^2 = (N_F S_{21})^2 + N_F^2$$

Equation 6: Forward Transmission Phase Uncertainty (Worst Case Computation)

$$\Delta S_{21(\text{worst})} = \sin^{-1} \left(\frac{\sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}}{S_{21}} \right) + C_{T21} + C_{T22}$$

where:

$$\text{Systematic} = E_{XF} + S_{21} (E_{TF} + E_{XF} S_{11} + E_{LF} S_{22} + E_{XF} E_{LF} (S_{21} S_{12} + S_{11} S_{22}) + \sin(A_F))$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = S_{21}^2 (C_{M1}^2 + C_{M2}^2 + C_{RM1}^2 S_{11}^2 + C_{RM2}^2 S_{22}^2)$$

$$R^2 = S_{21}^2 ((R_{T1} + R_{R1} S_{11})^2 + (R_{T2} + R_{R2} S_{22})^2)$$

$$\text{Noise}^2 = (N_T S_{21})^2 + N_F^2$$

Equation 7: Forward Transmission Magnitude Uncertainty (RSS Computation)

$$\Delta S_{21(\text{RSS})} = \sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}$$

where:

$$\text{Systematic}^2 = E_{XF}^2 + S_{21}^2 (E_{TF}^2 + E_{XF}^2 S_{11}^2 + E_{LF}^2 S_{22}^2 + E_{XF}^2 E_{LF}^2 (S_{21}^2 S_{12}^2 + S_{11}^2 S_{22}^2) + A_M^2)$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = S_{21}^2 (C_{M1}^2 + C_{M2}^2 + C_{RM1}^2 S_{11}^2 + C_{RM2}^2 S_{22}^2)$$

$$R^2 = S_{21}^2 (R_{T1}^2 + R_{R1}^2 S_{11}^2 + R_{T2}^2 + R_{R2}^2 S_{22}^2)$$

$$\text{Noise}^2 = (N_T S_{21})^2 + N_F^2$$

Equation 8: Forward Transmission Phase Uncertainty (RSS Computation)

$$\Delta S_{21(\text{phase})} = \sqrt{\left(\sin^{-1} \left(\frac{\sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}}{S_{21}} \right) \right)^2 + C_{T21}^2 + C_{T2}^2}$$

where:

$$\text{Systematic}^2 = E_{XF}^2 + S_{21}^2 \left(E_{TF}^2 + E_{XF}^2 S_{11}^2 + E_{LF}^2 S_{12}^2 + E_{XF}^2 E_{LF}^2 (S_{21}^2 S_{12}^2 + S_{11}^2 S_{22}^2) + \sin^2(A_F) \right)$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = S_{21}^2 \left(C_{M21}^2 + C_{M2}^2 + C_{M21}^2 S_{11}^2 + C_{M2}^2 S_{22}^2 \right)$$

$$R^2 = S_{21}^2 \left(R_{T1}^2 + R_{R1}^2 S_{11}^2 + R_{T2}^2 + R_{R2}^2 S_{22}^2 \right)$$

$$\text{Noise}^2 = (N_F S_{21})^2 + N_F^2$$

Reverse Transmission Uncertainty

Equation 9: Reverse Transmission Magnitude Uncertainty (Worst Case Computation)

$$\Delta S_{12(\text{mag})} = \sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}$$

where:

$$\text{Systematic} = E_{XR} + S_{12} \left(E_{TX} + E_{XR} S_{22} + E_{LR} S_{11} + E_{XR} E_{LR} (S_{21} S_{12} + S_{11} S_{22}) + A_M \right)$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = S_{12}^2 \left(C_{M21}^2 + C_{M2}^2 + C_{M21}^2 S_{11}^2 + C_{M2}^2 S_{22}^2 \right)$$

$$R^2 = S_{12}^2 \left(R_{T1} + R_{R1} S_{11} \right)^2 + \left(R_{T2} + R_{R2} S_{22} \right)^2$$

$$\text{Noise}^2 = (N_F S_{12})^2 + N_F^2$$

Equation 10: Reverse Transmission Phase Uncertainty (Worst Case Computation)

$$\Delta S_{12(\text{phase})} = \sin^{-1} \left(\frac{\sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}}{S_{12}} \right) + C_{T21} + C_{T2}$$

where:

$$\text{Systematic} = E_{XR} + S_{12} \left(E_{TX} + E_{XR} S_{22} + E_{LR} S_{11} + E_{XR} E_{LR} (S_{21} S_{12} + S_{11} S_{22}) + \sin(A_F) \right)$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = S_{12}^2 \left(C_{M21}^2 + C_{M2}^2 + C_{M21}^2 S_{11}^2 + C_{M2}^2 S_{22}^2 \right)$$

$$R^2 = S_{12}^2 \left(R_{T1} + R_{R1} S_{11} \right)^2 + \left(R_{T2} + R_{R2} S_{22} \right)^2$$

$$\text{Noise}^2 = (N_F S_{12})^2 + N_F^2$$

Equation 11: Reverse Transmission Magnitude Uncertainty (RSS Computation)

$$\Delta S_{12(\text{mag})} = \sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}$$

where:

$$\text{Systematic}^2 = E_{\text{SR}}^2 + S_{12}^2 \left(E_{\text{TR}}^2 + E_{\text{SR}}^2 S_{22}^2 + E_{\text{LR}}^2 S_{11}^2 + E_{\text{SR}}^2 E_{\text{LR}}^2 (S_{21}^2 S_{12}^2 + S_{11}^2 S_{22}^2) \right) + A_{\text{M}}^2$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = S_{21}^2 \left(C_{\text{MR1}}^2 + C_{\text{MR2}}^2 + C_{\text{MR1}}^2 S_{11}^2 + C_{\text{MR2}}^2 S_{22}^2 \right)$$

$$R^2 = S_{21}^2 \left(R_{\text{F1}}^2 + R_{\text{R1}}^2 S_{11}^2 + R_{\text{F2}}^2 + R_{\text{R2}}^2 S_{22}^2 \right)$$

$$\text{Noise}^2 = (N_{\text{r}} S_{21})^2 + N_{\text{f}}^2$$

Equation 12: Reverse Transmission Phase Uncertainty (RSS Computation)

$$\Delta S_{12(\text{phase})} = \sqrt{\left[\sin^{-1} \left(\frac{\sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}}{S_{12}} \right) \right]^2 + C_{\text{TR1}}^2 + C_{\text{TR2}}^2}$$

where:

$$\text{Systematic}^2 = E_{\text{SR}}^2 + S_{12}^2 \left(E_{\text{TR}}^2 + E_{\text{SR}}^2 S_{22}^2 + E_{\text{LR}}^2 S_{11}^2 + E_{\text{SR}}^2 E_{\text{LR}}^2 (S_{21}^2 S_{12}^2 + S_{11}^2 S_{22}^2) + \sin^2(A_{\text{p}}) \right)$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = S_{21}^2 \left(C_{\text{MR1}}^2 + C_{\text{MR2}}^2 + C_{\text{MR1}}^2 S_{11}^2 + C_{\text{MR2}}^2 S_{22}^2 \right)$$

$$R^2 = S_{21}^2 \left(R_{\text{F1}}^2 + R_{\text{R1}}^2 S_{11}^2 + R_{\text{F2}}^2 + R_{\text{R2}}^2 S_{22}^2 \right)$$

$$\text{Noise}^2 = (N_{\text{r}} S_{21})^2 + N_{\text{f}}^2$$

Reverse Reflection Uncertainty

Equation 13: Reverse Reflection Magnitude Uncertainty (Worst Case Computation)

$$\Delta S_{22(\text{worst})} = \sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}$$

where:

$$\text{Systematic} = E_{DR} + E_{RR}S_{22} + E_{SR}S_{22}^2 + S_{21}S_{12} (E_{LR} + 2E_{SR}E_{LR}S_{22} + E_{LR}^2S_{11}) + A_{2R}S_{22}$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = C_{RM2}^2 (1 + S_{22}^4) + 4C_{TM2}^2 S_{22}^2 + C_{RM1}^2 S_{21}^2 S_{12}^2$$

$$R^2 = (R_{R2} (1 + S_{22}^2) + 2R_{T2} S_{22})^2 + (R_{R1} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_T S_{22})^2 + N_F^2$$

Equation 14: Reverse Reflection Phase Uncertainty (Worst Case Computation)

$$\Delta S_{21(\text{phase})} = \sin^{-1} \left(\frac{\sqrt{(\text{Systematic} + \text{Stability})^2 + \text{Noise}^2}}{S_{22}} \right) + 2C_{TR2}$$

where:

$$\text{Systematic} = E_{DR} + E_{RR}S_{22} + E_{SR}S_{22}^2 + S_{21}S_{12} (E_{LR} + 2E_{SR}E_{LR}S_{22} + E_{LR}^2S_{11}) + \sin(A_F) S_{11}$$

$$\text{Stability} = \sqrt{C^2 + R^2}$$

$$C^2 = C_{RM2}^2 (1 + S_{22}^4) + 4C_{TM2}^2 S_{22}^2 + C_{RM1}^2 S_{21}^2 S_{12}^2$$

$$R^2 = (R_{R2} (1 + S_{22}^2) + 2R_{T2} S_{22})^2 + (R_{R1} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_T S_{22})^2 + N_F^2$$

Equation 15: Reverse Reflection Magnitude Uncertainty (RSS Computation)

$$\Delta S_{22(\text{mag})} = \sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}$$

where:

$$\text{Systematic}^2 = E_{DR}^2 + E_{RR}^2 S_{22}^2 + E_{SR}^2 S_{22}^4 + S_{21}^2 S_{12}^2 (E_{LR}^2 + 4E_{SR}^2 E_{LR}^2 S_{22}^2 + E_{LR}^4 S_{11}^2) + A_{2R}^2 S_{22}^2$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = C_{RM2}^2 (1 + S_{22}^4) + 4C_{TM2}^2 S_{22}^2 + C_{RM1}^2 S_{21}^2 S_{12}^2$$

$$R^2 = (R_{R2} (1 + S_{22}^2) + 2R_{T2} S_{22})^2 + (R_{R1} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_T S_{22})^2 + N_F^2$$

Equation 16: Reverse Reflection Phase Uncertainty (RSS Computation)

$$\Delta S_{22(\text{phase})} = \sqrt{\left[\sin^{-1} \left(\frac{\sqrt{\text{Systematic}^2 + \text{Stability}^2 + \text{Noise}^2}}{S_{22}} \right) \right]^2 + 4C_{TP2}^2}$$

where:

$$\text{Systematic}^2 = E_{DR}^2 + R_{RR}^2 S_{22}^2 + R_{SR}^2 S_{22}^4 + S_{21}^2 S_{12}^2 (R_{LR}^2 + 4E_{SR}^2 E_{LR}^2 S_{22}^2 + E_{LR}^4 S_{11}^2) + \sin^2(A_F) S_{22}^2$$

$$\text{Stability}^2 = C^2 + R^2$$

$$C^2 = C_{SM2}^2 (1 + S_{22}^4) + 4C_{DM2}^2 S_{22}^2 + C_{SM1}^2 S_{11}^2 S_{12}^2$$

$$R^2 = (R_{R2} (1 + S_{22}^2) + 2R_{T2} S_{22})^2 + (R_{R1} S_{21} S_{12})^2$$

$$\text{Noise}^2 = (N_T S_{22})^2 + N_F^2$$

Sources of Systematic Errors

The residual (after measurement calibration) **systematic errors** result from imperfections in the calibration standards.

For **reflection measurements**, the associated residual errors are:

- residual directivity
- residual source match
- residual load match
- residual reflection tracking

For **transmission measurements**, the additional residual errors are:

- residual crosstalk
- residual source match
- residual load match
- residual transmission tracking

The listing below shows the **abbreviations** used for residual systematic errors that are in the uncertainty equations.

- **EDF** = forward residual directivity
- **ESF** = forward residual source match
- **ERF** = forward residual reflection tracking
- **EXF** = forward crosstalk
- **ELF** = forward load match
- **ETF** = forward transmission tracking
- **EDR** = reverse residual directivity
- **ESR** = reverse residual source match
- **ERR** = reverse residual reflection tracking
- **EXR** = reverse crosstalk
- **ELR** = reverse load match
- **ETR** = reverse transmission tracking
- **AM** = magnitude dynamic accuracy
- **Ap** = phase dynamic accuracy

All measurements are affected by **dynamic accuracy**. Dynamic accuracy

includes: errors during internal self-calibration routines, gain compression in the microwave frequency converter (sampler) at high signal levels, errors generated in the synchronous detectors, localized non-linearities in the IF filter system, and from LO leakage into the IF signal paths.

Sources of Random Errors

The **random error** sources are

- noise
- connector repeatability
- interconnecting cable stability

There are two types of noise in any measurement system:

1. low level noise (noise floor)
2. high level noise (trace noise)

Low level noise is the broadband noise floor of the receiver which can be reduced through averaging or by changing the IF bandwidth.

High level noise or trace noise is due to the noise floor of the receiver, and the phase noise of the LO source inside the test set. It is worsened by reducing the IF bandwidth. Using a high stability 10 MHz time base can reduce high level noise.

Connector repeatability is the random variation encountered when connecting a pair of RF connectors. Variations in both reflection and transmission can be observed.

Cable stability is dependent on the cable used and the amount of cable movement between calibration and measurement.

The listing below shows the **abbreviations** used for random errors in the error models and uncertainty equations.

- **N_F** = noise floor
- **N_T** = trace noise
- **C_{R1}** = port 1 cable reflection stability
- **C_{TM1}** = port 1 cable magnitude transmission stability
- **C_{TP1}** = port 1 cable phase
- **C_{TM2}** = port 2 cable magnitude transmission stability
- **C_{TP2}** = port 2 cable phase transmission stability
- **R_{R1}** = port 1 connector reflection repeatability
- **R_{T1}** = port 1 connector transmission repeatability

transmission stability

- **C_{R2}** = port 2 cable reflection stability

- **R_{R2}** = port 2 connector reflection repeatability

- **R_{T2}** = port 2 connector transmission repeatability

Determining Expected System Performance

Improper connection techniques and contact surfaces can degrade measurement accuracy.

Proper connection techniques include using a torque wrench with proper torque limits, ensuring that the connector pin depths meet specifications, ensuring that the center conductor of sliding loads is properly set, and observing proper handling procedures for beadless airlines.

Contact surface errors are caused by improper cleaning procedures, scratches, worn plating, and rough seating. View more information on [connector care](#)

If proper connection techniques and connector care is observed, the following table provides an indication of connector repeatability.

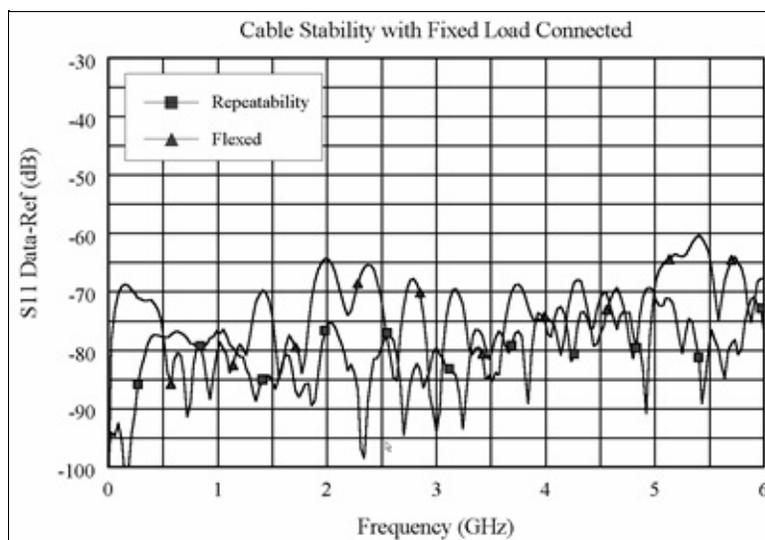
Connector Repeatability (R_{R1} , R_{R2} , R_{T1} , and R_{T2})				
Connector Type			Connector Type	
Frequency Range	Repeatability		Frequency Range	Repeatability
2.4-mm			3.5-mm	
0 to 2 GHz	0.0002		0 to 2 GHz	0.0001
2 to 20 GHz	0.0004		2 to 8 GHz	0.0003
20 to 36 GHz	0.0006		8 to 20 GHz	0.0006
36 to 40 GHz	0.0008		20 to 26.5 GHz	0.0010
7-mm			Type-N	
0 to 2 GHz	0.0001		0 to 2 GHz	0.0006
2 to 8 GHz	0.0003		2 to 8 GHz	0.0006
8 to 18 GHz	0.0006		8 to 18 GHz	0.0010
Type-F			Waveguide	
0 to 3 GHz	0.0006		0 to 40 GHz	0.0002

Determining Cable Stability Terms (C_{R1} , C_{R2} , C_{TM1} , C_{TM2} , C_{TP1} , C_{TP2})

Cable stability is dependent on the cable used and the amount of cable movement between calibration and measurement. Values for **cable reflection stability** are determined by connecting a fixed load to the free end of the cable and measuring the change in reflection coefficient after flexing the cable through the normal range of cable movement for a particular setup. **Cable transmission stability** is determined by connecting a short to the free end of the cable and measuring the change in reflection coefficient due to changes in cable position.

Graphics 1-3 demonstrate concepts useful in determining cable stability. In each case, a cable (part number 8120-4779) was connected to port 1, with a fixed load connected to the free end. A reference trace is obtained by measuring S_{11} with the free end held close to port 2 and storing the results in memory. Two additional S_{11} measurements are made; one with the cable flexed out to its straight position and the other with the cable positioned back to the same location as reference trace. As shown in Graphic 1, the flexed position demonstrates the effect of moving the cable after calibration. The repeatability trace demonstrates the stability of the cable when moved to its original position.

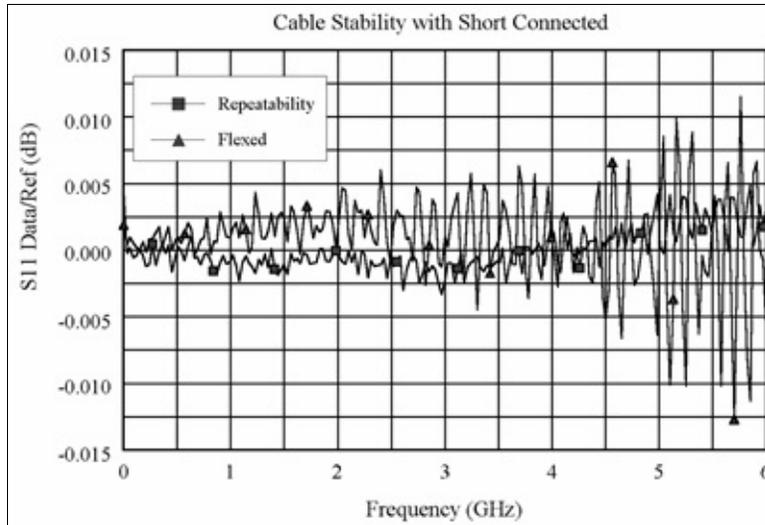
Graphic 1



Graphic 1 demonstrates the concepts useful in determining cable reflection stability. A fixed load is connected to the free end. The DATA-MEM feature

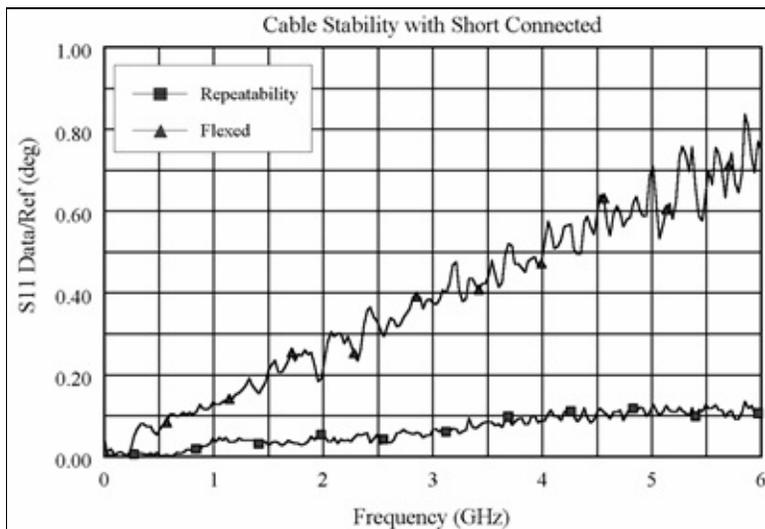
provides an indication of the cable reflection stability. A 60-dB peak on the chart yields a reflection stability estimated as $10^{(-60/20)}$ or 0.001.

Graphic 2



Graphic 2 and Graphic 3 demonstrate the concepts useful in determining cable transmission stability. A short is connected to the free end. The DATA/MEM feature provides an indication of the two-way cable transmission stability. The one-way transmission magnitude stability is determined by dividing the two-way magnitude measurement by two before it is converted to linear. A 0.013-dB peak on the chart yields transmission magnitude stability estimated as $10^{(0.013/40)}$ -1 or 0.00075. The one-way transmission phase stability is determined by dividing the two-way phase measurement by two.

Graphic 3



Cable movement often has a much larger effect on phase measurements than magnitude measurements.

See Also

[Measurement Errors](#)

[What is Measurement Calibration?](#)

[Why is Calibration Necessary?](#)

Last Modified:

3-Mar-2010 Added RSS uncerts

Get and Put Data: [Measurement](#) | [Cal](#) | [Power Cal](#) | [Custom](#) Other: [GPIB Pass-through](#) | [VISA Pass-through](#) | [Capabilities](#) | [Status/Events](#) | [FIFO and FastCW](#) | [Speed up Measurements!](#) | [Ground Loop](#)

Description	SCPI	
Get X-Axis values (variant)	SENS:X:VAL	
Get X-Axis values (typed)	N/A	
Get X-axis for trace	Calc:X?	
Get X-Axis values (Meas object)	N/A	
Get Measurement Data FROM the Analyzer		
Get receiver data	Calc:RDATA?	
Specifies ASCII or REAL type for data transfers	Format:Data	
Get complex or formatted data from the measurement or memory result buffer	Calc:Data	
Gets SnP data for the specified ports.	CALC:DATA:SNP:PORTS?	
Get ALL SnP data	Calc:Data:SnP	
Put Measurement Data INTO the Analyzer		
Put complex or formatted data into the measurement or memory result buffer	Calc:Data	
Get Calibration Data FROM the Analyzer		

Get variant Error Term data	SENS:CORR:CSET:DATA CSET:ETERm[:DATA]?	
Get variant Error Term data by text filter	CSET:ETERm:CATalog?	
Put Calibration Data INTO the Analyzer		
Put variant Error Term data	SENS:CORR:CSET:DATA CSET:ETERm[:DATA]	
Power Calibration Data		
Get variant cal data	SOUR:POW:CORR:DATA	
Get typed cal data	SOUR:POW:CORR:DATA	
Put variant cal data	SOUR:POW:CORR:DATA	
Put typed cal data	SOUR:POW:CORR:DATA	
Get and Put Custom Measurement Data		
Get and Put Custom data	Calc:Data	

Capabilities		
Many queries regarding the capability of a specific PNA		SYST:CAPabilities
Read installed options		*Opt?

Status Commands		
Status Registers		GP-IB/Status
*OPC;*WAI		GP-IB/Common_Commands

FIFO Data Buffer (N5264A Only)	
FIFO ON OFF	SYST:FIFO[:STATe]
Read number of data points	SYST:FIFO:DATA:COUNt?
Read data	SYST:FIFO:DATA?
Read data compact form	None
Clear data	SYST:FIFO:DATA:CLEAr

Other N5264A Commands	
FastCW	SENS:SWE:TYPE:FACW
Enable Point Averaging	SENS:AVER:MODE
Enable Point Sweep	SENS:SWE:GEN:POINtsweep
Set Trace Sweep	SENS:SWE:TRIG:MODE

Ground Loop De-embedding/Embedding commands	
De-embedding	
Sets and returns the Capacitance value	CALC:FSIM:GLOOp:DEEMbed:C
Sets and returns the Inductance value	CALC:FSIM:GLOOp:DEEMbed:L
Sets and returns the Resistance value	CALC:FSIM:GLOOp:DEEMbed:R
Turns ON or OFF De-embedding	CALC:FSIM:GLOOp:DEEMbed:STATe
Specifies the circuit model type	CALC:FSIM:GLOOp:DEEMbed:TYPE

Specifies the filename of the s1p file to load	CALC:FSIM:GLOop:DEEMbed:USER
Embedding	
Sets and returns the Capacitance value	CALC:FSIM:GLOop:EMBed:C
Sets and returns the Inductance value	CALC:FSIM:GLOop:EMBed:L
Sets and returns the Resistance value	CALC:FSIM:GLOop:EMBed:R
Turns ON or OFF Embedding	CALC:FSIM:GLOop:EMBed:STATe
Specifies the circuit model type	CALC:FSIM:GLOop:EMBed:TYPE
Specifies the filename of the s1p file to load	CALC:FSIM:GLOop:EMBed:USER

Equation Editor and MATLAB®

MATLAB can be used with Equation Editor in two different ways:

1. When you install a full version of MATLAB on the PNA, MATLAB functions can be called directly from Equation Editor.

1. Install the full 32 bit version of MATLAB.
2. On the PNA in the Equation Editor dialog, click enable MATLAB
3. You can then start calling MATLAB from within your equation editor entry field.
4. Here are a few example of how you would do this:
 - Matlab("S11.*S22") -> produces an array of multiplying S11*S22
 - Matlab("phase(S11)")-> produces an array of unwrapped phase of the S11 trace.

2. Compile a MATLAB *.dll using the full version on your PC. Then import the *.dll into Equation Editor

Learn how to Import Functions into Equation Editor

The MATLAB Compile Runtime on the PNA is currently 7.14, which is shipped with R2010b (7.11).

If you compile your *.dll using that version of MATLAB, then you don't need to change the version of MCR on the PNA.

1. Determine the version of MATLAB you will use.
 - You will need this version of MATLAB installed on your development machine (Step 2). You will also need a C++ compiler (Step 3).
 - You will need the appropriate version of MATLAB Compiler Runtime (MCR) installed on your target machines (Step 5).
 - To see installed version of MCR, check the following locations:
 - On a 32-bit Windows system:
 - C:\Program Files\Matlab\Matlab Compiler Runtime\
 - C:\Program Files\Matlab\Matlab Component Runtime\
 - On a 64-bit Windows system:
 - C:\Program Files (x86)\Matlab\Matlab Compiler Runtime\
 - C:\Program Files (x86)\Matlab\Matlab Component Runtime\
 - Use the chart below to compare MATLAB, MCR, and Compiler versions.
 - 2. Install MATLAB (32-bit) on your development machine.
 - If your development system is 64-bit, manually navigate to \bin\win32\ on your install disk and run setup.exe to use the 32-bit installer. This requirement will be removed in future releases.
 - You must install with the MATLAB Compiler toolbox.
 - 3. Install a compatible C++ compiler on your development machine.
 - For R2011b through R2013b, Windows SDK 7.1 with .NET 4.0 is sufficient.

- Due to issues with the SDK installer, it is recommended to separately install .NET first, if not already installed.
 - The current SDK installer will also fail if Visual Studio redistributables are installed.
 - For a list of compatible compilers, see <http://www.mathworks.com/support/compilers/>
 - Note that LCC is only a C compiler, not C++, and is not an acceptable option.
4. In MATLAB, select your installed compiler by running the command:
`>> mbuild -setup`
 5. Install the correct MCR on all target machines. This requires a reboot even if not prompted to do so by the installer.

Creating a MATLAB DLL

1. Open the MATLAB Compiler Deployment Tool, either through the user interface or with `>> deploytool`, and choose to make a "C++ Shared Library" project.
2. Add any .m files to your project that you need in the DLL through the Deployment Tool window. This includes any helpers for functions need to execute.
 - a. Functions must take in 0 to 32 arguments and return 1 value to be used on traces.
 - b. The return value can be an array the size of a trace or a single value.
 - c. Functions not conforming aren't directly accessible from Equation Editor but must be included if other functions rely on them.
3. Optional: Add function descriptors. These are separate functions which provide prototype (default) arguments for another function, named 'myFuncdesc' for every function.
 - a. These functions must take no arguments and return a single string, of the form:
`'arg1,arg2,...'` or `'arg1,arg2,...;tooltip'`
 where `arg1,arg2,...` is a list of arguments, and `tooltip` is an optional tooltip text to be used in future releases.

4. Build the project. This may take a few minutes and, if there are no errors, will generate project, src and distrib folders. The DLL will be under the distrib folder.
 - a. Some of the other generated files may be useful but are not needed for Equation Editor.
5. Optional: Package all the distributable files.
 - a. This package can also include the appropriate MCR installer (as large as 0.5 GB).
 - b. In general this step isn't needed, only the generated DLL is required.

Notes about Writing Scripts

The MATLAB functions that will be accessed directly by the Equation Editor must follow a specific format, as noted briefly above.

- Inputs: 0 to 32 vectors with dimensions [1, Sweep Size].
 - Constants (0, e, channel(), etc.) are expanded and passed as [1, Sweep Size] vectors.
- Outputs: 1 vector with dimensions [1, Sweep Size] or [1, 1].
 - Outputs size [1, 1] are expanded to [1, Sweep Size] vectors automatically.

Functions with other input/output sets are not directly accessible from Equation Editor but may be included in your DLL and used by other functions, meaning helpers can be used without risk.

Existing functions that do not fit these parameters can be included along with wrapper functions which convert the parameters and outputs within the MATLAB environment.

Last Modified:

12-Dec-2013 New topic

Print, Save, and Recall Data

- [Save and Recall a File](#)
- [Print a Displayed Measurement](#)
- Drive Mapping

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

Save /Recall | Manage Files | Manage Folders | Print | Read Clock

Description	SCPI	
Save / Recall		
Save Instrument States (*.csa, *.cst, *.sta, *.cal)	MMEMory:STORE	
Save Data (except snp)	MMEM:STOR:DATA	
Recall Files	MMEMory:LOAD	
Recall softkey list sort preference	SYST:PREF:ITEM:MRU	
Reads SNP data for the specified ports.	CALC:DATA:SNP:PORTs?	
Saves SNP data for the specified ports.	CALC:DATA:SNP:PORTs:Save	
Sets format for .SNP files	MMEM:STOR:TRAC:FORM:SNP	
Manage Files		
List Files	MMEMory:CATalog	
Copy Files	MMEMory:COPY	
Move Files	MMEMory:MOVE	
Delete Files	MMEMory:DELeTe	
Manage Folders		
Change	MMEMory:CDIRectory	
Delete	MMEMory:RDIRectory	
Make	MMEMory:MDIRectory	
Read directory location for the specified file type	SYST:CONF:DIR	
Print		
Print	HCOPY	
Saves image of PNA screen to file. (Print to File)	HCOPY:FILE	
Return the display image in arbitrary binary block	HCOPY:SDUMp:DATA?	
Set format of display image	HCOPY:SDUMp:DATA:FORM	

Read Date and Time	SCPI	
Read the last modified date of a Cal Set	CSET DATE?	
Read the last modified time of a Cal	CSET:TIME?	

Set		
Read the last modified date of a file	MMEM:DATE?	
Read the last modified time of a file	MMEM:TIME?	

[Save / Recall](#) | [Manage Files](#) | [Manage Folders](#) | [Print](#) | [Read Clock](#)

Description	SCPI	
Save / Recall		
Save Instrument States (*.csa, *.cst, *.sta, *.cal)	MMEMory:STORe	
Save Data (except snp)	MMEM:STOR:DATA	
Recall Files	MMEMory:LOAD	
Recall softkey list sort preference	SYST:PREF:ITEM:MRU	
Reads SNP data for the specified ports.	CALC:DATA:SNP:PORTs?	
Saves SNP data for the specified ports.	CALC:DATA:SNP:PORTs:Save	
Sets format for .SNP files	MMEM:STOR:TRAC:FORM:SNP	
Manage Files		
List Files	MMEMory:CATalog	
Copy Files	MMEMory:COpy	
Move Files	MMEMory:MOve	
Delete Files	MMEMory:DELeTe	
Manage Folders		
Change	MMEMory:CDIRectory	
Delete	MMEMory:RDIRectory	
Make	MMEMory:MDIRectory	
Read directory location for the specified file type	SYST:CONF:DIR	
Print		
Print	HCOPy	
Saves image of PNA screen to file. (Print to File)	HCOPy:FILE	
Return the display image in arbitrary binary block	HCOPy:SDUMp:DATA?	
Set format of display image	HCOPy:SDUMp:DATA:FORM	

Read Date and Time	SCPI	
Read the last modified date of a Cal Set	CSET DATE?	
Read the last modified time of a Cal Set	CSET:TIME?	
Read the last modified date of a file	MMEM:DATE?	
Read the last modified time of a file	MMEM:TIME?	

Learning about GPIB

The following topics can help you learn more about controlling the PNA using SCPI and the GPIB.

- COM versus SCPI
- **GPIB Fundamentals**
- **The Rules and Syntax of SCPI Commands**
- **Configure for SCPI / SICL over GPIB**
- **Configure for VISA / SICL over LAN**
- **Calibrating the PNA Using SCPI**
- **Getting Data from the PNA using SCPI**
- The PNA as a USB Device
- Remote Control of SCPI USB Devices Connected to a PNA
- **Reading the PNA Status Registers**
- **New** **Referring to Traces, Measurements, Channels, and Windows Using SCPI**
- **Synchronizing the PNA and Controller**
- **Shut Down the PNA Remotely**

See more PNA programming information and examples at:<http://na.support.keysight.com/pna/programming/>

Your Programs on Windows 7

Programs that run on Workstations

If your program is running on a separate workstation and talking to the PNA using SCPI or DCOM, you may encounter problems when your program saves data to the root directory of the PNA HDD. This is because of User Account Control (UAC), a new security model with Windows 7.

In the UAC default security setting, programs will not be able to save files to the root folder of the PNA. Try changing the UAC setting to the lowest security level. If it runs, but it is not acceptable to rewrite the program, then leave the UAC in this setting. The PNA software runs with UAC on either of these settings.

To access UAC settings:

1. Minimize the PNA application
2. Click the Windows **Start** button.
3. In **Search Programs and Files**, type **UAC**

Default setting

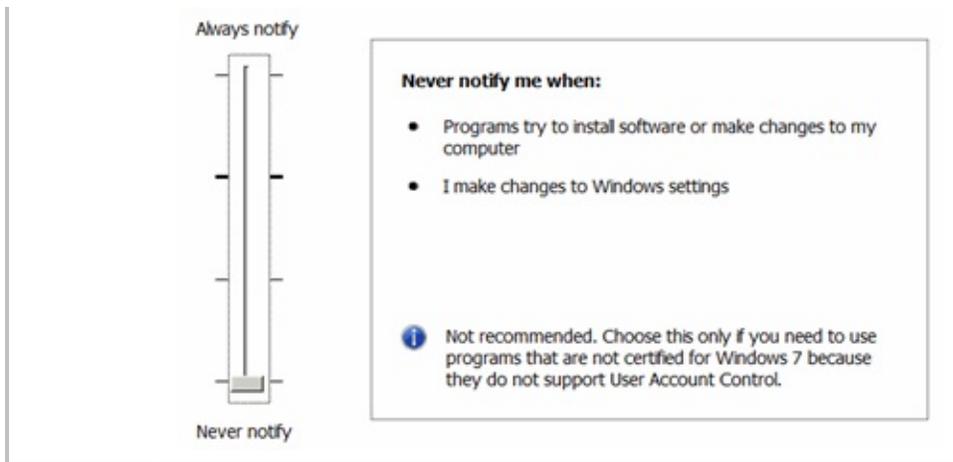


Default - Notify me only when programs try to make changes to my computer

- Don't notify me when I make changes to Windows settings

 Recommended if you use familiar programs and visit familiar websites.

Lowest security setting



Programs that run on the PNA

The following issues relate to problems that may occur when you install and run your programs on the PNA. Most programs that run in Windows XP will run without changes on Windows 7. However, there are differences between the two operating systems that could break more complicated programs. These incompatibilities are the same issues that software developers deal with on personal computers.

32-bit Programs

If your 32-bit program is saving files, Windows 7 will redirect memory access to correct for different file locations. If problems occur, the path locations may need to be modified. See new path locations.

16-bit Programs

On Windows XP, there was a compatibility layer for 16-bit programs. This compatibility layer has been removed. So, your very old 16-bit programs will no longer be able to run on the PNA.

Special note about .NET applications with 64-bit Windows

In Visual Studio 2008 and earlier, the default .NET platform target was AnyCPU. On Windows 7 64-bit, .NET applications that are written against the "AnyCPU" platform will compile down to a native 64-bit application. This can cause odd failures in your code as you will not be able to load any 3rd party libraries that are 32-bit only. You should always be compiling .NET applications as the "x86" platform. Starting with Visual Studio 2010, the default platform was changed to x86.

Last Modified:

9-Aug-2013 New topic (A.10.00)

Local Lockout (LLO) Message

Normally, a GPIB instrument is put in remote mode by asserting the Remote Enable (REN) GPIB line. At that time, all front panel keys (except the Local key) are locked to prevent user interaction.

Sending the LLO message over the GPIB further locks out the keyboard, mouse, and Local key during execution of your GPIB program. The syntax of the LLO message depends on the GPIB driver you are using. Consult your GPIB driver software users manual.

The PNA requires these two actions to occur in order:

1. Controller sends the LLO (Local Lockout) message
2. Controller asserts the REN (Remote Enable) GPIB line

The PNA will then go into remote mode with full lockout capability.

This feature is also supported using SICL over LAN.

Use the LocalLockoutState COM command when using TCPIP/LAN.

Last Modified:

7-Nov-2013 Added COM command

LXI Command

SCPI Command Tree

LXI:IDENtify[:STATe] <bool>

(Read-Write) Sets and returns the status of the LXI LAN status indicator on the LAN Status dialog.

Parameters

<bool> Choose from:
OFF or **0** - Changes the LXI Status indicator to 'NORMAL' and closes the dialog if it was opened by this command.
ON or **1** - Changes the LXI Status indicator to 'IDENTIFY' and opens the dialog if it was not already open.

Examples `LXI:IDEN 1`
`lxi:identify:state off`

Query Syntax LXI:IDENtify[:STATe]?

Return Type Boolean

Default OFF

SENSe:TEMPerature? <char>,<mod>

(Read-only) Returns the temperature on the Receiver microcircuit.

- To read temperature from the SFP, press **System**, then **Service**, then **Utilities**, then **Receiver Temperature**.
- The temperature reading is updated with every sweep.

Parameters

<char> Choose from **CELSius** or **FAHRenheit**

<mod> Module number. If unspecified, value is set to 1. [Learn how to determine module number.](#)

Examples `SENS:TEMP? CELS`
`sense:temperature? fahrenheit,2`

Return Type Numeric

Default Not Applicable

SYSTem:SERVice:PVERify Commands

Controls and queries settings for editing the Instrument Calibration Verification field on the Help About dialog. [Learn more](#).

SYSTem:SERVice:PVERify

INTerval

LAST

NEXT

Click on a keyword to view the command details.

see Also

- [SCPI Command Tree](#)
-

SYSTem:SERVice:PVERify:INTerval <days>,<mod>

(Read-Write) Sets and reads the interval between Instrument Calibrations. One year (365 days) is recommended.

Parameters

<days> (Integer) Interval in days.

<mod> (Integer) Slot number in which the module resides.

Examples

'Set 1 year interval

```
SYST:SERV:PVER:INT 365,1
```

Query Syntax SYSTem:SERVice:PVERify:INTerval? <mod>

Return Type Numeric

Default 0

SYSTem:SERVice:PVERify:LAST <date>,<mod>

(Read-Write) Sets and reads the date of the last Instrument Calibration.

Parameters

<date> Date in the form: yyyy,mm,dd

<mod> (Integer) Slot number in which the module resides.

Examples

'The following sets the date of the last calibration for the module in slot 1 to July 19, 2014

SYST:SERV:PVER:LAST 2014,07,19,1

The following reads the previous setting

SYST:SERV:PVER:LAST? 1

'2014,7,9

Query Syntax SYSTem:SERVice:PVERify:LAST? <mod>

Return Type Comma-separated integers

Default 0,0,0 (not set)

SYSTem:SERVice:PVERify:NEXT? <mod>

(Read-only) Reads the due date of the next Instrument Calibration.

Parameters

<mod> (Integer) Slot number in which the module resides.

Examples SYST:SERV:PVER:NEXT? 1

'returns: 2015,7,9

Return Type Numeric in the form: yyyy,mm,dd

Default Not Applicable

Calculate:FSimulator:Balun Commands

CALCulate:FSIMulator:BALun:

BPORt

| OFFSet

| PHASe

| POWer

| SWEep

| PHASe

| START

| STOP

CZConversion

| BPOrt

| IMAG

| REAL

| Z0

| R

| STATE

| LPOrt

| IMAG

| REAL

| Z0

| R

DEVice

DMCircuit

| BPOrt

| PARAmeters

| C

| G

| L

| R

| TYPE

| USER

| FILEname

| LPOrt

| PARAmeters

| C

| G

| L

| R

| TYPE

| USER

| **FILename**
| **STATe**
DZConversion
| **BPOrt**
| **IMAG**
| **REAL**
| **Z0**
| **R**
| **STATe**
| **LPOrt**
| **IMAG**
| **REAL**
| **Z0**
| **R**
FIXTure
| **OFFSet**
| **PHASe**
| **POWER**
| **SWEEp**
| **PHASe**
PARAmeter
| **BALSended**
| **[:DEFine]**
| **BBALanced**
| **[DEFine]**
| **SBALanced**
| **[DEFine]**
| **SSBalanced**
| **[DEFine]**
PHASe
| **SWEEp**
| **STATe**
STIMulus
| **MODE**
TOPology
| **BALSended**
| **[PPorts]**
| **BBALanced**
| **[PPorts]**
| **SBALanced**
| **[PPorts]**
| **SSBalanced**
| **[PPorts]**

[\[Keyword \]](#)

Click on a keyword to view the command details.

see Also

- [Example Programs](#)
- [Learn about Balanced Measurements](#)
- [Learn about iTMSA](#)
- [Synchronizing the Analyzer and Controller](#)
- [SCPI Command Tree](#)

Notes:

Critical Note: CALCulate commands act on the selected measurement. You can select one measurement for each channel using [Calc:Par:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

- [CALC:PAR:CAT?](#) alone can NOT be used to return a balanced measurement parameter. If a balanced measurement transform is being performed, then additional querying of the CALC:FSIM system is required to determine the balanced parameter type. [See an example](#).
- **BPORT versus LPORT commands** - For each command in this subsystem that includes a BPORT keyword, there is an LPORT equivalent. The commands are identical except for the way in which the balanced / logical port numbers are specified:
 - The BPORT commands refer to the Balanced port number. There can only be up to two balanced ports. This method is compatible with the ENA network analyzer.
 - The LPORT commands refer to the Logical port number. A balanced port can appear as either logical port 1, 2, or 3. These are the references as they appear in the front-panel user interface.

Topology	Logical Port	Balanced Port
Single-Bal	1	N/A
	2	1

Single-Single-Bal	1	N/A
	2	N/A
	3	1
Bal-Bal	1	1
	2	2

[Learn more about logical ports.](#)

CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:OFFSet: <value>

(Read-Write) Sets the phase offset between the two balanced stimulus ports. This command only applies when **CALC:FSIM:BAL:STIM:MOD** is set to a True Mode - Not Single-Ended. Requires Opt 460. Learn more about iTMSA Power and Phase offset.

[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.
- <pnum> Balanced port number. Choose from ports **1** or **2**.
- <value> Phase offset value in degrees.

Examples

```
CALC:FSIM:BAL:BPOR:OFFS:PHAS 10  
calculate2:fsimulator:balun:bport:offset:phase 300  
See example iTMSA program
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:OFFSet:PHASe?

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:OFFSet: <value>

(Read-Write) Sets the phase offset between the two balanced stimulus ports. This command only applies when **CALC:FSIM:BAL:STIM:MOD** is set to a True Mode - Not Single-Ended. Requires Opt 460. Learn more about iTMSA Power and Phase offset.

[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.
- <pnum>** Balanced port number. Choose from ports **1** or **2**.
- <value>** Power offset value in dB.

Examples

```
CALC:FSIM:BAL:BPOR:OFFS:POW 2  
calculate2:fsimulator:balun:bport:offset:power .2  
See example iTMSA program
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:OFFSet:POWer?

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:SWEep: <value>

(Read-Write) Sets the start value for a phase sweep.
Learn more about Phase Sweep.

[See Critical Note](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.
- <pnum> Balanced port number. Choose any PNA port. Only one port can have phase sweep.
- <value> Phase sweep start value in degrees. Choose a value between 0 and 360.

Examples

```
CALC:FSIM:BAL:BPOR:SWE:PHAS:STAR 10  
calculate2:fsimulator:balun:bport:sweep:phase:start 5
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:SWEep:PHASe:s

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:SWEep:<value>

(Read-Write) Sets the stop value for a phase sweep.

Learn more about Phase Sweep.

[See Critical Note](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.
- <pnum> Balanced port number. Choose any PNA port. Only one port can have phase sweep.
- <value> Phase sweep stop value in degrees. Choose a value between 0 and 360.

Examples

```
CALC:FSIM:BAL:BPOR:SWE:PHAS:STOP 10  
calculate2:fsimulator:balun:bport:sweep:phase:stop 5
```

Query CALCulate<cnum>:FSIMulator:BALun:BPORt<pnum>:SWEep:PHASe:s

Syntax

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:CZConversion:BPORt<pnum>:IMAG<value>

(Read-Write) Sets the imaginary part of the impedance value for the common port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<value> Imaginary part of the Impedance value in Units. Choose a number between -1E18 and 1E18.

Examples

```
CALC:FSIM:BAL:CZC:BPOR:IMAG 0
calculate2:fsimulator:balun:czconversion:bport:imag 300
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:CZConversion:BPORt<pnum>:IMAG<value>

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:CZConversion:BPORt<pnum>:REAL<value>

(Read-Write) Sets the real part of the impedance value for the common port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment.
[Learn more.](#)

<value> Real part of the Impedance value in Units. Choose a number between 0 and 1E18.

Examples

```
CALC:FSIM:BAL:CZC:BPOR:REAL 25
calculate2:fsimulator:balun:czconversion:bport:real 50
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:CZConversion:BPORt<pnum>:REAL

Return Type Numeric

Default See [Common Mode Port Z Conversion Default](#)

CALCulate<cnum>:FSIMulator:BALun:CZConversion:BPORt<pnum>:REAL<value>

(Read-Write) Sets the real part of 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.

See [Critical Note](#)

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<port> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assigned.
[Learn more.](#)

<value> Impedance value in ohms. Choose a number between 0 to 1E7.

Examples

```
CALC:FSIM:BAL:CZC:BPOR:Z0 50  
calculate2:fsimulator:balun:czconversion:bport:z0:r 75
```

Query Syntax CALCulate<num>:FSIMulator:BALun:CZConversion:BPORt<port>:Z

Return Type Numeric

Default See [Common Mode Port Z Conversion Default](#)

CALCulate<num>:FSIMulator:BALun:CZConversion:STATE <bool>

(Read-Write) Sets the common port impedance conversion function ON/OFF. Must also set the fixture simulator function to ON using [CALC:FSIM:STAT](#).

See [Critical Note](#)

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

<bool> State of common port impedance conversion function. Choose from
OFF (or 0) Conversion OFF
ON (or 1) Conversion ON

Examples `CALC:FSIM:BAL:CZC:STAT 1`
`calculate2:fsimulator:balun:czconversion:state off`

Query Syntax `CALCulate<cnum>:FSIMulator:BALun:CZConversion:STATE?`

Return Type Boolean

Default Off

CALCulate<cnum>:FSIMulator:BALun:CZConversion:LPORt<pnum>:IMAG<value>

(Read-Write) Sets the imaginary part of the impedance value for the common port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports **1**, **2**, or **3**.

Note: See [Balanced port versus Logical port](#).

<value> Imaginary part of the Impedance value in Units. Choose a number between -1E18 and 1E18.

Examples `CALC:FSIM:BAL:CZC:LPOR:IMAG 0`
`calculate2:fsimulator:balun:czconversion:lport:imag 300`

Query Syntax `CALCulate<cnum>:FSIMulator:BALun:CZConversion:LPORt<pnum>:IMAG<value>`

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:CZConversion:LPORt<pnum>:REAL<value>

(Read-Write) Sets the real part of the impedance value for the common port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<value> Real part of the Impedance value in Units. Choose a number between 0 and 1E18.

Examples

```
CALC:FSIM:BAL:CZC:LPOR:REAL 25  
calculate2:fsimulator:balun:czconversion:lport:real 50
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:CZConversion:LPORt<pnum>:R

Return Type Numeric

Default See [Common Mode Port Z Conversion Default](#)

CALCulate<cnum>:FSIMulator:BALun:CZConversion:LPORt<pnum>:REAL<value>

(Read-Write) Sets the real part of 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.

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.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<value> Impedance value in ohms. Choose a number between 0 to 1E7.

Examples

```
CALC:FSIM:BAL:CZC:LPOR:Z0 50
calculate2:fsimulator:balun:czconversion:lport:z0:r 75
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:CZConversion:LPORt<pnum>:Z

Return Type Numeric

Default See [Common Mode Port Z Conversion Default](#)

CALCulate<cnun>:FSIMulator:BALun:DEVICE <char>

(Read-Write) Selects the device type for the balanced 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.

<char> **BBALanced** - Balanced - Balanced device (4 ports).

BALSended - Balanced - Single-ended device (3 ports).

SBALanced - Single-ended - Balanced device (3 ports).

SSBalanced - Single-ended - Single-ended - Balanced device (4 ports).

Examples

```
CALC:FSIM:BAL:DEV SSB
calculate2:fsimulator:balun:device bbal
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:DEVICE?

Return Type Character

Default SBALanced

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum> <value>

(Read-Write) Sets the Capacitance value of the differential matching circuit.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<value> Capacitance value in farads. Choose a number between -1E18 to 1E18

Examples

```
CALC:FSIM:BAL:DMC:BPOR:PARAMeters:C 10E-6  
calculate2:fsimulator:balun:dmcircuit:bport:parameters:c 1E-9
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:PARAMeters:C <value>

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum> <value>

(Read-Write) Sets the Conductance value of the differential matching circuit.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<value> Conductance value in siemens. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:BPOR:PARAMeters:G 1E3  
calculate2:fsimulator:balun:dmcircuit:bport:parameters:g 1E-3
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:PARa

Return Type Numeric

Default 0

**CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:
<value>**

(Read-Write) Sets the Inductance value of the differential matching circuit.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment.
[Learn more.](#)

<value> Inductance value in henries. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:BPOR:PARAMeters:L 3E-3  
calculate2:fsimulator:balun:dmcircuit:bport:parameters:lE-10
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum> <value>

(Read-Write) Sets the Resistance value of the differential matching circuit.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment.
[Learn more.](#)

<value> Resistance value in ohms. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:BPOR:PARAMeters:R 100  
calculate2:fsimulator:balun:dmcircuit:bport:parameters:r 4E3
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>[:TYPE] <char>

(Read-Write) Sets the differential matching circuit type. To select a user-defined circuit, specify IN ADVANCE the 2-port touchstone filename with **CALC:FSIM:BAL:DMC:BPOR:USER:FILENAME**. If you do not specify the appropriate file and you select **USER**, an error occurs and **NONE** is automatically selected.

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.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<char> Circuit type. Choose from:

NONE - Specifies no-circuit.

PLPC - Specifies the circuit that consists of shunt L and shunt C.

USER - Specifies the user-defined circuit.

Examples

```
CALC:FSIM:BAL:DMC:BPOR2 PLPC  
calculate2:fsimulator:balun:dmcircuit:bport1:type none
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:TYPE

Return Type Character

Default PLPC

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:USER:FILENAME<string>

(Read-Write) Specifies the 2-port touchstone file in which the information on the user-defined differential matching circuit is saved. Following this command, send **CALC:FSIM:BAL:DMC:BPOR2 USER**. If the specified file does not exist, an error occurs and the type of differential matching circuit is set to USER.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This works the same as the ENA network analyzer. If there is only one balanced port, Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<string> File name and extension (.s2P) of the differential matching circuit. Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents/Document Recall". If you recall from a different folder, specify the full path name.

Examples

```
CALC:FSIM:BAL:DMC:BPOR:USER:FILENAME 'myfile.s2p'  
calculate2:fsimulator:balun:dmcircuit:bport:user:filename "C:/Program Files/Keysight/Network Analyzer/Documents/myFile.s2P"
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:BPORt<pnum>:USER:FILENAME<string>

Return Type String

Default Not Applicable

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:STATE<bool>

(Read-Write) Sets the differential matching circuit embedding function ON/OFF. Must also set the fixture simulator function to ON using **CALC:FSIM:STAT**.

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> State of differential matching circuit embedding function. Choose from
- OFF (or 0)** Matching circuit OFF
 - ON (or 1)** Matching circuit ON

Examples

```
CALC:FSIM:BAL:DMC:STAT 1
calculate2:fsimulator:balun:dmcircuit:state off
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:STATE?

Return Type Boolean

Default Off

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum><value>

(Read-Write) Sets the Capacitance value of the differential matching circuit.

See [Critical Note](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.
- <pnum> Logical port number. Choose from logical ports **1**, **2**, or **3**.

Note: See [Balanced port versus Logical port](#).

<value> Capacitance value in farads. Choose a number between -1E18 to 1E18

Examples

```
CALC:FSIM:BAL:DMC:LPOR:PARAMeters:C 10E-6  
calculate2:fsimulator:balun:dmcircuit:lport:parameters:c 1E-9
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum> <value>

(Read-Write) Sets the Conductance value of the differential matching circuit.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<value> Conductance value in siemens. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:LPOR:PARAMeters:G 1E3  
calculate2:fsimulator:balun:dmcircuit:lport:parameters:g 1E-3
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum> <value>

(Read-Write) Sets the Inductance value of the differential matching circuit.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<value> Inductance value in henries. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:LPOR:PARAMeters:L 3E-3  
calculate2:fsimulator:balun:dmcircuit:lport:parameters:IE-10
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum> <value>

(Read-Write) Sets the Resistance value of the differential matching circuit.

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<value> Resistance value in ohms. Choose a number between -1E18 to 1E18.

Examples

```
CALC:FSIM:BAL:DMC:LPOR:PARAMeters:R 100  
calculate2:fsimulator:balun:dmcircuit:lport:parameters:r 4E3
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:PARa

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>[:TYPE] <char>

(Read-Write) Sets the differential matching circuit type. To select a user-defined circuit, specify IN ADVANCE the 2-port touchstone filename with **CALC:FSIM:BAL:DMC:LPOR:USER:FILENAME**. If you do not specify the appropriate file and you select **USER**, an error occurs and **NONE** is automatically selected.

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.

<pnum> Logical port number. Choose from logical ports **1**, **2**, or **3**.

Note: See [Balanced port versus Logical port](#).

<char> Circuit type. Choose from:

NONE - Specifies no-circuit.

PLPC - Specifies the circuit that consists of shunt L and shunt C.

USER - Specifies the user-defined circuit.

Examples

```
CALC:FSIM:BAL:DMC:LPOR2 PLPC
```

```
calculate2:fsimulator:balun:dmcircuit:lport1:type none
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:TYPE

Return Type Character

Default PLPC

CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:TYPE<string>

(Read-Write) Specifies the 2-port touchstone file in which the information on the user-defined differential matching circuit is saved. Following this command, send **CALC:FSIM:BAL:DMC:BPOR2 USER**. If the specified file does not exist, an error occurs and the type of differential matching circuit is set to USER.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports 1, 2, or 3.

Note: See [Balanced port versus Logical port](#).

<string> File name and extension (.s2P) of the differential matching circuit. Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents". If you recall from a different folder, specify the full path name.

Examples

```
CALC:FSIM:BAL:DMC:LPOR:USER:FILE 'myfile.s2p'  
calculate2:fsimulator:balun:dmcircuit:lport:user:filename "C:/Program Files/Keysight/Network Analyzer/Documents/myFile.s2P"
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DMCircuit:LPORt<pnum>:USER:FILE

Return Type String

Default Not Applicable

CALCulate<cnum>:FSIMulator:BALun:DZConversion:BPORt<pnum>:IMAG<value>

(Read-Write) Sets the imaginary part of the impedance value for the differential port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnum> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment.

[Learn more.](#)

<value> Imaginary part of the Impedance value in Units. Choose a number between -1E18 and 1E18.

Examples

```
CALC:FSIM:BAL:DZC:BPOR:IMAG 0  
calculate2:fsimulator:balun:dczconversion:bport:imag 300
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DZConversion:BPORt<pnum>:IMAG<value>

Return Type Numeric

Default 0

CALCulate<cnum>:FSIMulator:BALun:DZConversion:BPORt<pnum>:REAL<value>

(Read-Write) Sets the real part of the impedance value for the differential port impedance

conversion function.

[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.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<value> Real part of the Impedance value in Units. Choose a number between 0 and 1E18

Examples

```
CALC:FSIM:BAL:DZC:BPOR:REAL 50
calculate2:fsimulator:balun:dzconversion:bport:real 75
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DZConversion:BPORt<pnum>:R

Return Type Numeric

Default See [Differential Port Z Conversion Default](#)

CALCulate<cnum>:FSIMulator:BALun:DZConversion:BPORt<pnum>:R<value>

(Read-Write) 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.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel.

that channel. If unspecified, <cnun> is set to 1.

<pnum> Balanced port number. Choose from ports **1** or **2**.

Note: The numbering of logical ports is different from balanced ports. This command works the same as the ENA network analyzer. If there is only one balanced port, it is Balanced Port 1, regardless of the port mapping assignment. [Learn more.](#)

<value> Impedance value in ohms. Choose a number between 0 to 1E7

Examples

```
CALC:FSIM:BAL:DZC:BPOR:Z0 50  
calculate2:fsimulator:balun:dzconversion:bport:z0:r 75
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:DZConversion:BPORt<pnum>:Z

Return Type Numeric

Default See [Differential Port Z Conversion Default](#)

CALCulate<cnun>:FSIMulator:BALun:DZConversion:STATE <bool>

(Read-Write) Sets the differential port impedance conversion function ON/OFF. Must also set the fixture simulator function to ON using [CALC:FSIM:STAT](#).

[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.

<bool> State of the differential port impedance conversion function. Choose from

OFF (or 0) Differential port impedance conversion OFF

ON (or 1) Differential port impedance conversion ON

Examples

```
CALC:FSIM:BAL:DZC:STAT 1  
calculate2:fsimulator:balun:dzconversion:state off
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:DZConversion:STATE?

Return Type Boolean

Default Off

CALCulate<cnun>:FSIMulator:BALun:DZConversion:LPORt<pnum>:IMAG<value>

(Read-Write) Sets the imaginary part of the impedance value for the differential port impedance conversion function.

See [Critical Note](#)

Parameters

<cnun> Channel number of the measurement. There must be a selected measurement for that channel. If unspecified, <cnun> is set to 1.

<pnum> Logical port number. Choose from logical ports **1, 2, or 3**.

Note: See [Balanced port versus Logical port](#).

<value> Imaginary part of the Impedance value in Units. Choose a number between -1E18 and 1E18.

Examples

```
CALC:FSIM:BAL:DZC:LPOR:IMAG 0  
calculate2:fsimulator:balun:dczconversion:lport:imag 300
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:DZConversion:LPORt<pnum>:IMAG<value>

Return Type Numeric

Default 0

CALCulate<cnun>:FSIMulator:BALun:DZConversion:LPORt<pnum>:IMAG<value>

(Read-Write) Sets the real part of the impedance value for the differential port impedance conversion function.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports **1**, **2**, or **3**.

Note: See [Balanced port versus Logical port](#).

<value> Real part of the Impedance value in Units. Choose a number between 0 and 1E18

Examples

```
CALC:FSIM:BAL:DZC:LPOR:REAL 50
calculate2:fsimulator:balun:dzconversion:lport:real 75
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DZConversion:LPORt<pnum>:R

Return Type Numeric

Default See [Differential Port Z Conversion Default](#)

CALCulate<cnum>:FSIMulator:BALun:DZConversion:LPORt<pnum>:REAL<value>

(Read-Write) 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.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.

<pnum> Logical port number. Choose from logical ports **1**, **2**, or **3**.

Note: See [Balanced port versus Logical port](#).

<value> Impedance value in ohms. Choose a number between 0 to 1E7

Examples

```
CALC:FSIM:BAL:DZC:LPOR:Z0 50  
calculate2:fsimulator:balun:dzconversion:lport:z0:r 75
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:DZConversion:LPORt<pnum>:Z

Return Type Numeric

Default See [Differential Port Z Conversion Default](#)

CALCulate<cnum>:FSIMulator:BALun:FIXTure:OFFSet:PHASe<bool>

(Read-Write) Sets and reads the state of "Phase Offset - Offset as Fixture" with True Mode balanced measurements.

Learn more about iTMSA phase and power offset.

[See Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected true mode balanced measurement on that channel. If unspecified, <cnum> is set to 1.

<bool> State of phase Offset as Fixture.

OFF (or 0) Offset is applied but is NOT included as a fixture in the output calculations.

ON (or 1) Offset is applied and included as a fixture in the output calculations.

Examples

```
CALC:FSIM:BAL:FIXT:OFFS:PHAS 0  
calculate2:fsimulator:balun:fixture:offset:phase on  
See example iTMSA program
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:FIXTureOFFSet:PHASe?

Return Type Boolean

Default Off

CALCulate<cnum>:FSIMulator:BALun:FIXTure:OFFSet:POWe <bool>

(Read-Write) Sets and reads the state of "Power Offset - Offset as Fixture" with True Mode balanced measurements.

Learn more about iTMSA phase and power offset.

See [Critical Note](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected true mode balanced measurement on that channel. If unspecified, <cnum> is set to 1.

<bool> State of power Offset as Fixture.

OFF (or 0) Offset is applied but is NOT included as a fixture in the output calculations.

ON (or 1) Offset is applied and included as a fixture in the output calculations.

Examples

```
CALC:FSIM:BAL:FIXT:OFFS:POW 0  
calculate2:fsimulator:balun:fixture:offset:power on  
See example iTMSA program
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:FIXTureOFFSet:POWer?

Return Type Boolean

Default Off

CALCulate<cnum>:FSIMulator:BALun:FIXTure:PHASe <bool>

(Read-Write) Sets and reads the state of "Phase Sweep - Offset as Fixture" (labeling on

GUI).

Learn more about iTMSA Phase Sweep.

[See Critical Note](#)

Parameters

- <cnun> Channel number of the measurement. There must be a selected true mode balanced measurement on that channel. If unspecified, <cnun> is set to 1.
- <bool> State of phase sweep offset as a fixture:
OFF (or 0) Phase Sweep offset disabled.
ON (or 1) Phase Sweep offset enabled.

Examples

```
CALC:FSIM:BAL:FIXT:PHAS 0  
calculate2:fsimulator:balun:fixture:phase on
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:FIXTure:PHASe?

Return Type Boolean

Default Off

CALCulate<cnun>:FSIMulator:BALun:PARAMeter<n>:BALSen<char>

(Read-Write) For a Balanced-Single-ended device type, selects the measurement parameter for the specified trace. Set device type using **CALC:FSIM:BAL:DEV**

[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.
- <n> Trace number on the specified channel <cnun>
- <char> Balanced - Single-ended Measurement parameter. Choose from:

--	--

Sdd11	Sdc11	Sds12
Scd11	Scs11	Scs12
Ssd21	Ssc21	Sss22
Imb	CMRR1 (Ssd21/Ssc21)	CMRR2 (Sds12/Scs12)

Examples

CALC:FSIM:BAL:PAR:BALS SDC11
calculate1:fsimulator:balun:parameter2:balsended:define imb

Query Syntax CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:BALSend[[:DE

Return Type Character

Default Sdd11

CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:BBALar <char>

(Read-Write) For a Balanced - Balanced device type, selects the measurement parameter : specified trace. Set device type using **CALC:FSIM:BAL:DEV**

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measureme that channel. If unspecified, <cnum> is set to 1.

<n> Trace number on the specified channel <cnum>

<char> Balanced- Balanced Measurement parameter. Choose from:

Sdd11	Sdd12	Sdc11	Sdc12
Sdd21	Sdd22	Sdc21	Sdc22

Scd11	Scd12	Scs11	Scs12
Scd21	Scd22	Scs21	Scs22
Imb1	Imb2	CMRR -(Sdd21/Scs21)	

Examples

```
CALC:FSIM:BAL:PAR:BBAL SDD12  
calculate1:fsimulator:balun: parameter2:bbalanced:define cmrr
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:BBALanced[:DE

Return Type Character

Default Sdd11

CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:SBALan <char>

(Read-Write) For a Single-ended - Balanced device type, selects the measurement parameter for the specified trace. Set device type using **CALC:FSIM:BAL:DEV**

See Critical Note

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.

<n> Trace number on the specified channel <cnum>

<char> Single-ended - Balanced Measurement parameter. Choose from:

Sss11	Ssd12	Scs12
Sds21	Sdd22	Sdc22
Scs21	Scd22	Scs22
Imb	CMRR1	CMRR2

	(Sds21/Scs21)	(Ssd12/Scs12)
--	---------------	---------------

Examples

CALC:FSIM:BAL:PAR:SBAL SSD12
calculate1:fsimulator:balun: parameter2:sbalanced:define imb

Query Syntax CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:SBALanced[:DE

Return Type Character

Default Sss11

**CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:SSBalan
 <char>**

(Read-Write) For a Single-ended - Single-ended - Balanced device type, selects the measurement parameter for the specified trace. Set device type using **CALC:FSIM:BAL:!**
 See [Critical Note](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement that channel. If unspecified, <cnum> is set to 1.
- <n> Trace number on the specified channel <cnum>
- <char> Single-ended - Single-ended - Balanced Measurement parameter. Choose f

Sss11	Sss12	Ssd13	Ssc13
Sss21	Sss22	Ssd23	Ssc23
Sds31	Sds32	Sdd33	Sdc33
Scs31	Scs32	Scd33	Scs33
Imb1	Imb2	CMRR1 (Sds31/Scs31)	CMRR2 (Sds32/Scs32)

Examples `CALC:FSIM:BAL:PAR:SSB SSD23`
`calculate1:fsimulator:balun: parameter2:ssbalanced:define imb1`

Query Syntax `CALCulate<cnum>:FSIMulator:BALun:PARAmeter<n>:SSBalanced[:DEL`

Return Type Character

Default Sss11

CALCulate<cnum>:FSIMulator:BALun:PARAmeter:STATE<bool>

(Read-Write) Turns balanced transform ON and 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.

<bool> State of balanced transform. Choose from
OFF (or 0) Balanced Transform OFF
ON (or 1) Balanced Transform ON

Examples `CALC:FSIM:BAL:PAR:STAT 1`
`calculate1:fsimulator:balun:parameter:state off`

Query Syntax `CALCulate<cnum>:FSIMulator:BALun:PARAmeter:STATE?`

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:BALun:PHASe:SWEep:STATE<bool>

(Read-Write) Sets and reads the state of phase sweep.

Learn more about iTMSA Phase Sweep.

[See Critical Note](#)

Parameters

<cnun> Channel number of the measurement. There must be a selected true mode balanced measurement on that channel. If unspecified, <cnun> is set to 1.

<bool> State of phase sweep:
OFF (or 0) Phase Sweep disabled.
ON (or 1) Phase Sweep enabled.

Examples

```
CALC:FSIM:BAL:PHAS:SWE:STAT 0  
calculate2:fsimulator:balun:phase:sweep:state on
```

Query Syntax CALCulate<cnun>:FSIMulator:BALun:PHASe:SWEep:STATE?

Return Type Boolean

Default Off

CALCulate<cnun>:FSIMulator:BALun:STIMulus:MODE <value>

(Read-Write) Sets the stimulus mode of the PNA source. True Mode settings requires [Opt 460](#).

Learn more about iTMSA.

[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.

<value> Stimulus mode. When a True-Mode is selected, the Balanced port powers are automatically uncoupled. Choose from:
SE - Single-Ended stimulus

TM - True-Mode stimulus
FTM - Forward only True-Mode stimulus
RTM - Reverse only True-Mode stimulus

Examples

```
CALC:FSIM:BAL:STIM:MODE SE  
calculate2:fsimulator:balun:stimulus:mode rtm  
See example program
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:STIMulus:MODE?

Return Type Character

Default SE

CALCulate<cnum>:FSIMulator:BALun:TOPology:BALSended[:<bPos>,<bNeg>,<se>]

(Read-Write) For a Balanced - Single-ended device type, maps the PNA ports to the DUT ports.

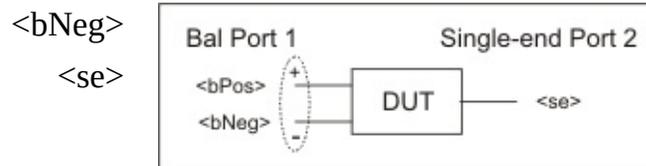
Set the Balanced - Single-ended device type using **CALC:FSIM:BAL:DEV**

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.

<bPos> PNA port number that connects to each of the following DUT ports:



Examples

```
CALC:FSIM:BAL:TOP:BALS 1,2,3  
calculate1:fsimulator:balun:topology:balsended:pports 4,3,2
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:TOPology:BALSended[:PPORTs]

Return Type Numeric - Returns three numbers separated by commas.

Default Not Applicable

CALCulate<cnum>:FSIMulator:BALun:TOPology:BBALanced[:<p1Pos>,<p1Neg>,<p2Pos>,<p2Neg>

(Read-Write) For a Balanced - Balanced device type, maps the PNA ports to the DUT ports.

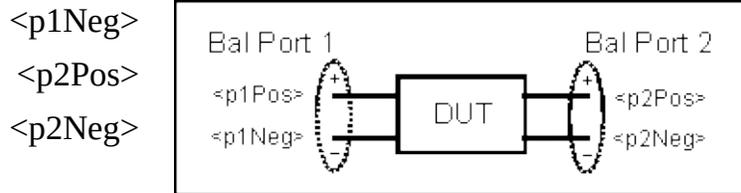
Set the Balanced - Balanced device type using [CALC:FSIM:BAL:DEV](#)

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.

<p1Pos> PNA port number that connects to each of the following DUT ports:



Examples

```
CALC:FSIM:BAL:TOP:BBAL 1,2,3,4
```

```
calculate1:fsimulator:balun: topology:bbalanced:pports 4,3,2,1
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:TOPology:BBALanced[:PPORTs]

Return Type Numeric - Returns four numbers separated by commas.

Default Not Applicable

CALCulate<cnum>:FSIMulator:BALun:TOPology:SBALanced[:<se>,<bPos>,<bNeg>

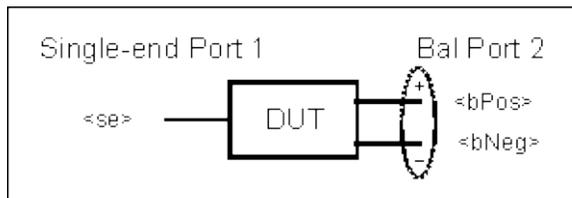
(Read-Write) For a Single-ended - Balanced device type, maps the PNA ports to the DUT ports.

Set the Single-ended - Balanced device type using **CALC:FSIM:BAL:DEV**

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.
- <se> PNA port number that connects to each of the following DUT ports:
- <bPos>
- <bNeg>



Examples

```
CALC:FSIM:BAL:TOP:SBAL 1,2,3
```

```
calculate1:fsimulator:balun: topology:sbalanced:pports 4,3,2
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:TOPology:SBALanced[:PPORTs]

Return Type Numeric - Returns three numbers separated by commas.

Default Not Applicable

CALCulate<cnum>:FSIMulator:BALun:TOPology:SSBalanced[:<se1>,<se2>,<bPos>,<bNeg>

(Read-Write) For a Single-ended - Single-ended - Balanced device type, maps the PNA ports to the DUT ports.

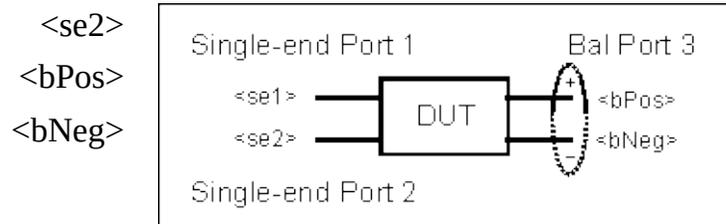
Set the Single-ended - Single-ended - Balanced device type using

CALC:FSIM:BAL:DEV

See [Critical](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <se1> PNA port number that connects to each of the following DUT ports:



Examples

```
CALC:FSIM:BAL:TOP:SSB 1,2,3,4
```

```
calculate1:fsimulator:balun:topology:ssbalanced:pports 4,3,2,1
```

Query Syntax CALCulate<cnum>:FSIMulator:BALun:TOPology:SSBalanced[:PPORts]

Return Type Numeric - Returns four numbers separated by commas.

Default Not Applicable

Last Modified:

25-Apr-2012 Added SE>Bal top (9.70)

3-Jun-2008 Added iTMSA commands

Calculate:FSimulator:Embed Commands

Specifies settings for embedding and de-embedding balanced (4-port) fixturing circuits.

CALCulate:FSIMulator:EMBed:

NETWork:

| **FILEname**

| **PMAP**

| **TYPE**

STATe

TOPology:

| **A:PORTs**

| **B:PORTs**

| **C:PORTs**

TYPE

Click a [blue](#) keyword to view the command details.

See Also

- [Example Programs](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

**CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:FILEnam
<string>**

(Read-Write) Specifies the 4-port touchstone file (*.s4p) in which the network to embed or de-embed resides. Following this command, send CALC:FSIM:EMB:NETW:TYPE. If the specified file does not exist, an error occurs

when type command is sent.

[Learn about 4-port network embedding.](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <n> Network position. Choose from 1 or 2.
- <string> File name and extension (.s4P) of the circuit. Files are stored in the default folder "C:/Program Files/Keysight/Network Analyzer/Documents". To recall from a different folder, specify the full path name.

Examples

```
10 calculate2:fsimulator:embed:network2:filename  
"c:\users\public\network analyzer\documents\myFile.s4P"  
20 calculate2:fsimulator:embed:network2:type embed
```

Query Syntax CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:FILename?

Return Type String

Default Not Applicable

CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:PMAP <inA>,<inB>,<outA>,<outB>

(Read-Write) Set and return the port mapping for a 4-port SNP file to be embedded.

[Learn about 4-port network embedding.](#)

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <n> Network position. Choose from 1 or 2.
- <inA> <inB> <outA> <outB> Port Mapping. Use four port numbers in any order.

Example `CALC:FSIM:EMB:NETW1:PMAP 1,3,2,4`

Query Syntax `CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:PMAP?`

Return Type Comma-separated numeric

Default 1,2,3,4

CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:TYPE <char>

(Read-Write) Specify the type of processing to take place on the specified 4-port network. First specify the network filename with `CALC:FSIM:EMB:NETW:FIL`.

[Learn about 4-port network embedding.](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<n> Network position. Choose from **1** or **2**.

<char> Processing type. Choose from :
NONE - The same as disabling.
EMBed - Add Network circuit.
DEEMbed - Remove Network circuit.

Example `10 CALC:FSIM:EMB:NETW2:FIL 'myFile.s4p'`
`20 CALC:FSIM:EMB:NETW2:TYPE EMBed`

Query Syntax `CALCulate<cnum>:FSIMulator:EMBed:NETWork<n>:TYPE?`

Return Type Character

Default NONE

CALCulate<cnum>:FSIMulator:EMBed:STATE <bool>

(Read-Write) Turns ON or OFF 4-port Network Embedding/De-embedding for all ports 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 4-port Network Embedding/De-embedding ON

OFF or 0 - Turns 4-port Network Embedding/De-embedding OFF

Examples

```
CALC:FSIM:EMB:STAT 1
calculate2:fsimulator:embed:state 0
```

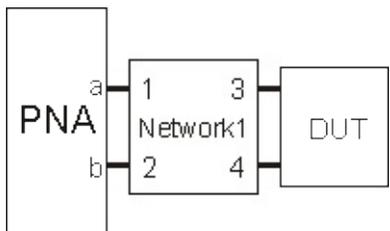
Query Syntax CALCulate<cnum>:FSIMulator:EMBed:STATe?

Return Type Boolean

Default OFF

CALCulate<cnum>:FSIMulator:EMBed:TOPology:A:PORTs <p1>,<p2>

(Read-Write) Specifies the PNA port connections when topology A is selected. Specify topology using **CALC:FSIM:EMBed:TYPE**.



Topology A

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<p1> PNA Port number assigned to **a** in above graphic.

<p2> PNA Port number assigned to **b** in above graphic.

Examples

```
CALC:FSIM:EMB:TOP:A:PORT 2,1  
calculate2:fsimulator:embed:topology:a:ports 1,2
```

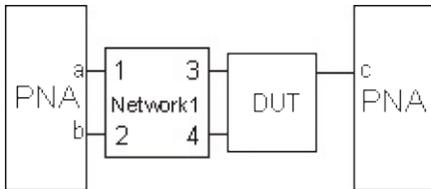
Query Syntax CALCulate<cnum>:FSIMulator:EMBed:TOPology:A:PORTs?

Return Type Numeric

Default 1,2

CALCulate<cnum>:FSIMulator:EMBed:TOPology:B:PORTs <p1>,<p2>,<p3>

(Read-Write) Specifies the PNA port connections when topology B is selected. Specify topology using **CALC:FSIM:EMBed:TYPE**.



Topology B

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<p1> PNA Port number assigned to **a** in above graphic.

<p2> PNA Port number assigned to **b** in above graphic.

<p3> PNA Port number assigned to **c** in above graphic.

Examples

```
CALC:FSIM:EMB:TOP:B:PORT 2,1,4
```

calculate2:fsimulator:embed:topology:b:ports 1,2,3

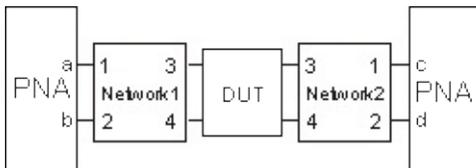
Query Syntax CALCulate<cnum>:FSIMulator:EMBed:TOPology:B:PORTs?

Return Type Numeric

Default 1,2,3

CALCulate<cnum>:FSIMulator:EMBed:TOPology:C:PORTs <p1>,<p2>,<p3>,<p4>

(Read-Write) Specifies the PNA port connections when topology C is selected. Specify topology using **CALC:FSIM:EMBed:TYPE**.



Topology C

Parameters

- <cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.
- <p1> PNA Port number assigned to **a** in above graphic.
- <p2> PNA Port number assigned to **b** in above graphic.
- <p3> PNA Port number assigned to **c** in above graphic.
- <p4> PNA Port number assigned to **d** in above graphic.

Examples

CALC:FSIM:EMB:TOP:C:PORT 2,1,4,3
calculate2:fsimulator:embed:topology:c:ports 1,2,3,4

Query Syntax CALCulate<cnum>:FSIMulator:EMBed:TOPology:C:PORTs?

Return Type Numeric

Default 1,2,3,4

CALCulate<cnum>:FSIMulator:EMBed:TYPE <char>

(Read-Write) Specifies the PNA / DUT topology. [Learn more about these and other PNA/DUT configurations.](#)

Parameters

<cnum> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <cnum> is set to 1.

<char> PNA / DUT topology. Choose from:

- **A** - 2 PNA/DUT Ports
- **B** - 3 PNA/DUT Ports
- **C** - 4 PNA/DUT Ports

Click links to see topologies and port assignment commands.

Examples

```
CALC:FSIM:EMB:TYPE A  
calculate2:fsimulator:embed:type c
```

Query Syntax CALCulate<cnum>:FSIMulator:EMBed:TYPE?

Return Type Character

Default A

Calculate:Marker:PNOP Commands

Initiates a Power Normal Operating Point marker search and reads the results.

CALCulate:MARKer:PNOP

BACKoff

| **GAIN?**

| **PIN?**

| **POUT?**

COMPression?

| **MAXimum?**

GAIN?

| **MAXimum?**

PIN?

| **MAXimum?**

POFFset

POUT?

| **MAXimum?**

Click on a keyword to view the command details.

See Also

- [PNOP Example](#)
- [Learn about PNOP Markers](#)
- [Other SCPI Marker commands](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more.](#)

CALCulate<cnum>:MARKer:PNOP:BACKoff <num>

(Read-Write) Turns on and sets markers 1, 2, 3, and 4 to calculate various PNOP parameters.

Either this command, or the **POFFset** command, will initiate the PNOP search markers. To turn off the PNOP markers, either turn them off individually or turn them **All Off**.

To search a User Range with the PNOP search, first activate marker 1 and set the desired **User Range**. Then send CALC:MARK:PNOP:BACK. 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.

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> Backoff value. Choose any number between **:-500** and **500**

Examples

```
CALC:MARK:PNOP:BACK?  
calculate2:marker:pnop:backoff 10
```

Query Syntax CALCulate<cnum>:MARKer:PNOP:BACKoff?

Return Type Numeric

Default ??

CALCulate<cnum>:MARKer:PNOP:BACKoff:GAIN?

(Read-only) Reads the power backoff gain value from a PNOP marker search.

PBO Gain = PBO Out - PBO In

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples **CALC:MARK:PNOP:BACK:GAIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:BACKoff:PIN?

(Read-only) Reads the power backoff input value from a PNOP marker search.

PBO In = Marker 2 X-axis

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples **CALC:MARK:PNOP:BACK:PIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:BACKoff:POUT?

(Read-only) Reads the power backoff output value from a PNOP marker search.

PBO Out = Marker 2 Y-axis

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:BACK:POUT?`

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:COMPression?

(Read-only) Reads the PNOP compression value from a PNOP marker search.

$\text{Pnop Comp} = \text{Pnop Gain} - \text{Linear Gain}$ (not shown on marker readout).

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:COMP?`

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:COMPression:MAXimum?

(Read-only) Reads the max compression value from a PNOP marker search.

Comp Max = Gain Max - Linear Gain (not shown on marker readout).

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples **CALC:MARK:PNOP:COMP:MAX?**

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:GAIN?

(Read-only) Reads the PNOP gain value from a PNOP marker search.

$\text{Pnop Gain} = \text{Pnop Out} - \text{Pnop In}$.

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:GAIN?`

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:GAIN:MAXimum?

(Read-only) Reads the max gain from a PNOP marker search.

Gain Max = PMax Out - PMax In

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples **CALC:MARK:PNOP:GAIN:MAX?**

Default Not applicable

CALCulate<num>:MARKer:PNOP:PIN?

(Read-only) Reads the PNOP input value from a PNOP marker search.

Pnop In = Marker 4 X-axis value

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP search.

See **Critical Note**

Parameters

<num> Channel number of the measurement. There must be a selected measurement on that channel. If unspecified, <num> is set to 1.

Examples **CALC:MARK:PNOP:PIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:PIN:MAXimum?

(Read-only) Reads the max input power from a PNOP marker search.

PMax In = Marker 3 X-axis value

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:PIN:MAX?`

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:POFFset <num>

(Read-Write) Turns on and sets markers 1, 2, 3, and 4 to calculate various PNOP parameters.

Either this command, or the **Backoff** command, will initiate the PNOP search markers. To turn off the PNOP markers, either turn them off individually or turn them **All Off**.

To search a User Range with the PNOP search, first activate marker 1 and set the desired **User Range**. Then send the CALC:MARK:PNOP:POFF command. 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.

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> Power Offset value in dB. Choose any number between **:-500** and **500**

Examples

```
CALC:MARK:PNOP:POFF 3  
calculate2:marker:pnop:poffset 10
```

Query Syntax CALCulate<cnum>:MARKer:PNOP:POFFset?

Return Type Numeric

Default ??

CALCulate<cnum>:MARKer:PNOP:POUT?

(Read-only) Reads the output power value of the offset marker from a PNOP marker search.

Pnop Out = Marker 4 Y-axis value

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:POUT?`

Default Not applicable

CALCulate<cnum>:MARKer:PNOP:POUT:MAXimum?

(Read-only) Reads the max output power from a PNOP marker search.

PMax Out = Marker 3 Y-axis value

Use **CALC:MARK:PNOP:BACK** or **CALC:MARK:PNOP:POFF** to initiate a PNOP 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.

Examples `CALC:MARK:PNOP:POUT:MAX?`

Default Not applicable

Calculate:Marker:PSaturation Commands

Initiates a Power Saturation marker search and reads the results.

CALCulate:MARKer:PSATuration

BACKoff

COMPression

| **MAXimum?**

| **SATuration?**

GAIN?

| **LINear?**

| **MAXimum?**

PIN?

| **MAXimum?**

POUT?

| **MAXimum?**

Click on a keyword to view the command details.

See Also

- [PSAT Example](#)
- [Learn about PSAT Markers](#)
- [Other SCPI Marker commands](#)
- [Synchronizing the Analyzer 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:MNUM](#) or [Calc:Par>Select](#). [Learn more](#).

CALCulate<cnum>:MARKer:PSATuration:BACKoff <num>

(Read-Write) Turns on and sets markers 1, 2, and 3 to calculate various Power Saturation parameters.

The <num> parameter sets and reads the back-off value for a Power Saturation marker search.

To turn off the Power Saturation markers, either turn them off individually or turn them **All Off**.

To search a User Range with the PSAT search, first activate marker 1 and set the desired **User Range**. Then send the CALC:MARK:PSAT:BACK 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.

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> Backoff value. Choose any number between **:-500** and **500**

Examples

```
CALC:MARK:PSAT:BACK 3  
calculate2:marker:psaturation:backoff 10
```

Query Syntax CALCulate<cnum>:MARKer:PSATuration:BACKoff?

Return Type Numeric

Default 0

CALCulate<cnum>:MARKer:PSATuration:COMPression:MAXi

(Read-only) Reads the compression maximum value from a PSAT marker search.

Comp Max = Gain Max - Gain Linear

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:COMP:MAX?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:COMPression:SATur

(Read-only) Reads the compression saturation value from a PSAT marker search.

Comp Sat = Gain Sat - Gain Linear

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:COMP:SAT?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:GAIN?

(Read-only) Reads the saturation gain value from a PSAT marker search.

Gain Sat = Psat Out - Psat In

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:GAIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:GAIN:LINear?

(Read-only) Reads the linear gain value from a PSAT marker search.

Gain Linear = Marker 1 - Y-axis value MINUS X-axis value.

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:GAIN:LIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:GAIN:MAXimum?

(Read-only) Reads the maximum gain value from a PSAT marker search.

Gain Max = PMax Out - PMax In

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:GAIN:MAX?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:PIN?

(Read-only) Reads the power saturation input value from a PSAT marker search.

Psat In = Marker 2 X-axis value

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:PIN?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:PIN:MAXimum?

(Read-only) Reads the maximum input power from a PSAT marker search.

PMax In = Marker 3 X-axis value

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples

```
CALC:MARK:PSAT:PIN:MAX?
```

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:POUT?

(Read-only) Reads the back-off output power from a PSAT marker search.

PSat Out = Marker 2 Y-axis value

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:POUT?**

Default Not applicable

CALCulate<cnum>:MARKer:PSATuration:POUT:MAXimum?

(Read-only) Reads the back-off output power from a PSAT marker search.

PMaxOut = Marker 3 Y-axis value

Use **CALC:MARK:PSAT:BACK** to initiate a PSAT 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.

Examples **CALC:MARK:PSAT:POUT:MAX?**

Default Not applicable

Last Modified:

18-Feb-2010 MX New topic

Set ECal States

This example cycles through the state settings on the first ECal module it finds on the USB bus.

The state settings include all of the ECal states on Port A, Port B and the AB thru path. The first state on a port-pair path such as AB is the thru state that is used during calibrations. The second state on that path is the "confidence state" which is the equivalent of an attenuator that is used by the ECal Confidence Check feature.

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 ECal State commands](#)

See Other SCPI Example Programs

```
Option Explicit  
Dim app  
Set app = CreateObject("AgilentPNA835x.Application")  
Dim scpi  
Set scpi = app.ScpiStringParser  
Dim moduleIndexList  
' These are 1-based indices as opposed to 0-based,  
' so if this query returns 0 it indicates there appear  
' to be no ECal modules connected.  
moduleIndexList = Split( scpi.Parse("SENS:CORR:CKIT:ECAL:LIST?"), ",")  
If CInt(moduleIndexList(0)) = 0 Then  
    MsgBox "No ECal module was found"
```

```
WScript.Quit(0)
End If
SetStates("A")
SetStates("B")
SetStates("AB")
MsgBox "Done"
Sub SetStates(path)
    Dim pathNumStates
    pathNumStates = CInt( scpi.Parse("CONT:ECAL:MOD1:PATH:COUN? " +
path) )
    Dim stateNum
    For stateNum = 1 To pathNumStates
        Dim stateNumStr
        stateNumStr = CStr(stateNum)
        Dim pathDescr
        If Len(path) = 1 Then
            pathDescr = "port " + path
        Else
            pathDescr = "path " + path
        End If
        Dim isOK
        isOK = MsgBox("Click OK to switch to state number " + stateNumStr + " of " +
pathDescr, vbOKCancel)
        If isOK = vbCancel Then WScript.Quit(0)
        scpi.Parse "CONT:ECAL:MOD1:PATH:STAT " + path + "," +
stateNumStr
    Next
End Sub
```

Last Modified:

16-Mar-2011 New topic

Display Color Commands

Controls the color settings of the PNA display.

DISPlay:COLor

[ABACkground](#)

[BACKground](#)

[GRAT1](#)

[GRAT2](#)

[ILABel](#)

[LIM1](#)

[LOAD](#)

[RESet](#)

[STORe](#)

TRACe

| **[DATA](#)**

| **[MARKer](#)**

| **[MEMory](#)**

| **[MMARker](#)**

Click on a keyword to view the command details.

see Also

- [Synchronizing the Analyzer and Controller](#)
 - [Learn about Display and Print Colors](#)
 - [SCPI Command Tree](#)
-

DISPlay:COLor<n>:ABACkground <num, num, num>

(Read-Write) Set and return the background color for the active window on the PNA display or hardcopy print.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:ABAC 10,10,10
```

```
display:color1:abackground 80,80,80
```

Query Syntax DISPlay:COLor<n>:ABACkground?

Return Type Numeric (n,n,n)

Default Display = 0,0,24 (Black)

Print = 255,255,255 (White)

DISPlay:COLor<n>:BACKground <num, num, num>

(Read-Write) Set and return the background color for the inactive windows on the PNA display or hardcopy print.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:BACK 10,10,10
```

```
display:color1:background 80,80,80
```

Query Syntax DISPlay:COLor<n>:BACKground?

Return Type Numeric (n,n,n)

Default Display = 0,0,0 (Black)

Print = 255,255,255 (White)

DISPlay:COLor<n>:GRAT1 <num, num, num>

(Read-Write) Set and return the labels and grid frame colors in the active window for the PNA display or hardcopy print. (Active labels, Grid frame)

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples `DISP:COL:GRAT1 10,10,10`
`display:color1:grat1 80,80,80`

Query Syntax DISPlay:COLor<n>:GRAT1?

Return Type Numeric (n,n,n)

Default Display = 175,175,175
Print = 0,0,0 (Black)

DISPlay:COLor<n>:GRAT2 <num, num, num>

(Read-Write) 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. (GRID)

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples **DISP:COL:GRAT2 10,10,10**
display:color1:grat2 80,80,80

Query Syntax DISPlay:COLor<n>:GRAT2?

Return Type Numeric (n,n,n)

Default Display = 100,100,100
Print = 50,50,50

DISPlay:COLor<n>:ILABel <num, num, num>

(Read-Write) Set and return the Inactive (not selected) Window Labels for the PNA display or hardcopy print.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples **DISP:COL:ILAB 10,10,10**
display:color1:ilabel 80,80,80

Query Syntax DISPlay:COLor<n>:ILABel?

Return Type Numeric (n,n,n)

Default Display = 160,160,160
Print = 0,0,0 (Black)

DISPlay:COLor<n>:LIM1 <num, num, num>

(Read-Write) Set and return the limit line color of failed traces or failure indicators (dots) and the word Fail.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color.
To find RGB values: from the **Display Colors dialog**, click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:LIM1 10,10,10  
display:color1:lim1 80,80,80
```

Query Syntax DISPlay:COLor<n>:LIM1?

Return Type Numeric (n,n,n)

Default Display = 255,20,20
Print = 255,20,20

DISPlay:COLor<n>:LOAD <value>

(Write-only) Load a color theme from a disc file.

Parameters

<n> Colors to load. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<value> String. Filename of the stored theme. The .colors suffix is automatically appended.

By default, files are stored in C:/Program Files/Keysight/Network Analyzer/Colors/. To store and load files from a different folder, specify the full path and filename.

Examples `DISP:COL:LOAD "myDisplayTheme"`

`display:color2:load "myPrintTheme"`

Query Syntax Not Applicable

Default Not Applicable

DISPlay:COLor<n>:RESet

(Write-only) Resets the current theme to the default PNA colors.

Parameters

<n> Colors to reset. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

Examples

```
DISP:COL:RES
```

```
display:color2:reset
```

Query Syntax Not Applicable

Default Not Applicable

DISPlay:COLor<n>:STORe <value>

(Write-only) Saves the current color theme to a disc file.

Parameters

<n> Colors to store. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<value> String. Filename. The .colors suffix is automatically appended. By default, files are stored in C:/Program Files/Keysight/Network Analyzer/Colors/. To store and load files from a different folder, specify the full path and filename.

Examples `DISP:COL:STOR "myDisplayTheme"`
`display:color2:store "myPrintTheme"`

Query Syntax Not Applicable

Default Not Applicable

DISPlay:COLor<n>:TRACe<nth>:DATA <num, num, num>

(Read-Write) Set and return the color of Data and Limit Lines for nth trace in a window.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<nth> Numeric. Relative trace number in the window for which colors are set. This is not necessarily the trace number. [Learn more](#).

Choose from 1 to 8.

If unspecified, <nth> is set to 1 (first trace).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color. To find RGB values: from the [Display Colors dialog](#), click **Change Color**, then **Define Custom Color**.

Examples `DISP:COL:TRAC2:DATA 10,10,10`
`display:color1:trace5:data 80,80,80`

Query Syntax DISPlay:COLor<n>:TRACe<nth>:DATA?

Return Type Numeric (n,n,n)

Default Varies for each trace.

DISPlay:COLor<n>:TRACe<nth>:MARKer <num, num, num>

(Read-Write) Set and return the color of data trace markers for nth trace in a window.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<nth> Numeric. Relative trace number in the window for which colors are set. This is not necessarily the trace number. [Learn more](#).

Choose from 1 to 8.

If unspecified, <nth> is set to 1 (first trace).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color. To find RGB values: from the [Display Colors dialog](#), click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:TRAC2:MARK 10,10,10
```

```
display:color1:trace5:marker 80,80,80
```

Query Syntax DISPlay:COLor<n>:TRACe<nth>:MARKer?

Return Type Numeric (n,n,n)

Default Varies for each trace.

DISPlay:COLor<n>:TRACe<nth>:MEMory <num, num, num>

(Read-Write) Set and return the memory trace color for nth trace in a window.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<nth> Numeric. Relative trace number in the window for which colors are set. This is not necessarily the trace number. [Learn more](#).

Choose from 1 to 8.

If unspecified, <nth> is set to 1 (first trace).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color. To find RGB values: from the [Display Colors dialog](#), click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:TRAC2:MEM 10,10,10
```

```
display:color1:trace5:memory 80,80,80
```

Query Syntax DISPlay:COLor<n>:TRACe<nth>:MEMory?

Return Type Numeric (n,n,n)

Default Varies for each trace.

DISPlay:COLor<n>:TRACe<nth>:MMARker <num, num, num>

(Read-Write) Set and return the color of memory trace markers for nth trace in a window.

Parameters

<n> Colors to modify. Choose from:

1 - Display colors

2 - Print colors

If unspecified, <n> is set to 1 (Display colors).

<nth> Numeric. Relative trace number in the window for which colors are set. This is not necessarily the trace number. [Learn more](#).

Choose from 1 to 8.

If unspecified, <nth> is set to 1 (first trace).

<num, num, num> Numeric. Red, Green, and Blue (RGB values) that specify a color. To find RGB values: from the [Display Colors dialog](#), click **Change Color**, then **Define Custom Color**.

Examples

```
DISP:COL:TRAC2:MMAR 10,10,10
```

```
display:color1:trace5:mmarker 80,80,80
```

Query Syntax DISPlay:COLor<n>:TRACe<nth>:MMARker?

Return Type Numeric (n,n,n)

Default Varies for each trace.

Last Modified:

23-Nov-2011 Fixed broken links

6-Aug-2009 MX New topic

SENSe:CORR:CKIT:ECAL:CHARacterize Commands

Controls the settings used to perform an ECal User Characterization. These commands do NOT perform the calibration that is required before measuring the ECal module. An S-Parameter channel must already be calibrated. [Learn more.](#)

SENSe:CORR:CKIT:ECAL:CHARacterize:

[ACQuire](#)

[CNUMber](#)

[CONNector](#)

| [CATalog?](#)

| [PORT<n>\[:SElect\]](#)

[DESCription](#)

| [PORT<n>\[:SElect\]](#)

| [\[STEP\]?](#)

| [USER](#)

| [VNA](#)

[DMEMory](#)

| [SAVE](#)

[ID](#)

[INITiate](#)

[INSitu](#)

| [ENABle](#)

| [\[STATe\]](#)

[SAVE](#)

[STEPs?](#)

Click on a keyword to view the command details.

Notes:

These commands provide for the following:

- Measure the ECal module with adapters, cables, or fixtures to be included in the User Characterization.
- Allow descriptive text to be entered.

- Save the User Characterization to the ECal module or PNA disk memory. Up to 12 User Characterizations can be stored in an ECal module. [Learn more.](#)

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.

see Also

- [Example - Perform an ECal User Characterization](#)
 - [Learn about ECal User Characterization](#)
 - [Synchronizing the Analyzer and Controller](#)
 - [SCPI Command Tree](#)
-

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:ACQuire STAN<step>

(Write-only) Initiates the measurement of the ECal module. The user characterization process must have been initiated first using **SENS:CORR:CKIT:ECAL:CHAR:INIT**. Currently, only ONE step is required to measure the ECal module.

Note: This command is an overlapped command. When ***OPC** is issued with it, the OPC bit in the PNAs Standard Event Status Register is not set until this command has completed its operation. [Learn more](#).

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1.
Channel number being calibrated.

<step> Integer User characterization step number to be measured.

Examples **SENS:CORR:CKIT:ECAL:CHAR:ACQ STAN1, *OPC**

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:CNUMber<n>

(Read-Write) Sets and reads the number to which the User Characterization will be stored in the ECal module. The number must be set before sending **SENS:CORR:CKIT:ECAL:CHAR:INIT** or the default value (1) will be used.

This command is NOT necessary when saving the User Characterization to the PNA disk memory.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

<n> User Characterization number. Choose a value between 1 and 12.

Examples **SENSe:CORR:CKIT:ECAL:CHAR:CNUM 2**

Query Syntax SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:CNUMber?

Return Type Numeric

Default 1

SENSe:CORRection:CKIT:ECAL:CHARacterize:CONNector:C/

(Read-only) Returns a list of connector names that are valid for use with user-characterized ECal modules. Use an item from the returned list to specify a connector for **SENS:CORR:CKIT:ECAL:CHAR:CONN:PORT**.

Use only factory-defined connector types when you store a user characterization to PNA disk memory.

Parameters None

Examples

SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?

'Example returned string: "APC 3.5 male, APC 3.5 female, Type N (50) female, Type N (50) male, APC 7, Type A (50), Type B"

Return Type Comma-separated string

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:CONNector[:SElect] <string>

(Read-Write) Specifies a connector type name for every ECal module port used during the user characterization. Valid connector names are returned using `SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?`

This command refers to the ECal ports by number instead of letter (1 = Port A, 2 = Port B and so forth). The connector names should be set for the ports before sending `SENS:CORR:CKIT:ECAL:CHAR:INIT`.

The following steps could be followed to ensure port connectors are specified correctly:

1. Use `SENS:CORR:CKIT:ECAL:CHAR:CONN:CAT?` to query available connectors before specifying the port connector.
2. Specify a connector type using this command. If the <string> parameter was incorrectly entered, an error will be returned.

Note: Use only factory-defined connector types when you store a user characterization to PNA disk memory.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

<n> ECal test port number for which a connector type will be specified. Choose 2 for a 2-port ECal module, 1 through 4 for a 4-port module.

<string> ECal connector type and gender (if applicable).

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.

Examples `SENS:CORR:CKIT:ECAL:CHAR:CONN:PORT2 "APC 3.5 female"`

Query Syntax `SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:CONNector:PORT[:SElect]?`

Return Type String

Default "No adapter"

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCript <stepN>

(Read-only) Returns the connection description for the specified step of the ECal user characterization process. The user characterization process must have been initiated first using **SENS:CORR:CKIT:ECAL:CHAR:INIT**.

Parameters

- <ch> Channel number being calibrated. If unspecified, value is set to 1
- <stepN> Integer - User characterization step number for which a description will be returned. Use **SENS:CORR:CKIT:ECAL:CHAR:STEP?** to query the number of steps.

Examples

```
SENSe:CORR:CKIT:ECAL:CHAR:DESC? 2
```

"Example return string:

```
Connect ECal Module Ports A and B to PNA Ports 1 and 2
```

Return Type String

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCript <string>

(Read-Write) Sets and reads the description of the person and/or company who is producing the ECal user characterization. This description is stored with the characterization.

Set this description before sending **SENS:CORR:CKIT:ECAL:CHAR:INIT** or the default (empty string) will be used.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

<string> Descriptive text, limited to 19 characters maximum.

Examples **SENSe:CORR:CKIT:ECAL:CHAR:DESC:USER "John Doe, Acme Inc."**

Query Syntax SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCription:USEI

Return Type String

Default " " (Empty String)

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCript <string>

(Read-Write) Sets and reads a description of the Vector Network Analyzer used to perform the User Characterization. This description is stored with the user characterization.

Set this description before sending **SENS:CORR:CKIT:ECAL:CHAR:INIT** or the default (empty string) will be used.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

<string> Descriptive text, limited to 14 characters maximum.

Examples **SENSe:CORR:CKIT:ECAL:CHAR:DESC:VNA "My PNA"**

Query Syntax SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCription:VNA

Return Type String

Default " " (Empty String)

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCript[:SElect] <string>

(Read-Write) 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 command refers to the ECal ports by number instead of letter (1 = Port A, 2 = Port B, and so forth). This description is stored with the user characterization.

Set this description before sending **SENS:CORR:CKIT:ECAL:CHAR:INIT** or the default (empty string) will be used.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

<n> ECal port number. Choose 1 to 2 for a 2-port ECal module, 1 to 4 for a 4-port module.

<string> Descriptive text, limited to 24 characters maximum.

Examples **SENSe:CORR:CKIT:ECAL:CHAR:DESC:PORT1 "3.5 mm adapter, 00001"**

Query Syntax SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DESCription:POR[:SElect]?

Return Type String

Default " " (Empty String)

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:DMEMor<charName>

(Write-only) Completes an ECal user characterization by writing the characterization data to the PNA disk. To write the characterization data to the ECal module, use **SENS:CORR:CKIT:ECAL:CHAR:SAVE**. A User Characterization can be saved to both PNA disk memory and ECal module memory.

Use this <charName> for performing future calibrations with this User Characterization. See **SENS:CORR:CKIT:ECAL:KNAM:INF?**

Note: An ECal confidence check can NOT be performed remotely from User Characterizations that are stored on the PNA disk.

Parameters

- <ch> Channel number being calibrated. If unspecified, value is set to 1.
- <charName> 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.

Examples

```
SENS:CORR:CKIT:ECAL:CHAR:DMEM:SAVE "DUT1 User Char"
```

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:ID <model, sn>

(Read-Write) 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.

Parameters

- <ch> Channel number being calibrated. If unspecified, value is set to 1
- <model, sn> Model and serial number of the ECal module to be characterized.

Examples `SENSe:CORR:CKIT:ECAL:CHAR:ID "N4433A,00001"`

Query Syntax `SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:ID?`

Return Type String

Default " " (Empty String)

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:INITiate [bool]

(Write-only) Initiates an ECal User Characterization. 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.

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.

Parameters

<ch> Channel number of a calibrated S-parameter channel. If unspecified, value is set to 1

[bool] Optional argument. If unspecified, value is set to 1.

ON (or 1) Check ECal memory to ensure that a new characterization with the channels current number of points will fit in the module memory. Select for User Characterizations to be stored in internal ECal memory.

OFF (or 0) Skip the check. Select for User Characterizations to be stored to PNA disk memory.

Examples

```
SENS:CORR:CKIT:ECAL:CHAR:INIT  
sense2:correction:ckit:ecal:characterize:initiate off
```

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:INSitu:EN

(Read-only) Returns whether the device that was specified by **SENS:CORR:CKIT:ECAL:CHAR:ID** is a CalPod module, which is capable of being characterized as an in-situ device. [Learn more.](#)

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1.

Examples **SENS:CORR:CKIT:ECAL:CHAR:INSitu:ENABLE?**

Return Type Boolean

1 Device is a CalPod module

0 Device is NOT a CalPod module

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:INSitu[:ST] <bool>

(Read-Write) Sets or returns whether the device (CalPod module) that was specified by **SENS:CORR:CKIT:ECAL:CHAR:ID** will be characterized as an in situ device. Learn more.

Parameters

- <ch> Channel number being calibrated. If unspecified, value is set to 1.
- <bool> In situ state. Choose from:
ON (or 1) - Characterize the CalPod module as an in situ device.
OFF (or 0) Do NOT characterize the CalPod module as an in situ device.

Examples **SENS:CORR:CKIT:ECAL:CHAR:INSitu 1**

Query Syntax SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:INSitu[:STAtE]?

Return Type Boolean

Default ON or 1

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:SAVE

(Write-only) Completes an ECal user characterization by writing the characterization data to the ECal module memory. To write the characterization data to PNA disk memory, use **SENS:CORR:CKIT:ECAL:CHAR:DMEM:SAVE**. A User Characterization can be saved to both PNA disk memory and ECal module memory.

Note: This command is an overlapped command. When ***OPC** is issued with it, the OPC bit in the PNAs Standard Event Status Register is not set until this command has completed its operation. [Learn more](#).

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1.

Examples **SENS:CORR:CKIT:ECAL:CHAR:SAVE, *OPC**

Query Syntax Not Applicable

Default Not Applicable

SENSe<ch>:CORRection:CKIT:ECAL:CHARacterize:STEPs?

(Read-only) Returns the number of steps required to measure the ECal module. Currently, only ONE is required.

Parameters

<ch> Channel number being calibrated. If unspecified, value is set to 1

Examples SENS:CORR:CKIT:ECAL:CHAR:STEP?

Return Type Numeric

Default Not Applicable

Last Modified:

12-Feb-2013	Added Insitu commands
24-Aug-2009	Added disk storage capability (9.0)
4-Nov-2008	New topic (8.33)

Perform a Guided Cal with Sliding Load

This example sets the sliding load behavior, then performs a Guided Calibration that uses a sliding load.

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 guided.vbs.

[Learn how to setup and run the macro.](#)

See [Guided Cal commands](#).

```
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
' Specify that any sliding loads should be measured using the
' remote iterative method rather than launching sliding load dialog.
scpi.Execute "sens:corr:coll:guid:pref:slid iter"
scpi.Execute "sens:corr:coll:guid:conn:port1 ""APC 3.5 female"" "
scpi.Execute "sens:corr:coll:guid:conn:port2 ""APC 3.5 male"" "
' 85052B cal kit uses sliding loads
scpi.Execute "sens:corr:coll:guid:ckit:port1 ""85052B"" "
scpi.Execute "sens:corr:coll:guid:ckit:port2 ""85052B"" "
scpi.Execute "sens:corr:coll:guid:init"
numSteps = scpi.Execute("sens:corr:coll:guid:steps?")
' Measure the standards
For i = 1 To numSteps
    step = "Step " + CStr(i) + " of " + CStr(numSteps)
    strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
    MsgBox strPrompt, vbOKOnly, step
    minIterations = scpi.Execute("sens:corr:coll:guid:iter:min? " + CStr(i))
```

```
For j = 1 To minIterations
  If minIterations > 1 Then MsgBox "Adjust/position the standard for
measurement " + CStr(j) + " of " + CStr(minIterations), vbOKOnly
  scpi.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next
iterationCount = scpi.Execute("sens:corr:coll:guid:iter:coun? " + CStr(i))
If iterationCount <> minIterations Then
  MsgBox "Unexpected error!", vbOKOnly, step
  scpi.Execute "sens:corr:coll:guid:iter:res " + CStr(i)
End If
Next
' Conclude the calibration
scpi.Execute "sens:corr:coll:guid:save"
```

Last modified:

4-Dec-2012 New topic

Status Command Keywords

The following keywords can be appended to the node or nodes that represent the Status register you want to control.

▪ **CONDition?**

▪ **ENABLE**

▪ **ENABLE?**

▪ **EVENT?**

▪ **MAP**

▪ **NTRansition**

▪ **PTRansition**

[Learn about Status Registers](#)

[SCPI Command Tree](#)

:CONDition?

Monitors the conditions as they occur REAL TIME. That is, a condition may occur, and then clear before the condition is read. Reading this register returns a 16-bit decimal weighted number.

:ENABLE <bit>

Enables register bits that will be monitored using the **service request (SRQ)** method. (To use the direct read method, you do not have to enable the bit.)

Default value for STATUS:QUESTIONable:ENABLE and STATUS:OPERation:ENABLE is 0: No bits enabled.

Default value for all other registers :ENABLE <bits> is 32767; ALL BITS ENABLED.

Therefore it is **ONLY** necessary to send the ENABLE keyword if you want to **DISABLE** some conditions. For example, to enable **ONLY** Trace1 (bit 2) of the LIMIT1 register (disable all other traces) , send:

STATUS:QUESTIONable:LIMit1:ENABLE 4

:ENABLE?

Read the enable register to verify the bits that you enabled. Returns a 16 bit weighted sum of the bits that are enabled.

[:EVENT]?

Query only - This is the Default keyword for most registers. Use it to determine if a condition has occurred. These bits remain set until they are read or otherwise cleared.

:MAP <bit>,<error>

Associates a bit in the User register with an error number. For example

STATUS:QUESTIONABLE:DEFINE:USER2:MAP 0,-113

0 is the bit that will be set

-113 is the error

When error -113 "Undefined Header" occurs, bit 0 in the USER2 register will be set to 1.

:NTRansition <bits>

Write-Read - Negative Transition register bits set the condition to be set on the Negative going (True to False) transition. Use this register if you are only interested in a condition changing from True to False.

:NTRansition?

queries the register to verify that you set a negative transition.

:PTRansition <bits>

Write-Read - Positive Transition register bits set the condition to be set on the False to True transition. Use this register if you are only interested in the change of a condition from False to True.

:PTRansition?

Queries the register to verify that you set a positive transition.

Perform a Cal All Channels Calibration

This example sets up an SMC channel and a standard channel. It then performs a 'Cal All Channels' calibration which calibrates both channels.

Note: The example does NOT modify any Cal All path 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 *.vbs.

[Learn how to setup and run the macro.](#)

[See CalAll SCPI commands](#)

[Learn about Cal All](#)

See Other SCPI Example Programs

```
Dim app
Dim scpi
' Create / Get the PNA application.
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
'Preset the analyzer
scpi.Parse "SYST:FPRreset"
'
'***** Setup Channel 1, Window 1 *****
' Create and turn on window 1
scpi.Parse "DISPlay:WINDow1:STATE ON"
'Define a measurement name in chan 1 with S21 parameter
scpi.Parse "CALC1:PARAmeter:DEFine:EXT 'MyMeas',S21"
'FEED the measurement to window 1, and give the new TRACe a number (1).
scpi.Parse "DISPlay:WINDow1:TRACe1:FEED 'MyMeas'"
```

'Modify stimulus settings

'Set IF Bandwidth to 700 Hz

scpi.Parse "SENSe1:BANDwidth 700"

'Set Center and Span Freq's to 1 GHz

scpi.Parse "SENSe1:FREQuency:CENTer 1ghz"

scpi.Parse "SENSe1:FREQuency:SPAN 1ghz"

'Set number of points to 11

scpi.Parse "SENSe1:SWEep:POINts 11"

'

'***** Setup Channel 2, Window 2 *****

' Create and turn on window 2

scpi.Parse "DISPlay:WINDow2:STATE ON"

' Create an SMC meas in channel 2

scpi.Parse "CALC2:CUST:DEF 'My SC21', 'Scalar Mixer/Converter', 'SC21'"

'Setup the new measurement in the active window

scpi.Parse "DISP:WIND2:TRAC:FEED 'My SC21'"

' Setup Mixer Stimulus *****

' Points and IFBW are channel settings

scpi.Parse "SENS2:SWEep:POINts 11"

scpi.Parse "SENS2:BANDwidth 1e3"

' The rest are mixer settings

scpi.Parse "SENS2:MIX:INPut:FREQ:MODE SWEPT"

scpi.Parse "SENS2:MIX:INPut:FREQ:STAR 3.6e9"

scpi.Parse "SENS2:MIX:INPut:FREQ:STOP 3.9e9"

scpi.Parse "SENS2:MIX:LO:FREQ:MODE FIXED"

scpi.Parse "SENS2:MIX:LO:FREQ:FIX 1.0e9"

scpi.Parse "SENS2:MIX:LO:POW 10"

scpi.Parse "SENS2:MIX:OUTP:FREQ:SID LOW"

scpi.Parse "SENS2:MIX:CALC Output"

scpi.Parse "SENS2:MIX:LO:NAME 'Port 3'"

scpi.Parse "SENS2:MIX:APPLY"

'

'***** Setup Cal All Channel *****

'Reset Cal All settings

```

scpi.Parse "SYST:CAL:ALL:RESet"
' Select channels to cal
' By default, all active channels are selected.
' VMC channels are NOT supported by Cal All.
scpi.Parse "SYST:CAL:ALL:SEL 1,2"
'Set IFBW
scpi.Parse "SYST:CAL:ALL:IFBW 1e3"
'Set power level
scpi.Parse "SYST:CAL:ALL:PORT2:SOUR:POWer 0"
'Set CalSet prefix. The meas class and channel number
'are appended to the User Cal set for each channel.
'If you don't set this, only Cal Registers will be generated.
scpi.Parse "SYST:CAL:ALL:CSET:PREFix 'MyCalAll'"
' Read unique Cal settings for Cal All channels
uniqSettings = scpi.Parse ("SYST:CAL:ALL:MCL:PROP:NAME:CAT?")
msgbox (uniqSettings)
' We want "Enable Phase Correction"
' Now find valid settings
uniqVal = scpi.Parse ("SYST:CAL:ALL:MCL:PROP:VAL:CAT? 'Enable Phase
Correction'")
'returns 'false,true' make it true.
scpi.Parse "SYST:CAL:ALL:MCL:PROP:VAL 'Enable Phase
Correction','true'"
'
'***** 'Perform Guided Cal on <ch> *****
'Read the Cal All channel number
ch = scpi.Parse ("SYST:CAL:ALL:GUIDed:CHAN?")
'Convert ch to integer
ch=CInt(ch)
' Specify the DUT connectors
scpi.Parse ("sens" &ch& ":corr:coll:guid:conn:port1 ""APC 3.5 female"" ")
scpi.Parse ("sens" &ch& ":corr:coll:guid:conn:port2 ""APC 3.5 male"" ")
scpi.Parse ("sens" &ch& ":corr:coll:guid:conn:port3 ""Not used"" ")
scpi.Parse ("sens" &ch& ":corr:coll:guid:conn:port4 ""Not used"" ")

```

```

' Select the Cal Kit for each port being calibrated.
scpi.Parse ("sens" &ch& ":corr:coll:guid:ckit:port1 ""85052D"" ")
scpi.Parse ("sens" &ch& ":corr:coll:guid:ckit:port2 ""85052D"" ")
' Initiate the calibration and query the number of steps
scpi.Parse ("sens" &ch& ":corr:coll:guid:init")
numSteps = scpi.Parse("sens" &ch& ":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)
strPrompt = scpi.Parse("sens" &ch& ":corr:coll:guid:desc? " + CStr(i))
MsgBox strPrompt, vbOKOnly, step
scpi.Parse ("sens" &ch& ":corr:coll:guid:acq STAN" + CStr(i)) + ";*OPC?"
' 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 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 occur automatically
' "SENSE:CORREction:COLLect:ACQUIRE
ECAL1,CHAR0,ASYNchronous;*OPC"
Next
' Conclude the calibration

```

```
scpi.Parse ("sens" &ch& ":corr:coll:guid:save")  
' Read the cal set names that were generated  
calsets = scpi.Parse ("system:cal:all:cset:catalog?")  
MsgBox calsets
```

Last modified:

7-Sep-2012 New topic

Product Support Overview

Learn about your Analyzer

- [Specifications](#)
- [M937x Configurations](#)
- [Analyzer Accessories](#)

Update your Analyzer

- [Firmware Update](#) (Agile Update)
- [Option Enable](#)
- [Instrument Calibration](#)

Problems with your Analyzer

- [Diagnostic Tools, Utilities, and Adjustments](#)
- [Troubleshoot the Analyzer](#)
- [About Error Messages](#)

Resources for your Analyzer

- [Technical Support](#)
- [Other Resources](#)

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

System Settings and Configuration

System Settings

- [Dialog Transparency](#)
- [Display Colors](#)
- [Frequency Blanking](#) (For security purposes)
- [Preferences](#)
- [Power Limit and Power Offset](#) (Prevents overpowering DUTs)
- [Preset the Analyzer](#)
- [System Impedance](#)
- [Receiver Temperature](#)
- [About Error Messages](#)

Caution: Avoid expensive repairs to your analyzer. Read [Electrostatic Discharge Protection](#).

Instrument Calibration Verification

When the annual instrument calibration has been performed on your M937xA module, you will notice that there is no room on the front panel for the traditional calibration 'sticker'. Instead, the calibration date and (next) calibration due date are appended to the M937xA Help About page.

The Calibration date is edited when you send your M937xA module to Keysight for Instrument Calibration.

You can edit these Help About fields, as well as set the calibration interval, using the **SYST:SERV:VER** SCPI commands.

Tutorials

- [Videos](#)
- [Application Notes](#)
- [Connector Care](#)
- [Electrostatic Discharge \(ESD\) Protection](#)
- [Network Analyzer Basics \(video\)](#)

Measurement Tutorials

- [Absolute Output Power](#)
- [AM-PM Conversion](#)
- [Amplifier Parameters](#)
- [Antenna Measurements](#)
- [Balanced Measurements](#)
- [Complex Impedance](#)
- [Comparing the VNA Delay Functions](#)
- [Deviation from Linear Phase](#)
- [Gain and Flatness](#)
- [Gain Compression](#)
- [Group Delay](#)
- [High-Gain Amplifier Measurements](#)
- [Phase Measurements](#)
- [Reverse Isolation](#)
- [Reflection Measurements](#)
- [Time Domain Measurements](#)

Caution: Avoid expensive repairs to your PNA. Read [Electrostatic Discharge Protection](#).

N937xA Front Panel Tour



See Also

- [Soft Front Panel Tour](#)

Status LED

The following LED colors indicate the M937xA status:

Green (solid) – Firmware is running but the M937xA is NOT sweeping.

Green (blinking) - Each blink indicates a measurement sweep. The M937xA may be sweeping faster than the indicator can blink.

Amber – Power is ON but the firmware is not running.

Red – A hardware error has occurred.

Off – Power is OFF.

Test Ports

All M937xA models are available with 2 test ports.

See [Specs](#) for more information about the Test port connectors and Input damage levels.

Ref In / OUT

IN 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.

OUT This SMB(m) connector outputs a 10 MHz frequency reference signal for use by other modules and test equipment.

Note: To use an external reference signal, you must make a setting in the M937xA.

- From the soft front panel, click **Utility**, then **System**, then **Configure**, then check **External Reference**.
- From SCPI, use [SENS:ROSC:SOUR](#)

Trigger Lines

Trig IN When enabled, the module accepts signals on this connector which indicates that the external devices is ready to be triggered.

Trig OUT When enabled, the module outputs signals on these connectors either before or after a measurement.

Trig Ready When enabled, the module outputs a 'READY' signal on this connector to other devices. This indicates that the module is ready to be triggered.

[Learn more about External Triggering](#)

LO Out / LO In

For Daisy-chaining two or more modules. See the M937xA Installation Guide for more information.



- **PXI has NO applications**

Perform a Guided QSOLT Cal

This example performs a Guided QSOLT calibration on a 4-port PNA.

Because the DUT port 1 is female and the other ports are male, a 'Zero Thru' can be used between port 1 and the other ports. If this were NOT the case, a "Defined Thru" would be needed in the listed Cal Kits for those ports. [Learn more about Thru methods.](#)

Although no standards are used for ports 2, 3, and 4, a Cal Kit must be defined for these ports.

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 qsolt.vbs.

See Also

[Learn more about QSOLT.](#)

[Learn how to setup and run the macro.](#)

See [Guided Cal commands.](#)

```
Set app = CreateObject("AgilentPNA835x.Application")
Set scpi = app.ScpiStringParser
scpi.Execute "Calc1:Par:Mnum 1"
scpi.Execute "Sens1:CORR:COLL:GUID:CONN:PORT1:SEL 'APC 3.5
female'"
scpi.Execute "Sens1:CORR:COLL:GUID:CONN:PORT2:SEL 'APC 3.5 male'"
scpi.Execute "Sens1:CORR:COLL:GUID:CONN:PORT3:SEL 'APC 3.5 male'"
scpi.Execute "Sens1:CORR:COLL:GUID:CONN:PORT4:SEL 'APC 3.5 male'"
scpi.Execute "Sens1:CORR:COLL:GUID:CKIT:PORT1:SEL '85052B'"
scpi.Execute "Sens1:CORR:COLL:GUID:CKIT:PORT2:SEL '85052B'"
scpi.Execute "Sens1:CORR:COLL:GUID:CKIT:PORT3:SEL '85052B'"
scpi.Execute "Sens1:CORR:COLL:GUID:CKIT:PORT4:SEL '85052B'"
scpi.Execute "Sens1:CORR:COLL:GUID:THRU:PORT 1,2,1,3,1,4"
```

```

scpi.Execute "Sens1:CORR:COLL:GUID:PATH:TMET 1,2,'Zero Thru'"
scpi.Execute "Sens1:CORR:COLL:GUID:PATH:TMET 1,3,'Zero Thru'"
scpi.Execute "Sens1:CORR:COLL:GUID:PATH:TMET 1,4,'Zero Thru'"
scpi.Execute "Sens1:CORR:COLL:GUID:PATH:CMET 1,2,'QSOLT1'"
scpi.Execute "SENS1:CORR:COLL:GUID:PATH:CMET 1,3,'QSOLT1'"
scpi.Execute "SENS1:CORR:COLL:GUID:PATH:CMET 1,4,'QSOLT1'"
' Initiate the calibration and query the number of steps
scpi.Execute "sens1:corr:coll:guid:init"
numSteps = scpi.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)
strPrompt = scpi.Execute("sens:corr:coll:guid:desc? " + CStr(i))
MsgBox strPrompt, vbOKOnly, step
scpi.Execute "sens:corr:coll:guid:acq STAN" + CStr(i)
Next
' Conclude the calibration
scpi.Execute "SENS1:CORR:COLL:GUID:SAVE"

```

Last modified:

18-Dec-2013 New topic

Perform Unguided THRU Response Cal

This example program performs Thru Response cals in both the forward and reverse directions. It does this by selecting the appropriate measurement right before acquiring the standard. The cal infers the direction from the measurement.

This program also demonstrates the use of the **SENSe:CORR:PREF:CSET:SAVE** command. The details are in the comments.

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.](#)

```
Dim App
```

```
Set App = CreateObject("AgilentPNA835x.Application")
```

```
Dim Parser
```

```
Set Parser = App.SCPIStringParser
```

```
'Preset and delete measurement
```

```
Parser.Parse "SYSTEM:FPRreset"
```

```
'The following commands determine how the cal set is saved.
```

```
'Pick one of the following preferences, comment the other
```

```
'Save cals to separate new USER CalSets
```

```
'Parser.Parse "SENS:CORR:PREF:CSET:SAVE USER"
```

```
'Save both cals to a single cal register
```

```
'Parser.Parse "SENS:CORR:PREF:CSET:SAVE CALR"
```

```
'Save both cals to a single currently selected CalSet or register
```

```
Parser.Parse "SENS:CORR:PREF:CSET:SAVE REUSE"
```

```
'
```

```
'Create a new S21 Measurement
```

```
Parser.Parse "CALCulate:PARAMeter:DEFine:EXT 'MyS21Meas',S21"
```

```
Parser.Parse "DISPlay:WINDow1 ON"
```

Parser.Parse "DISPlay:WINDow1:TRACe1:FEED 'MyS21Meas'"

'Create a new S12 Measurement

Parser.Parse "CALCulate:PARAmeter:DEFine:EXT 'MyS12Meas',S12"

Parser.Parse "DISPlay:WINDow1:TRACe2:FEED 'MyS12Meas'"

'Turn off continuous sweep

Parser.Parse "INITiate:CONTInuous OFF"

'Begin cals

'Select a cal kit

Parser.Parse "SENSe:CORRection:COLLect:CKIT:SElect 1"

'Perform a forward thru response cal

'Select the S21 Meas

Parser.Parse "CALCulate1:PARAmeter:SElect 'MyS21Meas'"

'Set the calibration method to Thru Response

Parser.Parse "SENSe1:CORRection:COLLect:METhod TRAN1"

MsgBox("Connect Thru between ports Then press OK")

Parser.Parse "SENSe1:CORRection:COLLect:ACQuire STAN4"

Parser.Parse "SENSe1:CORRection:COLLect:SAVE"

'Then perform a reverse thru response cal

'Change measurement to S12

Parser.Parse "CALCulate1:PARAmeter:SElect 'MyS12Meas'"

'Set the calibration method to Thru Response

Parser.Parse "SENSe1:CORRection:COLLect:METhod TRAN1"

'Ensure the thru connection is still in place

'Acquire Thru std in reverse direction

Parser.Parse "SENSe1:CORRection:COLLect:ACQuire STAN4"

'All standards have been measured.

Parser.Parse "SENSe1:CORRection:COLLect:SAVE"

'Turn ON continuous sweep

Parser.Parse "INITiate:CONTInuous ON"

MsgBox("The calibration has been completed")

Getting and Putting Data

This Rocky Mountain Basic example does the following:

1. Takes a sweep, and reads the formatted data trace into an array. The trace is read as a definite length block.
2. Instructs you to remove DUT
3. Downloads the trace back to the analyzer as an definite length block.

See Other SCPI Example Programs

```
100 DIM A$(10),Data1(1:51)
110 INTEGER Digits,Bytes
120 !
130 COM /Sys_state/ @Hp87xx,Score
140 ! Identify I/O Port
150 CALL Iden_port
160 !
170 !
180 OUTPUT @Hp87xx;"SYST:PRES"
190 !
200 OUTPUT @Hp87xx;"CALC:PAR:SEL 'CH1_S11_1'"
210 !
220 ! Set up the analyzer to measure 51 data points.
230 OUTPUT @Hp87xx;"SENS1:SWE:POIN 51;*OPC?"
240 ENTER @Hp87xx;Opc
250 !
260 ! Take a single sweep, leaving the analyzer
270 ! in trigger hold mode.
280 OUTPUT @Hp87xx;"ABOR;:INIT1:CONT OFF;:INIT1;*WAI"
290 !
300 ! Select binary block transfer
310 OUTPUT @Hp87xx;"FORM:DATA REAL,64"
320 !
330 ! Request the channel 1 formatted data array
```

```
340 ! from the analyzer.
350 OUTPUT @Hp87xx;"CALC:DATA? FDATA "
360 !
370 ! Turn on ASCII formatting on the I/O path.
380 ! It is needed for reading the header
390 ! information.
400 ASSIGN @Hp87xx;FORMAT ON
410 !
420 ! Get the data header. "A$" will contain the
430 ! "#" character indicating a block data transfer.
440 ! "Digits" will contain the number of characters
450 ! for the number of bytes value which follows.
460 ENTER @Hp87xx USING "%,A,D";A$,Digits
470 !
480 ! Get the rest of the header. The number of
490 ! bytes to capture in the data array will be
500 ! placed in "Bytes". Note the use of "Digits"
510 ! in the IMAGE string.
515 !
520 ENTER @Hp87xx USING "%,"&VAL$(Digits)&"D";Bytes
525 PRINT "HEADER",A$,Digits,Bytes
530 !
540 ! Turn off ASCII formatting on the I/O path;
550 ! it is not needed for transferring binary
560 ! formatted data.
570 ASSIGN @Hp87xx;FORMAT OFF
580 !
590 ! Get the data.
600 ENTER @Hp87xx;Data1(*)
610 !
620 ! Turn on ASCII formatting again.
630 ASSIGN @Hp87xx;FORMAT ON
640 !
650 ! Get the "end of data" character.
660 ENTER @Hp87xx;A$
670 !
680 ! Display the first three numbers in the array.
```

```
690 DISP "Trace: ";Data1(1);Data1(2);Data1(3);"..."
700 !
710 ! Use this time to visually compare the
720 ! numbers to the visible data trace.
730 WAIT 5
740 !
750 ! Prompt the operator to disconnect the test
760 ! device and how to continue the program.
770 DISP "Disconnect the test device -- Press Continue"
780 PAUSE
790 !
800 ! Update the display line.
810 DISP "Taking a new sweep...";
820 !
830 ! Take a sweep so the display shows new data.
840 OUTPUT @Hp87xx;":INIT1;*WAI"
850 DISP " Done."
860 WAIT 5
870 !
880 ! Send the header for an indefinite block length
890 ! data transfer.
900 DISP "Downloading saved trace...";
915 ! The first byte '3' indicates the next three digits equal number of
transfer bytes
916 ! The number of transfer bytes equals 8x the number of tracepoints.
920 OUTPUT @Hp87xx;"CALC:DATA FDATA, #3408";
930 !
940 ! Turn off ASCII formatting.
950 ASSIGN @Hp87xx;FORMAT OFF
960 !
970 ! Send the data array back to the analyzer.
980 OUTPUT @Hp87xx;Data1(*),END
990 !
1000 ! Turn on ASCII formatting again.
1010 ASSIGN @Hp87xx;FORMAT ON
1020 DISP " Done!"
1030 END
```

```
1040 !
1050
!*****
1060 ! Iden_port: Identify io port to use
1070 ! Description: This routines sets up the I/O port address for
1080 !     the SCPI interface. For "HP 87xx" instruments,
1090 !     the address assigned to @Hp87xx = 800 otherwise,
1100 !     716.
1110
!*****
1120 SUB Iden_port
1130   COM /Sys_state/ @Hp87xx,Scode
1140 !
1150   IF POS(SYSTEM$("SYSTEM ID"),"HP 87")<>0 THEN
1160     ASSIGN @Hp87xx TO 800
1170     Scode=8
1180   ELSE
1190     ASSIGN @Hp87xx TO 716
1200     Scode=7
1210   END IF
1220 !
1230 SUBEND !Iden_port
1240 !
```

Watch and subscribe to the latest **Keysight Network Analyzer videos** at:
<http://www.youtube.com/user/AgilentNetworkAnalyz?feature=watch>
(Internet connection required)