Measurement

Measurement

This chapter provides information related to the measurements done through E5071C.

- Setting Measurement Conditions
- Calibration
- Making Measurements
- Data Analysis
- Data Output
- Optimizing Measurements
- Measurement Examples
- Fixture Simulator

Setting Measurement Conditions

Setting Measurement Conditions

- Initializing Parameters
- Setting Channels and Traces
- Setting the System Z0
- Setting Stimulus Conditions
- Selecting Measurement Parameters
- Selecting a Data Format
- Setting the Scales
- Setting Window Displays

Initializing Parameters

The E5071C has three different initial settings as shown below.

Initial setting	Restore method
Preset state	Press **Preset** > **OK** the front panel or Execute the :SYST:PRES command
*RST state	Execute the *RST command
Factory default setting	E5071C factory (default) settings

The user can set items to be preset freely. For more information, see Setting the user preset function.

Other topics about Setting Measurement Conditions

Setting Channels and Traces

Other topics about Setting Measurement Conditions

Overview

The E5071C allows you to use up to 160 channels (when the number of traces is up to 9) to perform measurement under 160 different stimulus conditions.

For each channel, up to 4, 6, 9, 12 or 16 traces (measurement parameters) can be displayed. Because multiple traces can be displayed for each channel, no feature is provided to link the stimulus conditions between channels, and each channel is always independent of the others. In other words, for the E5071C, you need to set the measurement conditions and execute calibration for each channel you use for measurement.

With the E5071C, you can change the number of available channels and the upper limit of the number of traces. If you change the upper limit setting, you need to restart the firmware of the E5071C. Therefore, first, set the upper limit appropriately depending on the numbers of channels and traces necessary for your measurement.

When you set items whose setting target is channels/traces (refer to Parameter setting for each setup item), the target is the selected (active) channel/trace. You can specify only the displayed channels/traces as active channels/traces. Therefore, set the display of channels/traces before setting the measurement conditions.

Setting Upper Limit of Number of Channels/Traces

You can select the upper limits of the number of channels and the number of traces from the following combinations.

Channels	Traces	Maximum Number of Points	Supported Revision (Firmware Revision)	Note

1	4	1601	All	You may want to select the "1-channel, 4-trace" or "2-channel, 4-trace" configuration to save the time required to save/recall the instrument state file, since this takes longer with other configurations. More channels and traces requires longer measurement time.
4	16	1601		
9	9	1601		
12	6	1601		
16	4	1601		
16	16	1601		
24	12	1601		
36	9	1601		
2	8	3201	From A.09.50	
2	8	1601	Up to A.09.4x	
2	8	10001	From A.09.20 to A.09.4x	
2	16	10001	From A.09.50	
1	4	20001	All	
72	9	401	From A.08.10	For these setup, press **System** > **Service Menu** > **Channel/Trace Setup** In this configuration, when many of channel/trace/points are used, memory swap or memory shortage may occur during the measurement. It causes pauses or hang-up of measurement.
96	9	201		
160	9	201	From A.09.10	
6	16	10001	From A.11.0x	

4	9	20001		

- Its important to realize that the combination used to save a state file must be the same used to recall it; that is, you cannot save a state file with one combination of channels/traces and recall it with another combination. For more details, see Saving and Recalling Instrument State.

The selection procedure is as follows:

1. Press **System** > **Misc Setup** > **Channel/Trace Setup**. (**System** > **Service Menu** > **Channel/Trace Setup** for some settings)
2. Press the desired softkey to select the upper limits of the number of channels and the number of traces.
3. The dialog box that prompts "New Channel/Trace configuration will take effect after firmware restart." appears.
4. Click **OK**.
5. Press any softkeys or hardkeys besides **Preset** key.
6. The dialog box that prompts "Do you want to restart now?" appears.
7. Click **Yes**.
8. On choosing a multi-channel mode, it may take more time to restart the firmware. The screen may seem to freeze but will resume their normal operation once the settings are applied.

Setting Channel Display (Layout of Channels)

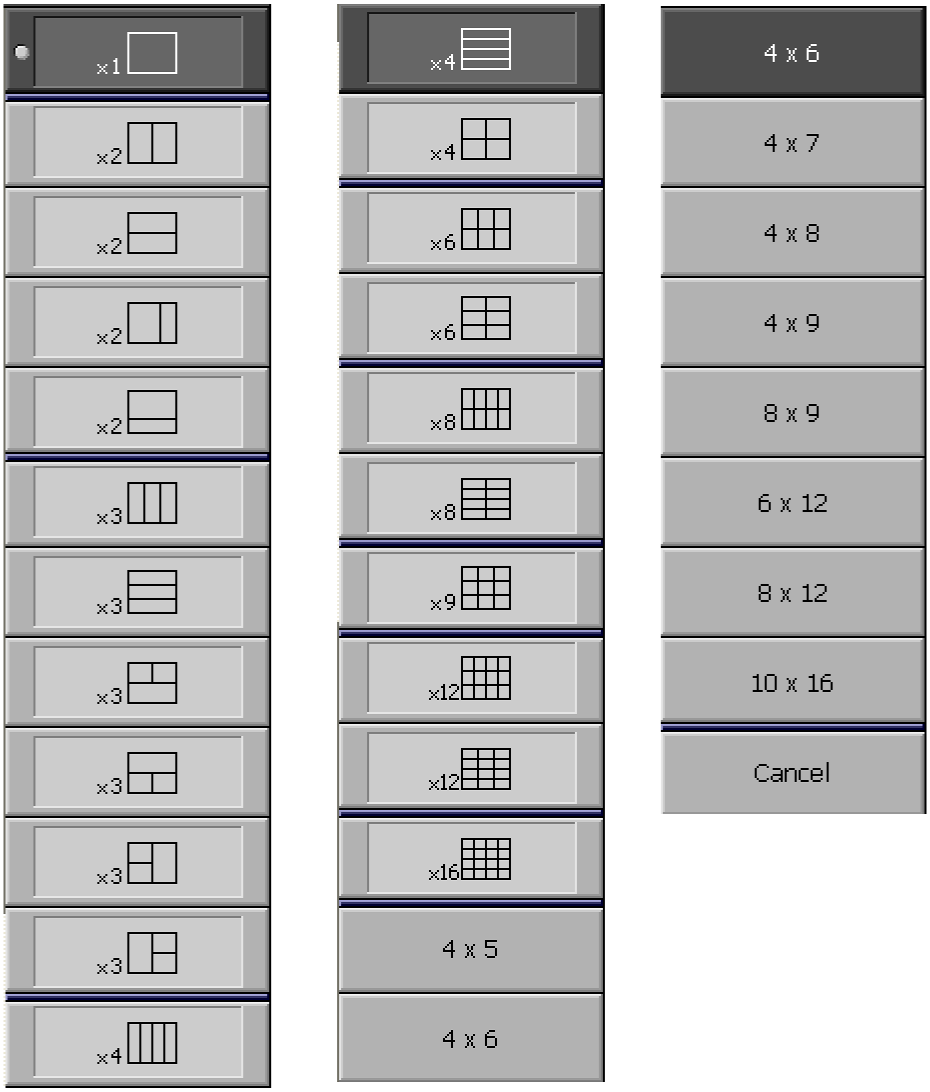
The measurement result for each channel is displayed in its dedicated window (channel window). You cannot have a single window display the measurement results from more than one channel. This means that the setting of the window layout determines the number of channels displayed on screen.

NOTE The execution of measurement for each channel does not depend on how the channel is displayed (channels that are not displayed can be measured). For information on executing measurement for each channel (trigger mode and trigger source), refer to Making Measurements.

The procedure for setting the window layout is as follows:

1. Press **Display** key > **Allocate Channels**.
2. Press the desired softkey to select the window layout shown below.

Windows Layout



e5071c061

NOTE From Firmware revision A.09.10, channel display configuration of 10×16 is added.
From Firmware revision A.08.10, channel display configuration of 8×9, 6×12 and 8×12 is added.

Setting Trace Display

Setting the number of traces

Depending on the measurement parameters of the traces displayed for each channel, the sweep necessary for each channel is executed. For more information, refer to Sweep Order in Each Channel.

You specify the trace display by setting the number of traces (upper limit of displayed trace numbers). For example, if you set the number of traces to 3, traces 1 through 3 are displayed.

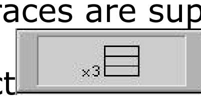
The procedure for setting the number of traces is as follows:

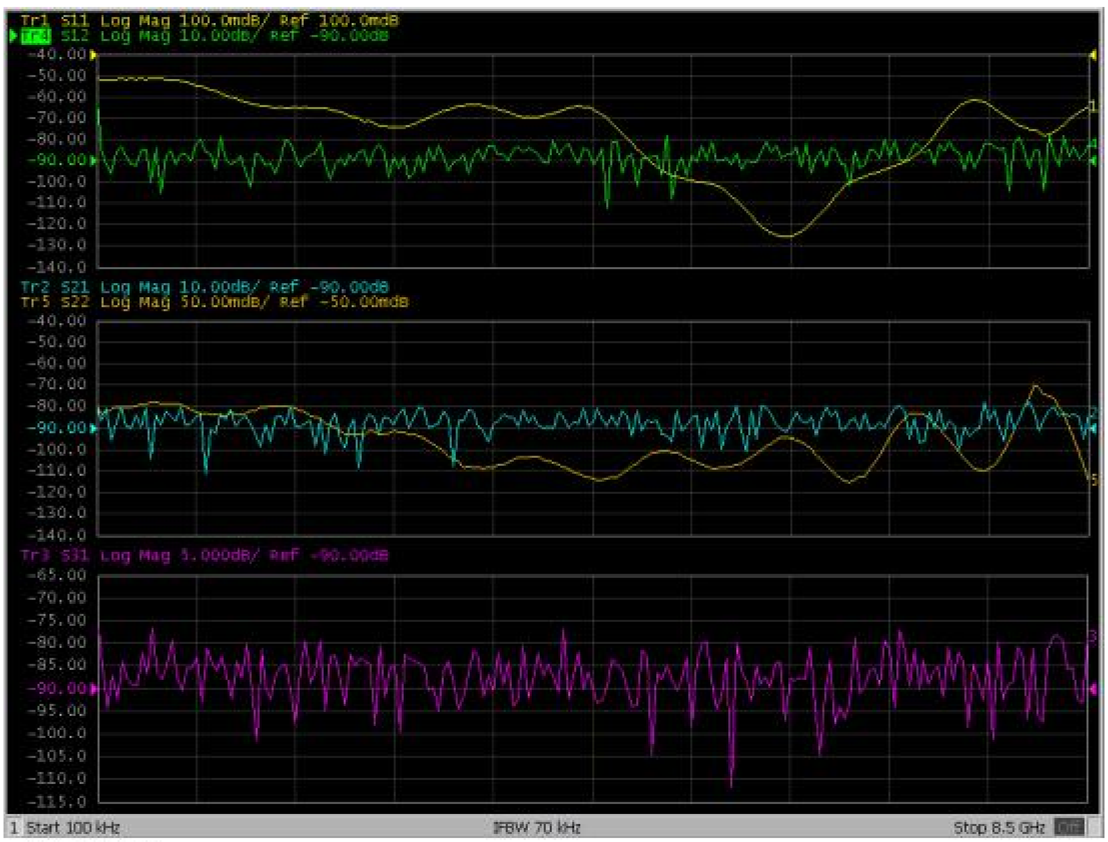
1. Press **Channel Next** or **Channel Prev** to select the channel for which you want to set the number of traces.
2. Press **Display** > **Number of Traces**.
3. Press the desired softkey to set the number of traces.

Setting trace layout (graph layout)

Traces are laid out and displayed in the order of the trace number from graph 1 according to the graph layout in the channel window.

You can select the graph layout from the windows layout.

If the number of traces is less than the number of graphs, nothing is displayed in the remaining area. If the number of traces you set exceeds the number of graphs, excess traces are superimposed from the first graph. For example, if you select  as the graph layout and set the number of traces to 5, graph 1 (Gr1 in Graph layout) and graph 2 (Gr2 in Graph layout) display traces 1 and 4 and traces 2 and 5, respectively, by superimposing, and graph 3 (Gr3 in Graph layout) displays only trace 3 as shown in the figure below.

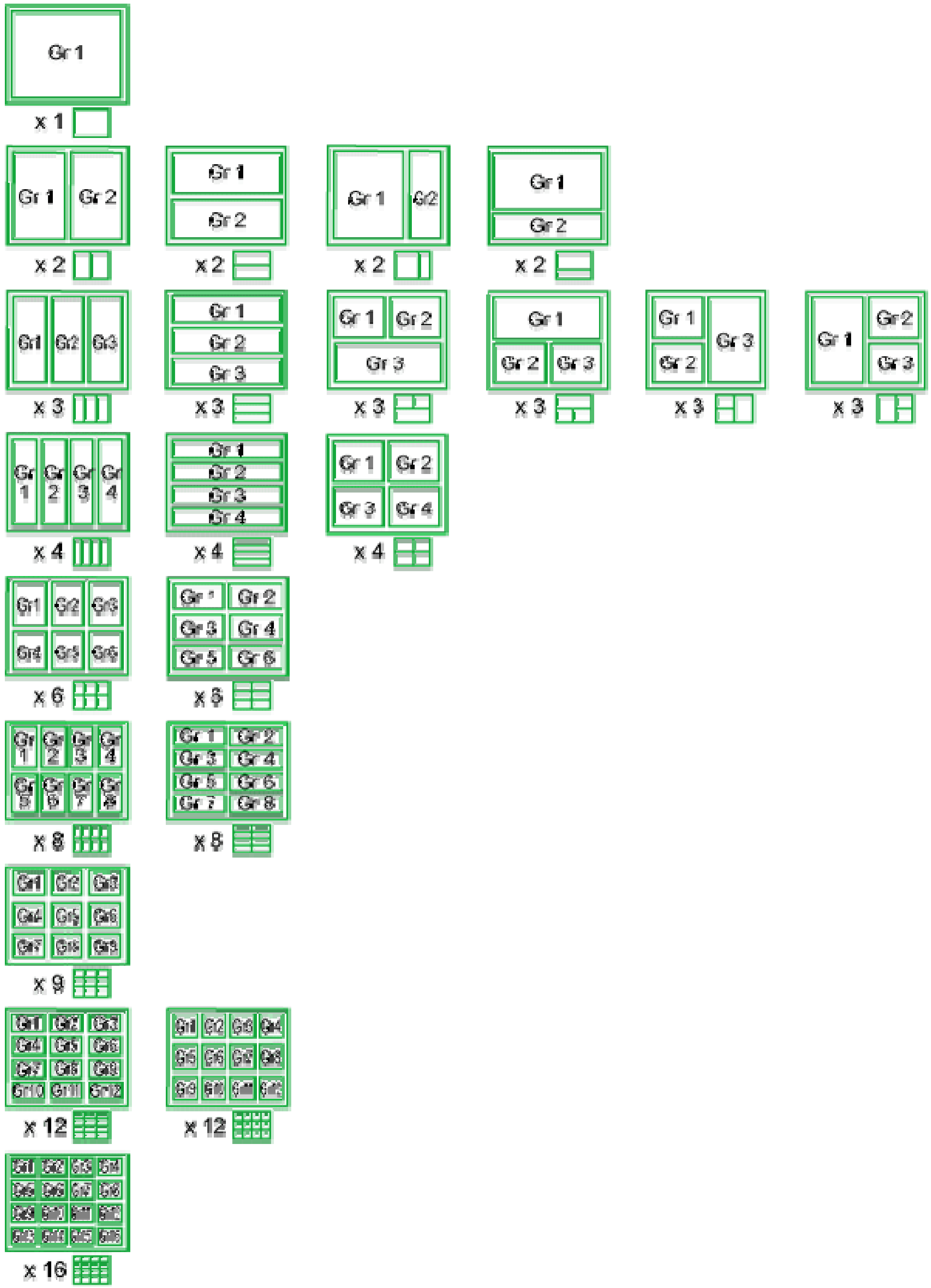


e5071c059

The procedure for setting the graph layout is as follows:

1. Press **Channel Next** or **Channel Prev** to select the channel for which you want to set the graph layout.
2. Press **Display** > **Allocate Traces.**
3. Press the desired softkey to select the graph layout shown below.

Graph Layout


Gr 1
x 1
Gr 1
Gr 2
x 2
x 2
x 2
x 2
Gr 3
x 3
x 3
x 3
x 3
x 3
x 3
Gr 4
x 4
x 4
x 4
x 6
x 6
x 8
x 8
x 9
x 12
x 12
x 16

e5071c471

Active Channel

The *active channel* is the one whose settings can currently be changed. The window frame of the active channel is displayed brighter than the window frames of the other channels. To change the settings specific to a certain channel, you must first activate the channel.

To change the active channel, use the following hardkeys:

Hardkey	Function
Channel Next	Change the active channel to the next channel with the larger channel number.
Channel Prev	Change the active channel to the previous channel with the smaller channel number.

Active trace

The active trace is the one whose settings can currently be changed. The trace name on the screen (for example, Tr3) of the current active trace is highlighted and indicated with ▶ to the left. To change the settings specific to a certain trace, you must first activate the trace.

To select the active trace, use the following hardkeys:

Hardkey	Function
Trace Next	Change the active trace to the next trace with the larger trace number.
Trace Prev	Change the active trace to the previous trace with the smaller trace number.

Setting Parameter for each Setup Item (Analyzer, Channel, Trace)

The following table lists the setting parameters and indicates the setup item (analyzer, channel, or trace) that each parameter controls along with the applicable setup key(s).

Parameter	Controlled Setup Items			Setup Key(s)
	Analyzer	Channel	Trace	
Stimulus Settings				
Sweep range		x		**Start**, **Stop**, **Center**, **Span**
Power, CW frequency		x		**Sweep Setup** > **Power**

Sweep time/Sweep delay time		x		**Sweep Setup** > **Sweep Time, Sweep Delay**
Number of points		x		**Sweep Setup** > **Points**
Segment sweep		x		**Sweep Setup** > **Sweep Type, Edit Segment Table, Segment Display**
Sweep mode		x		**Sweep Setup** > **Sweep Mode**
Trigger Settings				
Trigger source	x			**Trigger** > **Trigger Source/Restart/Trigger**
Trigger mode		x		**Trigger** > **Hold/Hold All Channels/Single/ Continuous/Continuous Disp Channels**
Response Settings				
Measurement parameter			x	**Meas**
Data format			x	**Format**
Scale, Electrical delay, Phase offset			x	**Scale**
Memory trace and data math			x	**Display** > **Display/Data-> Mem/ Data Math**
Window title		x		**Display** > **Edit Title Labe/ Title Label (ON/OFF)**
Graticule label in rectangular form		x		**Display** > **Graticule Label (ON/OFF)**
Color inversion	x			**Display** > **Invert Color**
Frequency display ON/OFF	x			**Display** > **Frequency (ON/OFF)**
Display update ON/OFF	x			**Display** > **Update (ON/OFF)**
Averaging		x		**Avg** > **Averaging Restart/ Avg Factor/Averaging (ON/OFF)**

Smoothing			x	**Avg** > **Smo Aperture/ Smoothing (ON/OFF)**
IF bandwidth		x		**Avg** > **IF Bandwidth**
Calibration		x		**Cal**
Marker			x	**Marker**, **Marker Search**, **Maker Fctn**
Analysis				
Fixture simulator		x		**Analysis** > **Fixture Simulator**
Time domain			x	**Analysis** > **Gating/ Transform**
Parameter conversion			x	**Analysis** > **Conversion**
Limit test			x	**Analysis** > **Limit Test**
Saving and recalling data	x			**Save/Recall**
Macro	x			**Macro Setup**, **Macro Run**, **Macro Break**
System				
Printing/Saving display Screen/Beeper/GRIB settings/Network Settings/Date & Time/Key Lock/Backlight/Firmware Revision/Service menu	x			**System**
Preset	x			**Preset**

Setting the System Z_0

The procedure for setting the system characteristic impedance (Z_0) is as follows:

1. Press **Cal** key.
2. Click **Set Z_0**, then input the system Z_0.

Other topics about Setting Measurement Conditions

Setting Stimulus Conditions

You can set the stimulus condition for each channel independently.

- Setting the Sweep Type
- Setting the Sweep Range
- Enable Stimulus Signal Output
- Setting power level
- Setting fixed frequency at power sweep
- Setting CW Time Sweep
- Setting Power Level with Auto Power Range
- Setting the Number of Points
- Setting the Sweep Time

Other topics about Setting Measurement Conditions

Setting the Sweep Type

You can select the sweep type from the following four types.

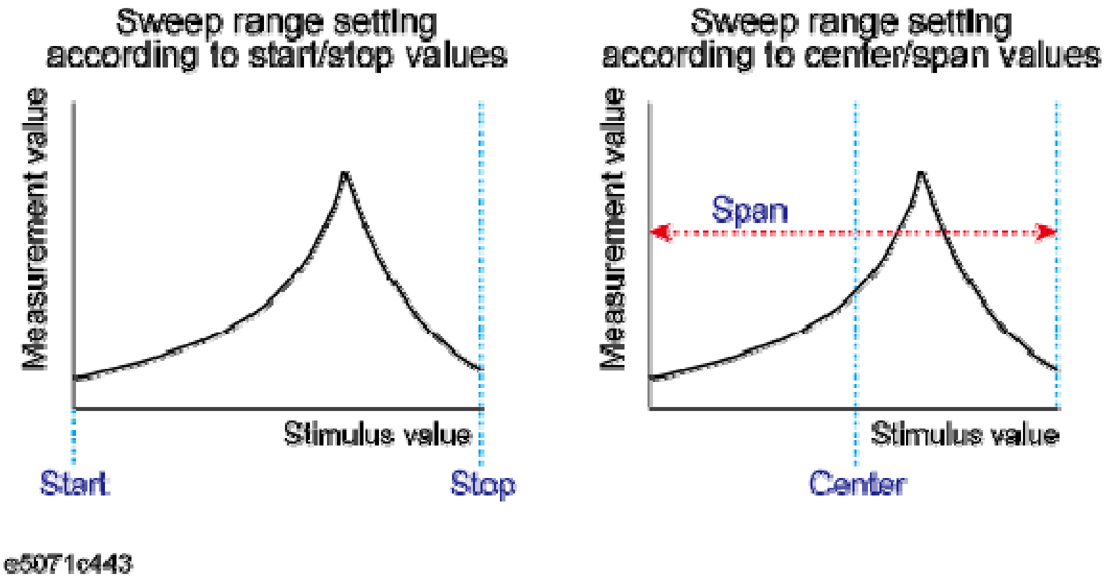
SoftKey	Description
Linear Freq	Sweeps frequencies in linear scale.
Log Freq	Sweeps frequencies in logarithmic scale.
Segment	Performs a sweep with linear sweep conditions (segments) combined. For more information, refer to Performing a Segment-by-Segment Sweep (segment sweep).
Power Sweep	Sweeps power levels in linear scale.

The procedure for selecting the sweep type is as follows:

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to set the sweep type.
2. Press **Sweep Setup** > **Sweep Type**.
3. Press the desired softkey to select the sweep type.
    4. To use the E5071C in CW Time sweep, refer to Setting CW Time Sweep

Setting the Sweep Range

There are two ways to set the sweep range: by specifying the lowest and the highest values and by specifying the center value and a span. Once the sweep range is set, it is possible to change the range by substituting the lowest value, the highest value, or the center value with a value (stimulus value) represented by a marker on the trace.



Setting the Sweep Range with the Lowest and Highest Values

1. Press **Channel Next**/**Channel Prev** keys to select the channel whose sweep range will be set.
2. Click **Start**, then input the lowest value.
3. Click **Stop**, then input the highest value.

Setting the Sweep Range with the Center Value and a Span

1. Press **Channel Next**/**Channel Prev** keys to select the channel whose sweep range will be set.
2. Click **Center**, then input the center value.
3. Click **Span**, then input the span value.

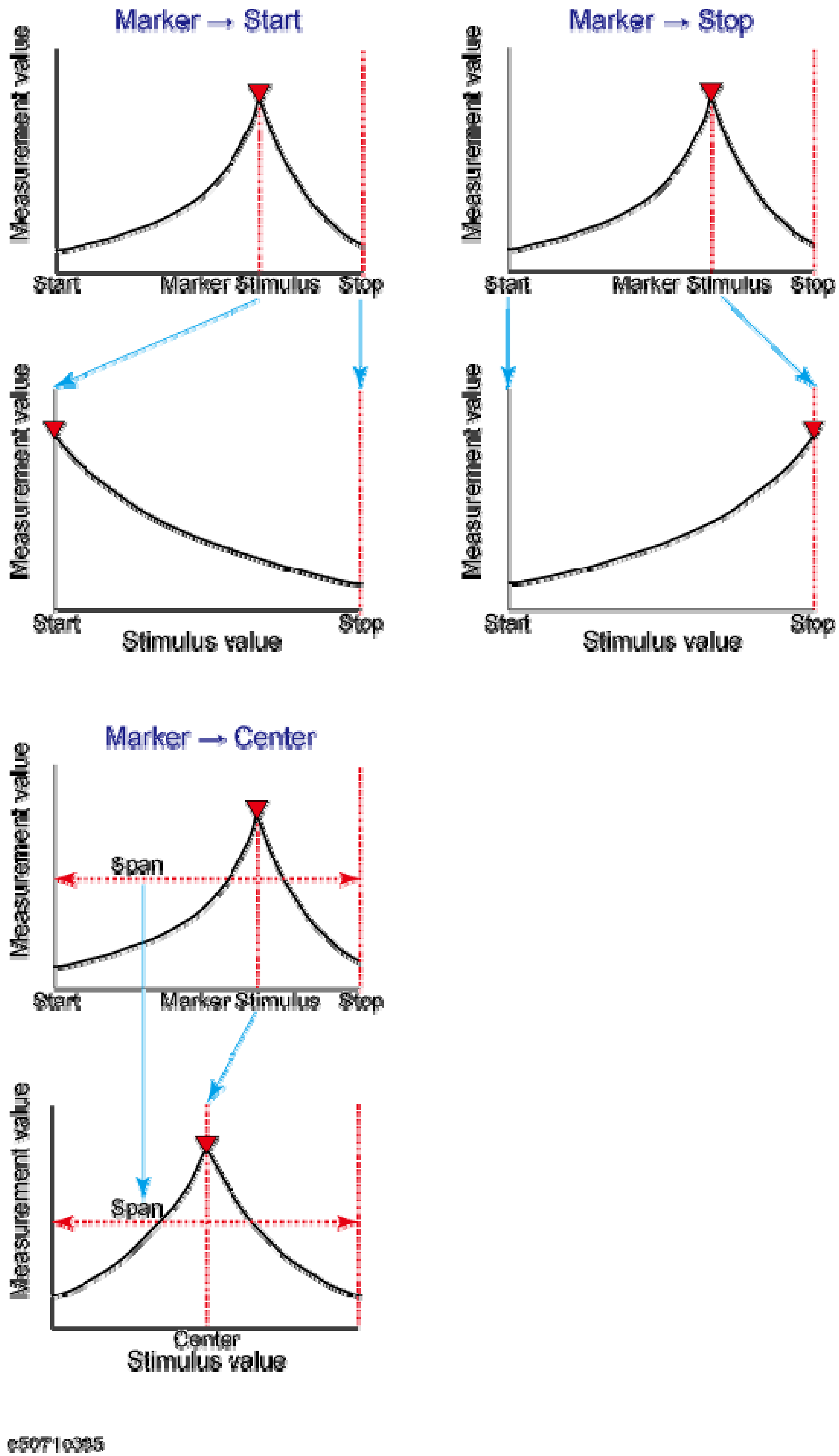
Setting the Sweep Range Using the Marker

1. In the channel window whose range must be set, place the *active marker* on the active trace to a position that corresponds to the new range (to the lowest, highest, or center value).
2. Press **Marker Fctn** key.
3. Click the softkey that corresponds to each value.

SoftKey	Function
Marker ->	Sets the lowest value to the stimulus value of the

Start	active marker on the currently active trace.
Marker -> Stop	Sets the highest value to the stimulus value of the active marker on the currently active trace.
Marker -> Center	Sets the center value to the stimulus value of the active marker on the currently active trace.

NOTE If the reference marker is on and the stimulus value of the active marker is expressed by a value relative to the reference marker, the absolute stimulus value will be used to set the new sweep rang.


Marker → Start
Measurement value
Start
Marker Stimulus
Stop
Measurement value
Start
Stop
Stimulus value
Marker → Stop
Measurement value
Start
Marker Stimulus
Stop
Measurement value
Start
Stop
Stimulus value
Marker → Center
Measurement value
Span
Start
Marker Stimulus
Stop
Measurement value
Span
Center
Stimulus value
e5071c365

Enable Stimulus Signal Output

You can turn on/off the stimulus signal output, but this will prevent you from performing measurement. Therefore, you will not normally use this feature. This is mainly used to turn the output back to on after it has been turned off by the power trip feature.

Follow these steps to turn the stimulus signal output on/off:

1. Press **Sweep Setup** key.
2. Click **Power** > **RF Out** (Each press toggles between on/off).

When set to off, "RF OFF" is displayed in Instrument Status Bar.

Power trip

The power trip is a feature that the instrument uses to automatically turn off the output of the stimulus signal to protect the instrument when a signal whose level exceeds the upper limit is inputted to the test port.

If the power output is automatically turned off by the power trip feature, remove the cause of the over-input and turn on the power output according to the above steps to restart the measurement.

Setting power level

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key.
3. Click **Power** > **Port Couple**, then select the on/off setting of the level coupling for all ports.

ON	The same power level is outputted to all ports.
OFF	A specific power level is outputted to each port independently.

NOTE The power level of port 1 is coupled with the power level for all ports.

4. If you change the on/off setting of the level coupling, all ports are automatically changed to the same level value as that of port 1.

4. Follows the following procedure according to Port Couple.

- When setting level for all ports (Port Couple ON)
  a. Click **Power**, then enter the power level.
- When setting level for each port (Port Couple OFF)

a. Press **Port Power**, then click the softkey corresponding to each port (Port 1 Power to Port 4 Power).

b. Enter the power level.

Correcting attenuation of power level (using power slope feature)

You can use the power slope feature to correct the attenuation of a power level so that it is simply proportional to the frequency (attenuation due to cables and so on), which improves the accuracy of the level actually applied to the DUT.

Turning power slope feature on/off

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key.
3. Click **Power** > **Slope [OFF]** (**Slope [ON]**). Each press toggles between on/off.

Setting correction coefficient (correction amount per 1 GHz)

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key.
3. Click **Power** > **Slope [xxx dB/GHz]** ("xxx" represents the current set value.).
4. Enter the correction coefficient using the **ENTRY** block keys on the front panel.

Setting fixed frequency at power sweep

The procedure for setting the fixed frequency (CW frequency) at the power sweep is as follows:

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key.
3. Click **Power** > **CW Freq**, then enter the fixed frequency.

Setting CW Time Sweep

The E5071C supports sweep type of CW Time where the E5071C is set to a single frequency, and the data is displayed versus time. To use the E5071C in CW time, follow the procedure explained below:

1. Press **Channel Next**/**Channel Prev** keys to select the channel whose sweep range will be set.
2. Press **Sweep Setup** > **Sweep Time**, then input the duration of the sweep which is displayed on X-axis.
3. Click **Span**, then input the span value as 0 Hz (zero span).

4. Press **Center**, then input the center value.
5. Press **Marker** to display the marker 1. The marker position shows the time for its measurement point.

Setting Power Level with Auto Power Range (options 2D5, 4D5, 2K5 and 4K5 only)

NOTE Auto Power range function works only with the E5071C options 2D5, 4D5, 2K5 and 4K5 having a mechanical source attenuator. Auto power range functionality is neither required nor available in other E5071C options which are having electrical source attenuator.

When the Auto Power Range set function is effective, the proper source attenuator and power range are selected automatically according to the output power setting of each channel.

Under the following conditions, turn OFF the Auto Power Range set function and set the power range and power level manually:

- When different power ranges are selected in multiple channel measurement settings. In this case, an error message is displayed to avoid source attenuator damage. For example, if Channel 1 power is at -35 dBm with the power range of -35 dBm to 0 dBm, and Channel 2 power is set at -5 dBm with the power range of -25 dBm to +10 dBm, this will causes an error because each selected power range is different. In such a case , turn OFF the Auto Range on Channel 2 and set the range as -35 to 0 manually, to set the same power range as of Channel 1.

CAUTION Continuous switching may damage source attenuator.

Setting the level

1. Press **Channel Prev**/**Channel Next** to select the channel for which you want to set the power level.
2. Press **Sweep Setup** key, then click **Power** > **Port Couple** and select the **ON**/**OFF** setting of the level coupling for all ports.

Value	Description
ON	The same power level is outputted to all ports.
OFF	A specific power level is outputted to each port independently.

1. The power level of port 1 is coupled with the power level for all ports when Port Couple is set to ON.

2. If you change the on/off setting of the level coupling, all ports are automatically changed to the same level value as that of port 1.

3. When setting level for all ports (**Port Couple ON**), click **Power** input the power level using the **ENTRY** block keys on the front panel.

When setting level for each port (**Port Couple OFF**), click **Port Power** and select the softkey corresponding to each port (**Port 1 Power** to **Port 4 Power**).

The table below describes the relation between source power level and step attenuation:

Source Power Range	Step Attenuation
-25 ~ +10 dBm	0 dB
-35 ~ 0 dBm	10 dB
-45 ~ -10 dBm	20 dB
-55 ~ -20 dBm	30 dB
-65 ~ -30 dBm	40 dB
-75 ~ -40 dBm	50 dB
-85 ~ -50 dBm	60 dB

For Auto Power Range setting, the minimum value of step attenuation is selected according to the available source power level. The table below describes the step attenuation selection based on the setting source power level:

Source Power Range	Step Attenuation
-25 ≤ power ≤ +10 dBm	0 dB
-35 ≤ power < -25 dBm	10 dB
-45 ≤ power < -35 dBm	20 dB
-55 ≤ power < -45 dBm	30 dB
-65 ≤ power < -55 dBm	40 dB
-75 ≤ power < -65 dBm	50 dB

-85 ≤ power < -75 dBm	60 dB

When Power Sweep is selected as sweep type, Step attenuation is selected with the start power condition and the maximum value of the Stop power (>= Start Power) is selected with the step ATT value.

For example:

- Case of Start Power= -26 dBm
    - Step ATT = 10 dB, Stop Power = -26 dBm ~ 0 dBm
- Case of Start Power = -25 dBm
    - Step ATT = 0 dB, Stop Power = -25 dBm ~ + 10 dBm
- If the source power range (start/stop) is located on the two ranges, the Step ATT is set with the latest condition as:
    - After setting Start = -26 dBm, set stop = + 10 dBm, then start is reset with - 25 dBm value.
    - Available power differs according to the measurement frequency. For example, at 0 dBm Attenuator range, the specified maximum available power level is as follows and reduces in proportion to inserted Attenuator range:

Frequency	Power Level
300 kHz to 1 MHz	+8 dBm
1MHz to 6 GHz	+10 dBm
6GHz to 8 GHz	+9 dBm
8GHz to 10.5 GHz	+7 dBm
10.5GHz to 15 GHz	+3 dBm
15 GHz to 20 GHz	0 dBm

Correcting attenuation of power level (using power slope feature)

You can use the power slope feature to correct the attenuation of a power level so that it is simply proportional to the frequency (attenuation due to cables and so on), which improves the accuracy of the level actually applied to the DUT.

Turning power slope feature on/off

1. Press **Channel Prev**/**Channel Next** to select the channel for which the power slope feature will be turned OFF.

2. Press **Sweep Setup** key, then click **Power** > **Slope** and select the **ON**/**OFF** setting of the power level.

Setting correction coefficient (correction amount for 1 GHz)

1. Press **Channel Prev**/**Channel Next** to select the channel for which you want to set the correction coefficient.
2. Press **Sweep Setup** key, then click **Power** > **Slope [xxx dB/GHz]** where "xxx" represents the current set value and input the correction coefficient using the **ENTRY** block keys on the front panel.

Setting the Number of Points

The number of points is the number of data items collected in one sweep. It can be set to any number from 2 to 1601 for each channel independently.

- To obtain a higher trace resolution against the stimulus value, choose a larger value for number of points.
- To obtain higher throughput, keep the number of points to a smaller value within an allowable trace resolution.
- To obtain higher measurement accuracy after calibration, perform calibration using the same number of points as in actual measurements.
    - When the upper limit of the number of channels and traces is set to **1Ch / 4 Tr 20001 Points**, the maximum number of measurement points will be 20001.

Setting the number of points

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key.
3. Click **Points**, then input the desired number of points.

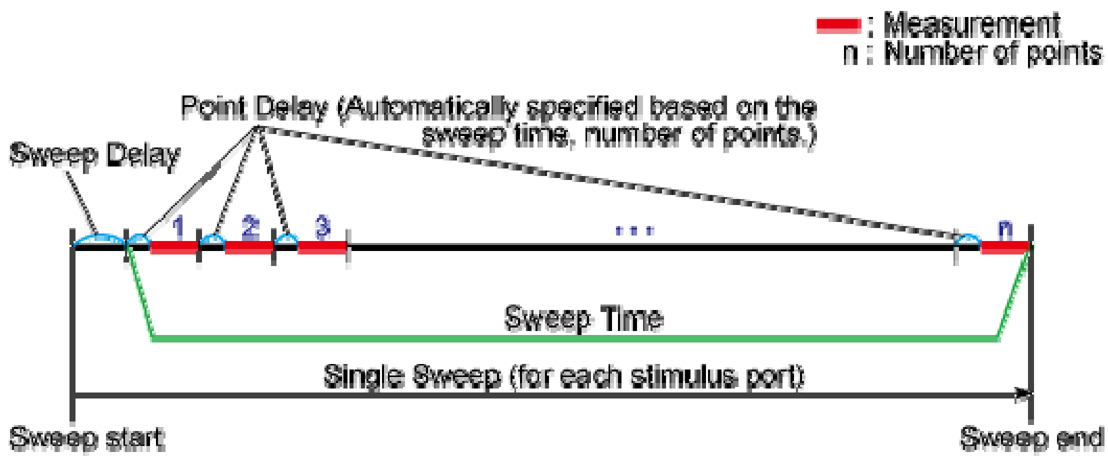
Setting the Sweep Time

Sweep time is the time it takes to complete a sweep for each stimulus (source) port. Two modes are available for setting the sweep time: manual sweep time mode and automatic sweep time mode.

Manual Sweep Time Mode	In this mode, the sweep time is set manually. Once the sweep time is set, changes in measurement conditions will not affect the sweep time as long as it is within the analyzer's capability. If the sweep time becomes lower than the analyzer's lower sweep time limit, the sweep time will be reset to the shortest time within the conditions. If the sweep time exceeds the analyzer's upper sweep time limit, the sweep time will be reset to

	the longest time within the conditions.
Automatic Sweep Time Mode	The sweep time is always kept to the shortest time possible with the current measurement conditions.

The following figure shows the definitions of the sweep time and the sweep delay time.



e5071c396

Sweep delay is time before starting a sweep for each stimulus (source) port

Setting Up the Sweep Time (Manual Sweep Time Mode)

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key, then Click **Sweep Time.**
3. Using the **ENTRY** block keys on the front panel, input the desired sweep time (in seconds).

If the previous operation mode was automatic sweep time mode, entering a new sweep time forces the analyzer to switch to manual sweep time mode.

Switching to Automatic Sweep Time Mode

1. Press **Channel Next**/**Channel Prev** keys to select the desired channel.
2. Press **Sweep Setup** key, then click **Sweep Time**, press **0** > **x1**.

By entering zero (seconds), automatic sweep time becomes effective.

Selecting Measurement Parameters

The E5071C allows users to evaluate the DUT (device under test) characteristics by using the following measurement parameters.

- S-parameters
- Absolute
- AUX Input

Other topics about Setting Measurement Conditions

Operation Procedure

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace for which measurement parameters will be set up.
2. Press **Meas**.
3. Click a softkey that corresponds to the desired measurement parameter.

S-parameters

S-parameters (scattering parameters) are used to evaluate how signals are reflected by and transferred through the DUT. An S-parameter is defined by the ratio of two complex numbers and contains information on the magnitude and phase of the signal. S-parameters are typically expressed as follows:

$S_{out\ in}$

out: port number of the DUT from which the signal is output

in: port number of the DUT to which the signal is input

For example, S-parameter S_{21} is the ratio of the output signal of port 2 on the DUT with the input signal of port 1 on the DUT, both expressed in complex numbers.

When the balance-unbalance conversion function is turned on, the Mixed mode S-parameters can be selected. Refer to Evaluating Balanced Devices (balance-unbalance conversion function for more information.

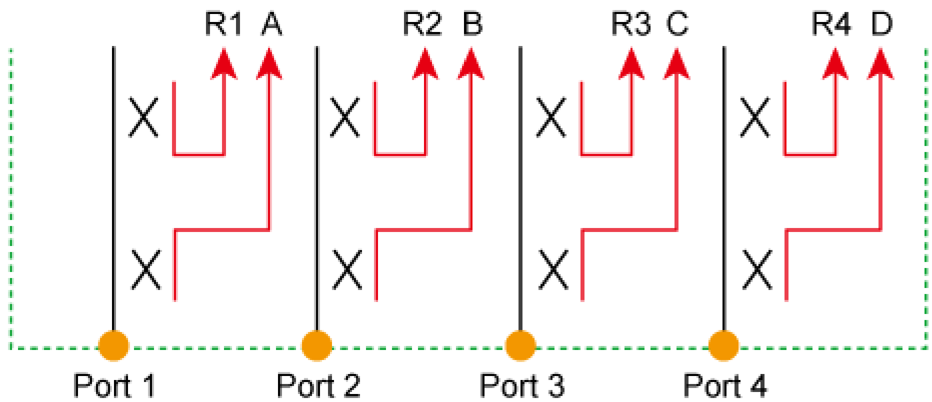
Absolute

Absolute shows the absolute power for reference and received signals on the port.

Softkey	Description
A (n)	Absolute measurement in Port 1, test receiver

B (n)	Absolute measurement in Port 2, test receiver
C (n)	Absolute measurement in Port 3, test receiver
D (n)	Absolute measurement in Port 4, test receiver
R1 (n)	Absolute measurement in Port 1, reference receiver
R2 (n)	Absolute measurement in Port 2, reference receiver
R3 (n)	Absolute measurement in Port 3, reference receiver
R4 (n)	Absolute measurement in Port 4, reference receiver

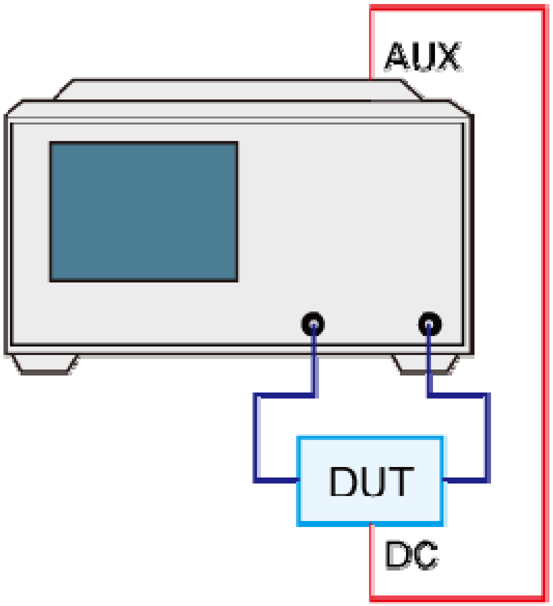
where n in the parentheses is the stimulus port number. For example, R1(1) means the reference level while the signal is output from the port 1, and A(2) means the received signal level into port 1 while the signal is output from the port 2.



AUX Input

The AUX Input Ports can be used to input DC signal for DC signal measurement. This is useful in cases where the DUT (Device Under Test) works on a DC supply and it is required to measure the DC supply along with other measurements of the DUT using the E5071C.

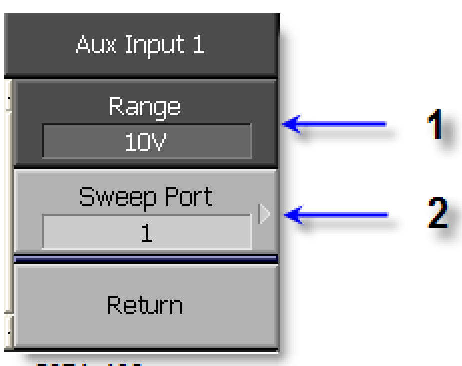
Example of AUX Input Measurement



e5071c209

To use AUX Input Measurement:

1. Select **Aux Input 1** or **Aux Input 2** (depending upon the Aux port used for connection) by pressing the **Meas** key.
2. Select **Range** (*1V* or *10 V*) [**1** in the figure below].



e5071c198

3. Select **Sweep Port** (*1-4*) [**2** in the figure above].
4. Click **Format** > **Real**.
5. Perform measurements as per normal.
    6. Sweep Port [**2** in the figure above] specifies the signal output port. For example, if sweep port is set to 1, the AUX input signal is measured while port 1 on the front panel outputs the RF signal.

Selecting a Data Format

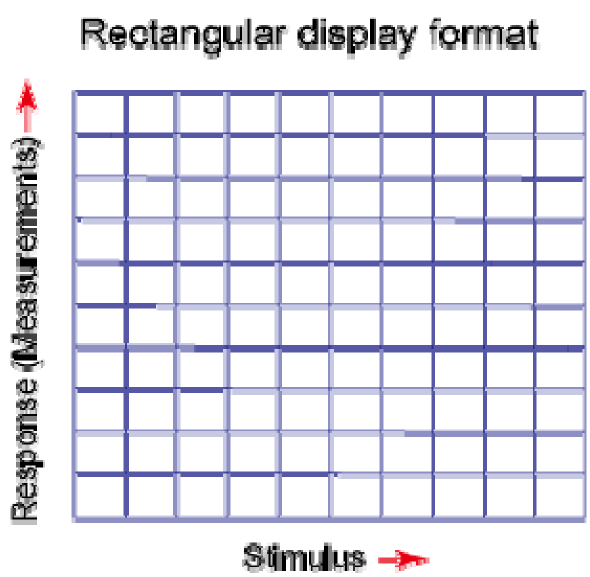
The E5071C allows you to display measured S-parameters by using the following data formats. The data format can be preset to factory settings using the Preset option.

- Rectangular display formats
- Polar format
- Smith chart format

Other topics about Setting Measurement Conditions

Rectangular display formats

Rectangular display formats draw traces by assigning stimulus values (linear scale) to the X-axis and response values to the Y-axis. Eight different formats are available depending on the selection of data for the Y-axis.



e5071c451

Type	Y-axis Data Type	Y-axis Unit	Application Examples
Log magnitude format	Magnitude	dB	• Return loss measurement • Insertion loss measurement (or gain measurement)

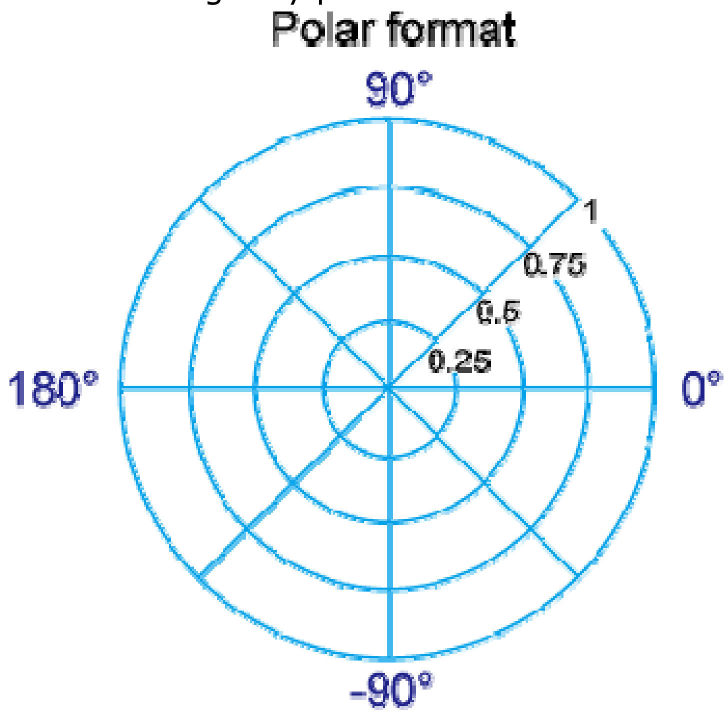
Phase format	Phase (displayed in range from -180 ° to +180 °)	Degrees (°)	Measurement of deviation from linear phase
Expanded phase format	Phase (can be displayed above +180 ° and below -180 °)	Degrees (°)	Measurement of deviation from linear phase
Positive phase format	Phase (displayed in range from 0 ° to +360 °)	Degrees (°)	Measurement of deviation from linear phase
Group delay format	Signal transfer delays within the DUT	Seconds (s)	Group delay measurement
Linear magnitude format	Magnitude	(Abstract number)	Reflection coefficient measurement
SWR format	$\frac{1+\rho}{1-\rho}$ (ρ: reflection coefficient)	(Abstract number)	Measurement of standing wave ratio
Real format	Real part of measured complex parameter	(Abstract number)	
Imaginary format	Imaginary part of measured complex parameter	(Abstract number)	

Polar format

In the polar format, traces are drawn by expressing the magnitude as a displacement from the origin (linear) and phase in an angle counterclockwise from the positive X-axis. This data format does not have a stimulus axis, so frequencies must be read by using the marker. The polar format allows users to select one of the following three data groups for displaying the marker response values.

- Linear magnitude and phase (°)
- Log magnitude and phase (°)

- Real and imaginary parts

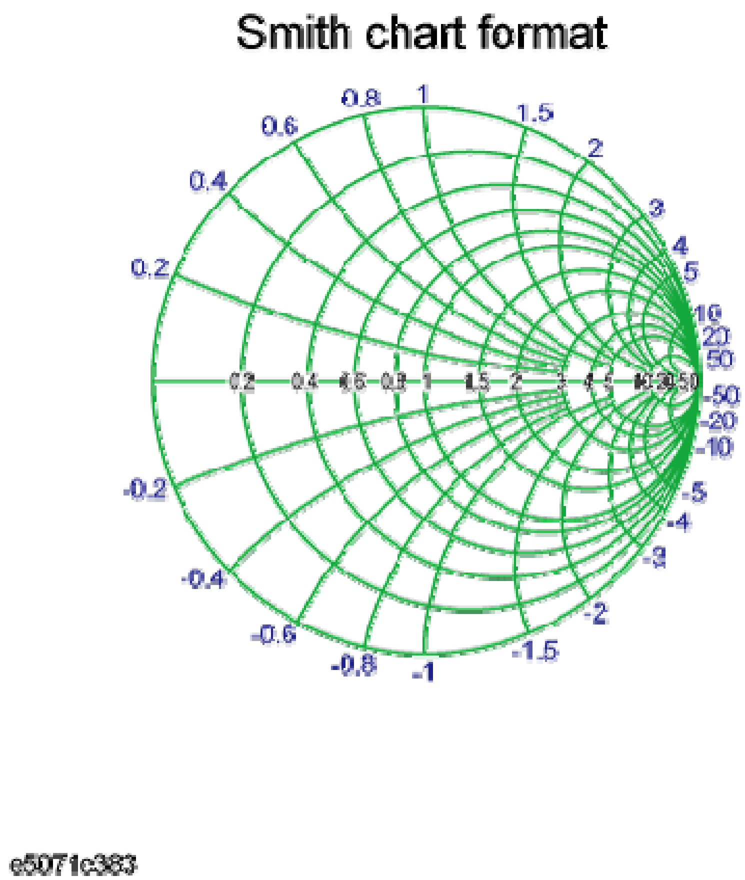


e5071c382

Smith chart format

The Smith chart format is used to display impedances based on reflection measurement data of the DUT. In this format, traces are plotted at the same spots as in the polar format. The Smith chart format allows users to select one of the following five data groups for displaying the marker response values.

- Linear magnitude and phase (°)
- Log magnitude and phase (°)
- Real and imaginary parts
- Resistance (ohm), Reactance (ohm), and inductance (H) or capacitance (F)
- Conductance (S), susceptance (S), and capacitance (F) or inductance (H)



Use the following procedure to select a data format.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace for which the data format will be set.
2. Press **Format.**
3. Press the softkey that corresponds to the desired data format.

Softkey	Function
Log Mag	Selects the log magnitude format
Phase	Selects the phase format
Group Delay	Selects the group delay format
Smith - Lin / Phase	Selects the Smith chart format (with linear magnitude and phase as the marker response values)

Smith - Log / Phase	Selects the Smith chart format (with log magnitude and phase as the marker response values)
Smith - Real / Imag	Selects the Smith chart format (with the real and imaginary parts as the marker response values)
Smith - R + jX	Selects the Smith chart format (with resistance and reactance as the marker response values)
Smith - G + jB	Selects the Smith chart format (with conductance and susceptance as the marker response values)
Polar - Lin / Phase	Selects the polar format (with linear magnitude and phase as the marker response values)
Polar - Log / Phase	Selects the polar format (with log magnitude and phase as the marker response values)
Polar - Real / Imag	Selects the polar format (with the real and imaginary parts as the marker response values)
Lin Mag	Selects the linear magnitude format
SWR	Selects the SWR (standing wave ratio) format
Real	Selects the real format
Imaginary	Selects the imaginary format
Expand Phase	Selects the expanded phase format
Positive Phase	Selects the positive phase format

Setting the Scales

- Auto Scale
- Manual scale adjustment on a rectangular display format

Other topics about Setting Measurement Conditions

Auto scale

The auto scale function is used to tailor each scale (scale/division and the reference line value) automatically in such a way that traces will appear at the proper size on the screen for easy observation.

NOTE The scale data can be preset to factory settings using the Preset option.

Single Trace Auto Scale

Follow the procedure below to perform the auto scale function on a specific trace.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace for which the auto scale function will be performed.
2. Press **Scale** > **Auto Scale**.

Auto Scale on All Traces Within a Channel

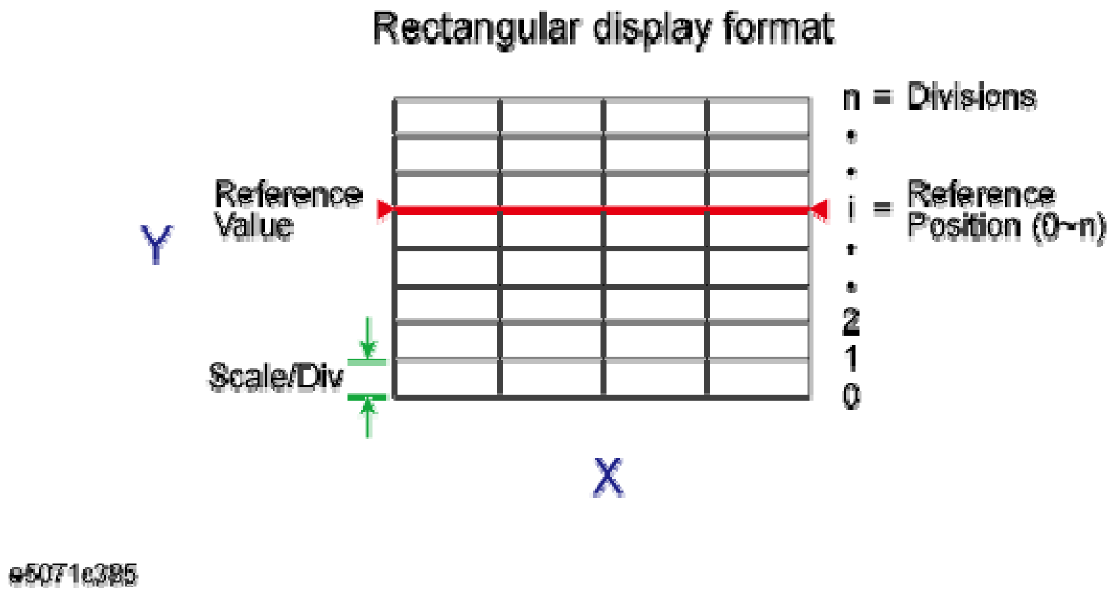
1. Press **Channel Next**/**Channel Prev** keys to select the channel for which the auto scale function will be performed.
2. Press **Scale** > **Auto Scale**.

Manual scale adjustment on a rectangular display format

For a rectangular display format, four parameters are used to manually adjust the scales.

Adjustable feature	Description
Divisions	Defines the number of divisions on the Y-axis. An even number from 4 to 30 must be used. Once set, it is commonly applied to all traces displayed in any rectangular format within that channel.
Scale/Division (Scale/Div)	Defines the number of increments per division on the Y-axis. The value applies only to the active trace.
Reference position	Defines the position of the reference line. The position must be specified using the number assigned to each division on the Y-axis starting at 0 (the least significant) running up to the number of divisions being used (the most significant). The position applies

	only to the active trace.
Reference line value (Reference Value)	Defines the value corresponding to the reference line. It must be set using the unit on the Y-axis. The reference line value applies only to the active trace.



Manually setting scales on a rectangular display format

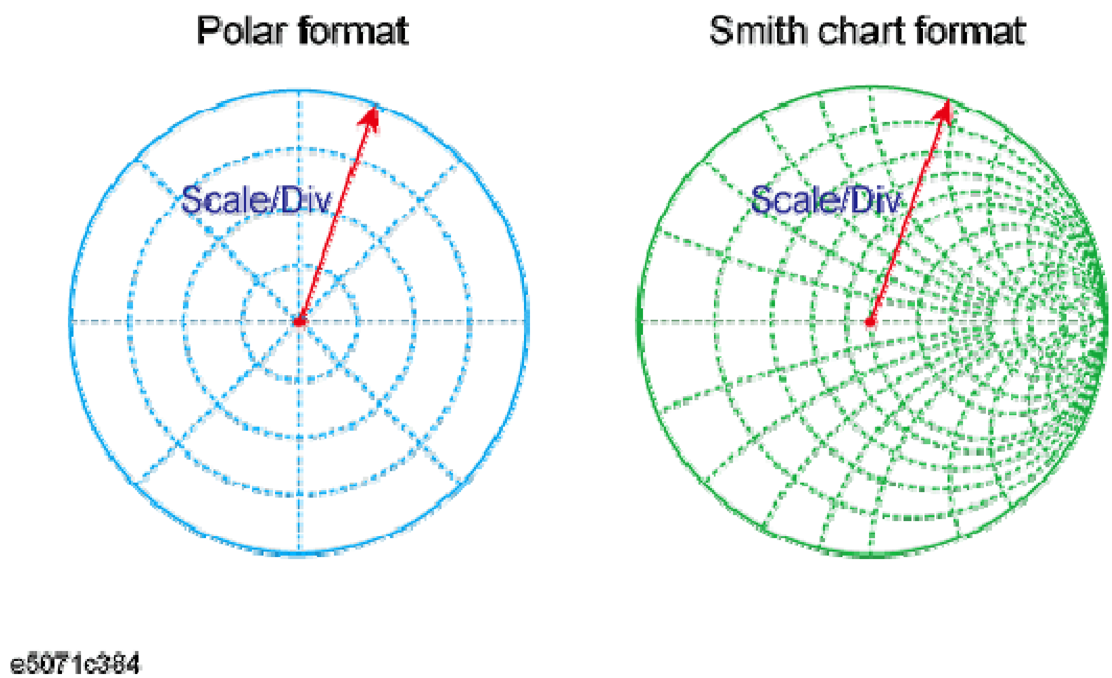
1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace for which scale features will be adjusted.
2. Press **Scale** key.
3. Select the softkey that corresponds to the particular feature that needs to be adjusted.

Softkey	Function
Divisions	Defines the number of divisions on the Y-axis.
Scale/Div	Defines the number of increments per division on the Y-axis.
Reference Position	Defines the position of the reference line.
Reference Value	Defines the value corresponding to the reference line.

NOTE It is also possible to turn off the display of graticule labels. For details, refer to Turning_off_the_display_of_graticule_labels.

Manual scale adjustment on the Smith chart/polar format

Manual scale adjustment on the Smith chart format or the polar format is done by using the displacement (**Scale/Div** of the outermost circle).



Manually setting scales on the Smith chart/polar format

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace for which the scale will be adjusted.
2. Press **Scale** key.
3. Click **Scale/Div**, then input the displacement of the outermost circle.

Setting the value of a reference line using the marker

When using a rectangular display format, it is possible to change the reference line value to be equal to the response value of the *active marker* on the active trace.

Setting the reference line value using the marker

1. Place the active marker on the active trace on the position that corresponds to the new reference line value.
2. Press **Scale** or **Marker Fctn** key.

3. Click **Marker -> Reference** to change the reference line value to the marker response value.
    4. If the reference marker is on and the stimulus value of the active marker is expressed using a value relative to the reference marker, the absolute stimulus value will be used to set the new reference line value.

Setting Window Displays

- Maximizing the specified window trace display
- Turning off the display of graticule labels
- Hiding Frequency Information
- Labeling a Window
- Setting Display Colors
- Setting Display Magnification
- Resizing the Screen

Other topics about Setting Measurement Conditions

Maximizing the specified window/trace display

When using multiple channels, it is possible to maximize a specific channel window on the screen. When multiple traces are displayed in a channel window, it is also possible to maximize a specific trace displayed within that channel window.

NOTE The Window/Trace Display data can be preset to factory settings using the Preset option

Maximizing a window

1. Press **Channel Next** (or **Channel Prev**) to select the channel whose window will be maximized.
2. Press **Channel Max** to maximize the channel window.
3. Press **Channel Max** one more time to reduce the window to its previous size.

Maximizing a trace display

1. Press **Channel Next** (or **Channel Prev**) to select the channel to which the trace belongs.
2. Press **Trace Next** (or **Trace Prev**) to select the trace whose display will be maximized.
3. Press **Trace Max** to maximize the trace display.
4. Press **Trace Max** one more time to reduce the display to its previous size.

Turning off the display of graticule labels

When using a rectangular display format, the graph area can be expanded to the left by turning off the display of graticule labels.

Turning off graticule label display

1. Press **Channel Next** (or **Channel Prev**) to select the channel for which graticule label display will be turned on or off.
2. Press **Display**.
3. Click **Graticule Label** to turn graticule label display on or off.

Hiding Frequency Information

You can hide the frequency information from the screen in order to ensure its confidentiality or for other reasons.

Hiding Frequency Information on the Screen

Follow the steps below to hide frequency information on the measurement screen.

1. Press **Display** key.
2. Click **Frequency** to turn off the frequency display.
3. Turning off the frequency display using **Display** > **Frequency** key does not erase the frequency display within the **Stimulus** softkey, which is turned on by pressing **Start**, **Stop**, **Center**, and **Span**. The display of the softkey bar itself can be switched on or off by pressing **Softkey On/Off**.

Hiding Softkey's Frequency Information

You can delete the frequency information from the measurement screen, which changes the frequency information displayed in the Stimulus softkey and the data entry area for Hz unit to asterisks (***).

1. Press **System** key.
2. Click **Service Menu**, then click **Security Level** and select any of the following options for the frequency display.

Softkey	Function
None	Displays the frequency information.
Low	Hides the frequency information with a series of asterisks. This can be turned to OFF by the Security Level menu.
High	Hides the frequency information with a series of asterisks. This cannot be turned to OFF by the Security Level menu. Resetting to OFF is only possible by executing Preset or Recall. o **Save/Recall** > **Save Trace Data** & **Save/Recall** > **Save SnP** is inactive when **Security Level** is set to **Low**/**High**.

Labeling a window

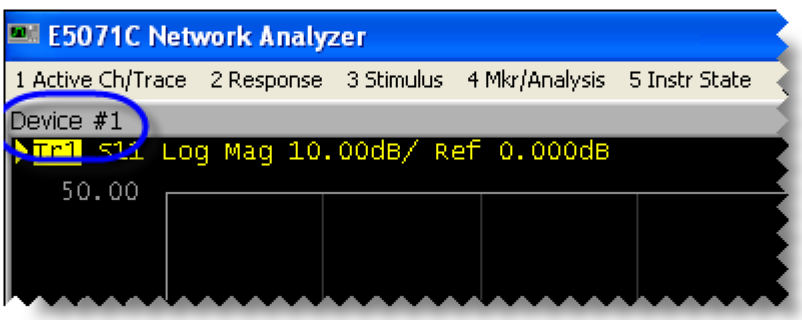
It is possible to assign a unique name to a channel and display it on the screen. This feature is useful in saving and/or printing measurement result for future reference.

Labeling a window

1. Press **Channel Next**/**Channel Prev** keys to select the channel to be labeled.
2. Press **Display** key.
3. Click **Edit Title Label**,then the title label input dialog box appears.

e5071c060

3. Using the keys in the dialog box, type a label and click **Enter**.
4. Click **Title Label** to turn on the title display. The title will appear within a frame at the top of the channel window.



e5071c195

Setting Display Colors

Selecting display mode

You can select the display mode of the LCD display from two modes: normal display (background: black) or inverted display (background: white). In normal display, the colors of items are preset so that you can recognize them easily on the display of the instrument. On the other hand, in inverted display, they are preset to colors obtained by nearly inverting the default settings of the normal display so that you can use data easily when storing it into a graphic file.

The selection procedure is as follows:

1. Press **Display**.
2. Click **Invert Color** to select the display color. OFF indicates the normal display; ON the inverted display.

Setting display color for each item

You can set the display color to the normal display or the inverted display separately for each of the following items.

- Data/memory trace
- Labels and lines of graphs
- File display of the limit test and limit lines
- Background

You set the color of each item by specifying the amounts of red (R), green (G), and blue (B) contained in the color. You can specify each level of R, G, and B in 6 steps (0 to 5). Therefore, 216 colors in total are available by combining them. The table below shows the R, G, and B values for the main colors as a reference.

	R	G	B		R	G	B		R	G	B
White	5	5	5	**Gray**	2	2	2	**Black**	0	0	0
Light red	5	3	3	**Red**	5	0	0	**Dark Red**	2	0	0
Light yellow	5	5	3	**Yellow**	5	5	0	**Dark Yellow**	2	2	0
Light green	3	5	3	**Green**	0	5	0	**Dark Green**	0	2	0
Light cyan	3	5	5	**Cyan**	0	5	5	**Dark cyan**	0	2	2
Light blue	3	3	5	**Blue**	0	0	5	**Dark Blue**	0	0	2
Light magent a	5	3	5	**Magent a**	5	0	5	**Dark Magent a**	2	0	2

The setting procedure is as follows:

1. Press **System** > **Misc Setup** > **Display Setup** > **Color Setup**
2. Click **Normal** (for normal display) or **Invert** (for inverted display).
3. Click the softkey corresponding to the item for which you want to set the display color.

Softkey	Function
Data Trace 1 to 16	Specifies the data trace of traces 1 to 16

Mem Trace 1 to 16	Specifies the memory trace of traces 1 to 16
Graticule Main	Specifies the graticule label and the outer lines of graphs
Graticule Sub	Specifies the grid of graphs
Limit Fail	Specifies the fail display in the limit test result
Limit Line	Specifies the limit line
Background	Specifies the background

4. Click **Red** (or, **Green**, or **Blue**).
5. Select the amount of the selected color from 0 to 5.

Resetting the display colors to the factory state

You can reset the display colors in normal display and inverted display to the preset factory state.

The selection procedure is as follows:

1. Press **System** > **Misc Setup** > **Display Setup** > **Color Setup.**
2. Click **Normal** (for normal display) or **Invert** (for inverted display).
3. Click **Reset Color** > **OK.**

Setting Display Magnification

You can reset the display magnification to Small, Normal or Large.

The selection procedure is as follows:

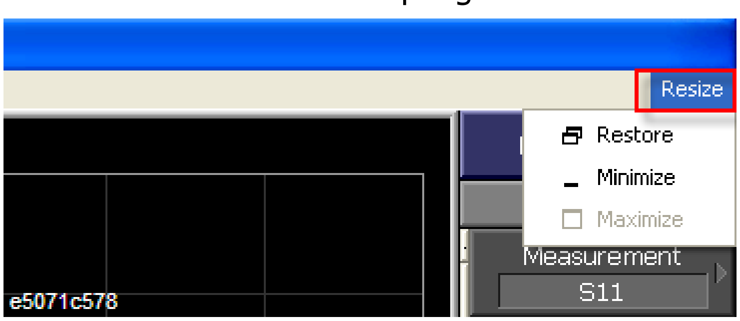
1. Press **System** > **Misc Setup** > **Display Setup** > **Magnification.**
2. Click **Normal, Small** or **Large**.

Resizing the Screen

You can resize the E5071C screen by minimizing, maximizing or restoring it to its original size.

The resizing procedure is as follows:

1. Click **Resize** at the top right corner of the screen.



2. A drop-down menu prompts and the available options are:

Softkey	Function
Restore	Restores the screen to its default size.
Minimize	Minimizes the screen.
Maximize	Displays the screen in full page size.

3. Click **Restore**,**Minimize** or **Maximize.**
4. When the screen is resized according to an option, its related softkey is disabled. For example, when the screen is displayed in full page size, **Maximize** is disabled.

Another option to minimize the E5071C screen is by using the Menu Bar and the procedure is as follows:

1. Press **Display**.
2. Click **Minimize E5071C.**

You can also hide and restore the title bar of the E5071C screen and the procedure is as follows:

1. Press **Display**.
2. Click **E5071C Title bar.**
3. Click **ON** to restore the title bar.
4. Click **OFF** to hide the title bar.

Calibration

Calibration

Overview

- Measurement Errors and their Characteristics
- Calibration Types and Characteristics
- Checking Calibration Status
- Clear Calibration

Basic Calibration

- Selecting Calibration Kit
- OPEN/SHORT Response Calibration (reflection test)
- THRU Response Calibration (transmission test)
- Enhanced Response Calibration
- 1-Port Calibration (reflection test)
- Full 2-Port Calibration
- Full 3-Port Calibration
- Full 4-Port Calibration

Calibration with ECal (Electronic Calibration)

- ECal (Electronic Calibration)
- ECal Driver Installation
- 1-Port Calibration Using a 2-Port ECal Module
- Calibration Using 4-port ECal

If you have 2 port Ecal:

- Full 2-Port Calibration Using the 2-Port ECal Module
- Full 3-Port and Full 4-Port Calibration using 2-Port ECal

Advanced Calibration with ECal

- Improving Calibration Accuracy along with ECal
- Confidence Check on Calibration Coefficients Using ECal
- Turning off ECal Auto-detect Function
- User-characterized ECal

Advanced Calibration

- Setting the Trigger Source for Calibration
- Sliding Offset and Load Calibration

- Modifying Calibration Kit Definition
- Specifying Different Standard for Each Frequency
- Receiver Calibration
- Power Calibration
- Partial Overwrite
- Simplified Calibration
- Adapter Removal-Insertion
- Unknown Thru Calibration
- Performing 8 term Calibration using External PC

Mixer Calibration

- Scalar-Mixer Calibration
- Vector-Mixer Calibration

TRL Calibration

- 2-port TRL calibration
- 3-port TRL calibration
- 4-port TRL calibration

Measurement Errors and their Characteristics

Other topics about Calibration

Overview

It is important to understand the factors contributing to measurement errors in order to determine the appropriate measures that should be taken to improve accuracy. Measurement errors are classified into three categories:

Drift Errors

Drift errors are caused by deviations in the performance of the measuring instrument (measurement system) that occur after calibration. Major causes are the thermal expansion of connecting cables and thermal drift of the frequency converter within the measuring instrument. These errors may be reduced by carrying out frequent calibrations as the ambient temperature changes or by maintaining a stable ambient temperature during the course of a measurement.

Random Errors

Random errors occur irregularly in the course of using the instrument. Since random errors are unpredictable, they cannot be eliminated by calibration. These errors are further classified into the following sub-categories depending on their causes.

- Instrument noise errors
- Switch repeatability errors
- Connector repeatability errors

Instrument noise errors

Instrument noise errors are caused by electric fluctuations within components used in the measuring instrument. These errors may be reduced by increasing the power of the signal supplied to the DUT, narrowing the IF bandwidth, or enabling sweep averaging.

Switch repeatability errors

Switch repeatability errors occur due to the fact that the electrical characteristics of the mechanical RF switch used in the measuring instrument change every time it is switched on. These errors may be reduced by carrying out measurements under conditions in which no switching operation takes place.

(You don't need to worry about these errors since the E5071C does not have mechanical RF switches).

Connector repeatability errors

Connector repeatability errors are caused by fluctuations in the electrical characteristics of connectors due to wear. These errors may be reduced by handling connectors with care.

Systematic Errors

Systematic errors are caused by imperfections in the measuring instrument and the test setup (cables, connectors, fixtures, etc.). Assuming that these errors are repeatable (i.e., predictable) and their characteristics do not change over time, it is possible to eliminate them mathematically at the time of measurement by determining the characteristics of these errors through calibration. There are six types of systematic errors, as follows.

Errors caused by signal leaks in the measuring system:

- Directivity
- Isolation (cross-talk)

Errors caused by reflections in the measuring system:

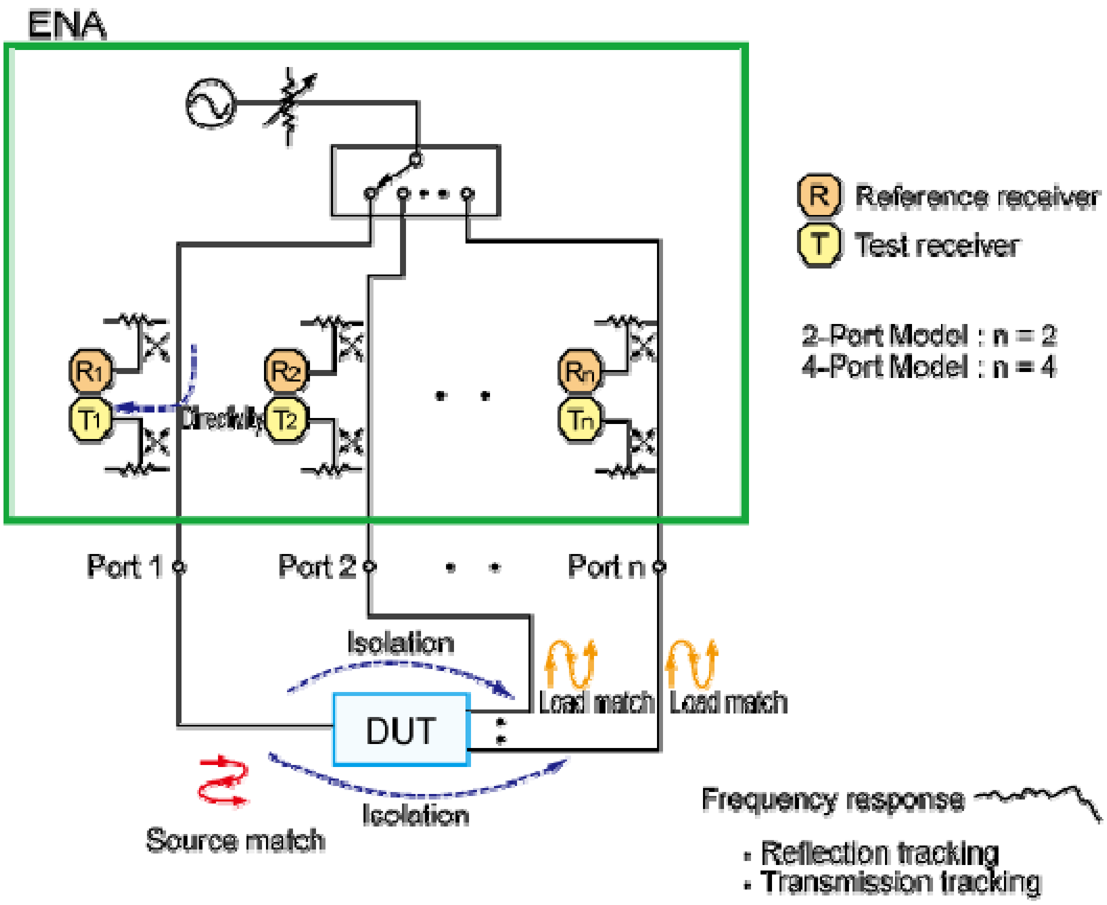
- Source match
- Load match

Errors caused by the frequency response of the receiver within the measuring instrument:

- Reflection tracking
- Transmission tracking

The E5071C has two receivers for each test port: the reference receiver and the test receiver (transmission measurement or reflection measurement). You can perform measurements with both of these receivers at the same time.

E5071C port architecture and systematic errors



e5071c240

Directivity error (Ed)

Directivity errors are caused by the fact that, in a reflection measurement, signals other than the reflection signal from the DUT are received by receiver T1 through the directivity coupler. When a certain port is a stimulus port, this error can be defined as a constant value for each stimulus port because the state of the termination at the other ports does not change. The number of directivity errors of the E5071C is the number of stimulus ports you use.

Ed1	Directivity error of port 1
Ed2	Directivity error of port 2
Ed3	Directivity error of port 3
Ed4	Directivity error of port 4

Isolation error (Ex)

An isolation error (crosstalk error) is caused by signals other than the transmission signal of the DUT leaking to the test receiver of the transmission measurement port in transmission measurements. When a certain port is a stimulus port, an isolation error is defined for each of the other ports. Therefore, the number of isolation errors for the E5071C is the total number of combinations of stimulus ports and response ports.

Ex21, Ex31, and Ex41	Isolation error when port 1 is a stimulus port
Ex12, Ex32, and Ex42	Isolation error when port 2 is a stimulus port
Ex13, Ex23, and Ex43	Isolation error when port 3 is a stimulus port
Ex14, Ex24, and Ex34	Isolation error when port 4 is a stimulus port

Source match error (Es)

A source match error is caused when the reflection signal of the DUT reflects at the signal source and enters the DUT again. When a certain port is a stimulus port, this error can be defined as a constant value for each stimulus port because the state of the signal source switch does not change. The number of source match errors in the E5071C is equivalent to the number of stimulus ports you use.

Es1	Source match error of port 1
Es2	Source match error of port 2
Es3	Source match error of port 3
Es4	Source match error of port 4

Load match error (El)

A load match error is caused when part, but not all, of the signal transmitted in the DUT reflects at a response port is measured by the receiver of the response port. When a certain port is a stimulus port, a load match error is defined for each of the other ports. Therefore, the number of load match errors for the E5071C is the total number of combinations of stimulus ports and response ports.

El21, El31, and El41	Load match error when port 1 is a stimulus port
El12, El32, and El42	Load match error when port 2 is a stimulus port
El13, El23, and El43	Load match error when port 3 is a stimulus port
El14, El24, and El34	Load match error when port 4 is a stimulus port

Reflection tracking error (Er)

A reflection tracking error is caused by the difference in frequency response between the test receiver and the reference receiver of a stimulus port in reflection measurements. This error can be defined as a constant value for each stimulus port because the combination of the test receiver and the reference receiver of a stimulus port is always the same. The number of reflection tracking errors for the E5071C is simply the number of stimulus ports you use.

Er1	Reflection tracking error of port 1
Er2	Reflection tracking error of port 2
Er3	Reflection tracking error of port 3
Er4	Reflection tracking error of port 4

Transmission tracking error (Et)

A transmission tracking error is caused by the difference in frequency response between the test receiver of a response port and the reference receiver of a stimulus port in transmission measurements. When a certain port is a stimulus port, a transmission tracking error is defined for each of the other ports. Therefore, the number of transmission tracking errors for the E5071C is the total number of combinations of stimulus ports and response ports.

Et21, Et31, and Et41	Transmission tracking error when port 1 is a stimulus port
Et12, Et32, and Et42	Transmission tracking error when port 2 is a stimulus port
Et13, Et23, and Et43	Transmission tracking error when port 3 is a stimulus port
Et14, Et24, and Et34	Transmission tracking error when port 4 is a stimulus port

Calibration Types and Characteristics

The table shows the different types of calibrations and the features of each method.

Calibration Method	Standard (s) Used	Corrected Error Factor	Measurement Parameters	Characteristics
No calibration	None	None	All parameters	• Low accuracy • Calibration not required
Response Calibration	OPEN or SHORT LOAD	Following 2 error terms: • Reflection Tracking (Er) • Directivity (Ed)	S11(Reflection characteristics at 1 port)	• Medium-level accuracy • Quick calibration • Isolation calibration improves the accuracy in a reflection measurement of a DUT with high return loss
	THRU LOAD	Following 2 error terms: • Transmission Tracking (Et) • Isolation (Ex)	S21 (1 direction transmission characteristics at 2 ports)	• Medium-level accuracy • Quick calibration • Isolation calibration improves the accuracy in a

				transmission measurement of a device with high insertion loss
1-Port Calibration	ECal module (2-port/4-port)	Following 3 error terms: • Directivity (Ed) • Source Match (Es) • Reflection Tracking (Er)	S11 (Reflection characteristics at 1 port)	• 1-port measurement with the highest degree of accuracy • Quick calibration with low chance of operator error
	OPEN SHORT LOAD			• Highly accurate 1-port measurement
Enhanced Response Calibration	Ecal module (2-port)	Following 5 error terms: • Directivity (Ed1) • Isolation (Ex21) • Source Match (Es1) • Transmission Tracking (Et21) • Reflection Tracking (Er1)	S11, S21 (1 direction transmission/Reflection characteristics at 2 ports)	• Highly accurate 2-port measurement (higher than response calibration) • Quick calibration with low chance of operator error

	OPEN SHORT LOAD THRU			• Highhly accurate 2-port measurement (higher than response calibration)
Full 2-Port Calibration	ECal module (2-port/4-port)	Following 12 error terms: • Directivity (Ed1,Ed2) • Isolation (Ex21,Ex12) • Source Match (Es1,Es2) • Load Match (El12,El21) • Transmission Tracking (Et21,Et12) • Reflection Tracking (Er1,Er2)	S11,S21,S12,S22 (All S-parameters at 2 ports)	• Highly accurate 2-port measurement • Quick calibration with low chance of operator error
	OPEN SHORT LOAD THRU			• Highly accurate 2-port measurement
Full 3-Port Calibration	ECal module (2-port/ 4-port)	Following 27 error terms: • Directivity (Ed1,Ed2,Ed3) • Isolation (Ex21,Ex31,Ex12,Ex32,Ex13,Ex23) • Source Match (Es1,Es2,Es3) • Load Match (El21,El31,El12,El32, El13,El23) • Transmission Tracking (Et21,Et31,Et12,Et32, Et13,Et23) • Reflection Tracking (Er1,Er2,Er3)	S11,S21,S31,S12,S22,S32, S13,S23,S33 (All S-parameters at 3 ports)	• Highly accurate 3-port measurement • Quick calibration with low chance of operator error
	OPEN SHORT LOAD THRU			• Highly accurate 3-port measurement

Simplified full 3-port calibration	OPEN SHORT LOAD THRU	Same as full 3-port calibration	S11, S21, S31, S12, S22, [S32], S13, [S23], S33 (Part of thru measurement can be skipped)	• High-accuracy 3-port measurement • Simpler procedure by skipping thru measurement
Full 4-Port Calibration	ECal module (2-port/ 4-port)	Following 48 error terms: • Directivity (Ed1,Ed2,Ed3,Ed4) • Isolation (Ex21,Ex31,Ex41,Ex12,Ex32,Ex42 ,Ex13,Ex23,Ex43,Ex14,Ex24,Ex34) • Source Match (Es1,Es2,Es3,Es4) • Load Match (El1,El2,El3,El4) • Transmission Tracking (Et21,Et31,Et41,Et12, Et32, Et42,Et13,Et23,Et43,Et14 ,Et24,Et34) • Reflection Tracking (Er1,Er2,Er3,Er4)	S11,S21,S31,S41,S12,S22, S32,S42, S13,S23,S33,S43,S14,S24, S34,S44 (All S-parameters at 4 ports)	• Highly accurate 4-port measurement • Quick calibration with low chance of operator error
	OPEN SHORT LOAD THRU			• Highly accurate 4-port measurement
Simplified full 4-port calibration	Open Short Load Thru	Same as full 4-port calibration	S11, S21, S31, [S41], S12, S22, [S32], [S42], S13, [S23], S33, S43, [S14], [S24], S34, S44 (Part of thru measurement can be skipped)	• High-accuracy 4-port measurement • Simpler procedure by skipping thru measurement
2-Port	Reflectio	Following 12 error	S11,S21,S12,S22	• Highly

TRL Calibration	n (OPEN or SHORT) THRU LINE MATCH	terms: • Directivity (Ed1,Ed2) • Source Match (Es1,Es2) • Load Match (El1,El2) • Transmission Tracking (Et21,Et12) • Reflection Tracking (Er1,Er2)	(All S-parameters at 2 ports)	accurate 2-port measurement • Effective for non-coaxial device measurement
3-Port TRL Calibration	Reflection (OPEN or SHORT) THRU LINE MATCH	Following 27 error terms: • Directivity (Ed1,Ed2,Ed3) • Source Match (Es1,Es2,Es3) • Load Match (El21,El31,El12,El32,El13,El23) • Transmission Tracking (Et21,Et31,Et12,Et32,Et13,Et23) • Reflection Tracking (Er1,Er2,Er3)	S11,S21,S31,S12,S22,S32,S13,S23,S33 (All S-parameters at 3 ports)	• Highly accurate 3-port measurement • Effective for non-coaxial device measurement
Simplified 3-port TRL calibration	Reflection (open or short) Thru Line Match	Same as 3-port TRL calibration	S11, S21, S31, S12, S22, [S32], S13, [S23], S33 (Part of thru (or line) and line (or match) measurement can be skipped)	• High-accuracy 3-port measurement Effective for non-coaxial device measurement • Simpler procedure by skipping thru/line/match measurement
4-Port TRL Calibra	Reflection (OPEN or	Following 48 error terms: • Directivity	S11,S21,S31,S41,S12,S22,S32,S42,S13, S23,S33,S43,S14,S24,S34,	Highly accurate 4-port

tion	SHORT) THRU LINE MATCH	(Ed1,Ed2,Ed3,Ed4) • Source Match (Es1,Es2,Es3,Es4) • Load Match (El1,El2,El3,El4) • Transmission Tracking (Et21,Et31,Et41,Et12, Et32,Et42, Et13,Et23,Et43,Et14,Et24,Et34) • Reflection Tracking (Er1,Er2,Er3,Er4)	S44 (All S-parameters at 4 ports)	measurement Effective for non-coaxial device measurement
Simplified 4-port TRL calibration	Reflection (open or short) Thru Line Match	Same as 4-port TRL calibration	S11, S21, S31, [S41], S12, S22, [S32], [S42], S13, [S23], S33, S43, [S14], [S24], S34, S44 (Part of thru (or line) and line (or match) measurement can be skipped)	• High-accuracy 4-port measurement Effective for non-coaxial device measurement • Simpler procedure by skipping thru/line/match measurement

Other topics about Calibration

Checking Calibration Status

- Execution Status of Error Correction for Each Channel
- Execution Status of Error Correction for Each Trace
- Acquisition Status of Calibration Coefficient for Each Channel

Other topics about Calibration

Execution Status of Error Correction for Each Channel

You can check the execution status of error correction for each channel with the error correction status.

The error correction status is indicated in the channel status bar in the lower part of the window by the symbols in the below table.

Symbol	Execution status of error correction
Cor (displayed in blue)	Error correction: On (enabled for all traces)
Cor (displayed in gray)	Error correction: On (enabled for some traces)
Off (displayed in gray)	Error correction: Off
--- (displayed in gray)	Error correction: On (no calibration data)
C? (displayed in blue)	Error correction: On (Interpolation is being executed or the IF bandwidth, power level, power range, sweep time, sweep delay time, sweep mode, or sweep type is different from that when the calibration was executed.)
C! (displayed in blue)	Error correction: On (Extrapolation is being executed.) In the option TDR, even after making deskew and calibration, this is displayed.

An asterisk (*) in the upper-right of the softkey indicates that the existing calibration coefficient will be cleared if you select the test port and execute acquisition of the calibration coefficient (clicking **Done**).

Execution Status of Error Correction for Each Trace

You can check the status of the error correction actually executed for each trace with the trace status area.

For a trace for which error correction is executed, the applied calibration type is indicated in the trace status area by the symbols in the table below.

Symbol	Calibration type
RO	Open response calibration
RS	Short response calibration
RT	Thru response calibration
ER	Enhanced response calibration
F1	1-port calibration
F2	Full 2-port calibration/2-port TRL calibration
F3	Full 3-port calibration/3-port TRL calibration
F4	Full 4-port calibration/4-port TRL calibration

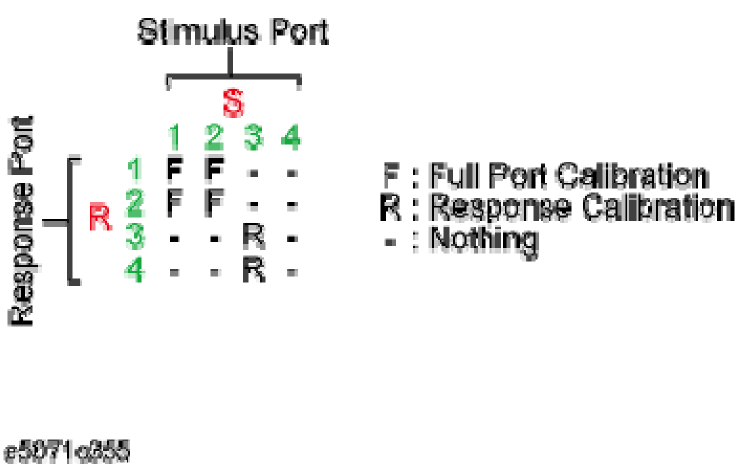
If none of the symbols described above is displayed, error correction is not executed for the trace.

Acquisition Status of Calibration Coefficient for Each Channel

You can check the acquisition status of the calibration coefficient for each channel with the calibration property.

The calibration property displays the acquisition status of the calibration coefficient between test ports for each channel in matrix format. The following figure shows an example of when the calibration coefficients have been acquired for the full 2-port calibration between test ports 1 and 2, the response calibration for test port 3, and the response calibration (THRU) between test ports 4 and 3.

Example of calibration property display



NOTE The simplified full 3/4-port calibration and the simplified 3/4-port TRL calibration are not discriminated from the normal full-port and TRL calibrations, and **F** is displayed.

Procedure to turn on/off calibration property display

Follow these steps to turn on/off the calibration property display.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to turn on/off the calibration property display.
2. Press **Cal** key.
3. Click **Property**. Each press toggles the on/off setting.

Conditions for clearing already acquired calibration coefficients

In the following cases, already acquired calibration coefficients are cleared.

- Executing preset clears all calibration coefficients.
- If S parameters required to calculate the calibration coefficient for the specified calibration type and test ports and those required for the existing calibration coefficient overlap, executing the acquisition of the calibration coefficient (measuring necessary data and then clicking the **Done** softkey) clears the calibration coefficient for which necessary S parameters overlap. Taking the Example of calibration property display as an example, if you acquire the calibration coefficient of the 1-port calibration for test port 4, neither calibration coefficient is cleared. On the other hand, if you acquire the calibration coefficient for the full 2-port calibration between test ports 2 and 3, the calibration coefficient of the full 2-port calibration between test ports 1 and 2 and that of the response calibration for test port 3 are cleared.

Clear Calibration

This softkey clears the user calibration data. When Calibration is done for a particular DUT, the data get stored in the E5071C. To clear this data, **Clear Calibration** can be used which removes the User calibration data from the E5071C.

Other topics about Calibration

Basic Calibrations

Selecting Calibration Kit

- Overview
- Procedure

Other topics about Basic Calibration

Overview

Before executing calibration, you need to select a calibration kit.

If you use a calibration kit other than a predefined one, you need to define it. If the connector type of the standard of the calibration kit you use has polarity (the distinction between male and female), you need to change the standard class definition of the calibration kit depending on the standard you actually use. For more information, see Modifying Calibration Kit Definition.

NOTE If you select a predefined calibration kit, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to select the calibration kit.
2. Press **Cal** Key.
3. Click **Cal Kit**, then select the calibration kit.
4. If the name (label) of the calibration kit has been changed, the label is displayed as the softkey.
5. An asterisk (*) on the upper right of the softkey corresponding to a predefined calibration kit indicates that its definition value has been changed from the factory setting by the user.

OPEN/SHORT Response Calibration (reflection test)

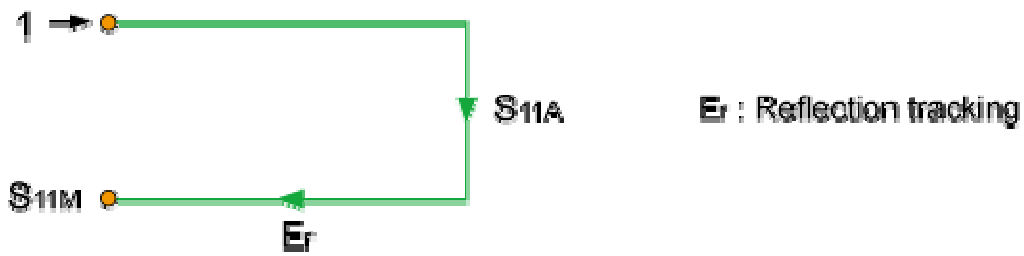
- Overview
- Procedure

Other topics about Basic Calibration

Overview

In OPEN or SHORT response calibration, calibration data are measured by connecting an OPEN or SHORT standard, respectively, to the desired test port. For frequency response, these calibrations effectively eliminate the reflection tracking error from the test setup reflection test using that port.

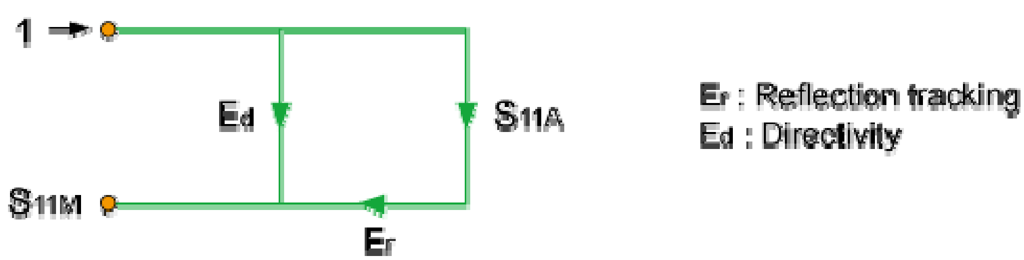
1-Port error model (OPEN/SHORT response)



e5071c398

It is also possible to carry out isolation calibration with a LOAD standard during OPEN/SHORT response calibration. An isolation calibration will eliminate the directivity error from the test setup in a reflection test using that port

1-Port error model (OPEN/SHORT response + isolation)



e5071c399

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

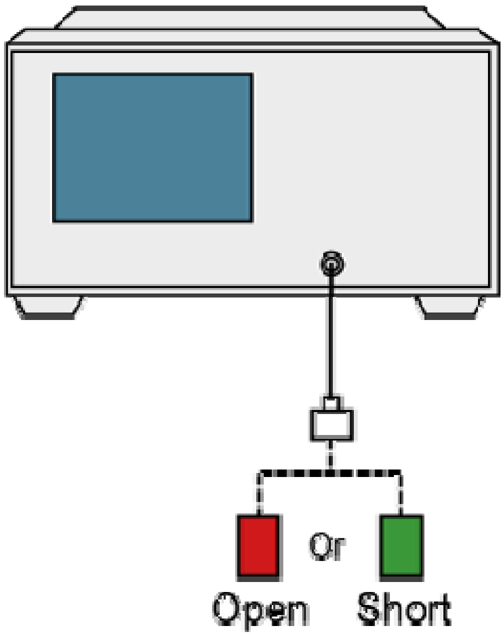
Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate.**
4. Select **OPEN or SHORT response calibration**.

Softkey	Function
Response (Open)	Displays softkeys for performing an OPEN response calibration (response calibration with an OPEN standard)
Response (Short)	Displays softkeys for performing a SHORT response calibration (response calibration with a SHORT standard)

5.
6. Click **Select Port.**
7. Select the test port upon which you will perform OPEN/SHORT response calibration.
8. Connect a calibration standard (OPEN or SHORT) to the selected test port (connector to which the DUT is to be connected).
9. Click **Open** or **Short** to start the calibration measurement.
10. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.
11. Connect a LOAD standard to the selected test port (connector to which the DUT is to be connected).
12. Click **Load (Optional)** to start the measurement on the LOAD standard.
13. Click **Done** to terminate the response calibration (and the LOAD isolation calibration) process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Connecting standards in OPEN/SHORT Response calibration



e5071c318

THRU Response Calibration (transmission test)

- Overview
- Procedure

Other topics about Basic Calibration

Overview

In THRU response calibration, calibration data are measured by connecting a THRU standard to the desired test port. This calibration effectively eliminates the frequency response transmission tracking error from the test setup in a transmission test using that port.

2-Port error model (THRU response)

e5071c400

It is also possible to carry out an isolation calibration using a LOAD standard in the process of THRU response calibration. An isolation calibration will eliminate isolation error (crosstalk error) from the test setup in a transmission test using that port.

2-Port Error model (THRU response + isolation)

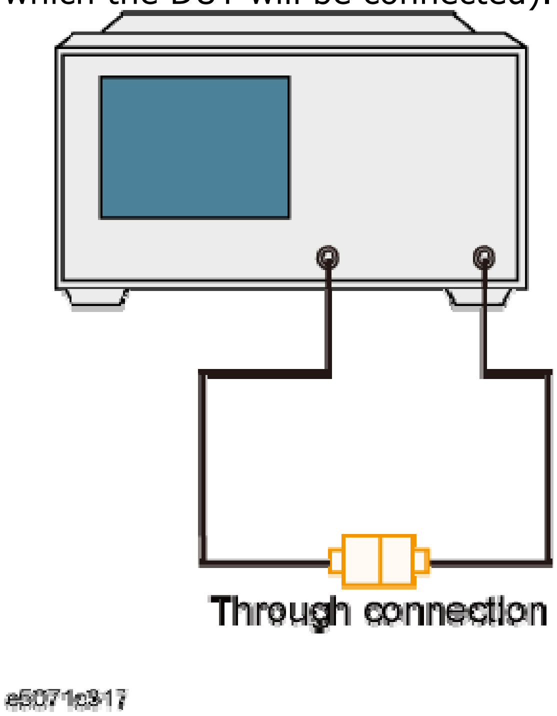
e5071c401

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **Response (Thru)** > **Select Ports**.

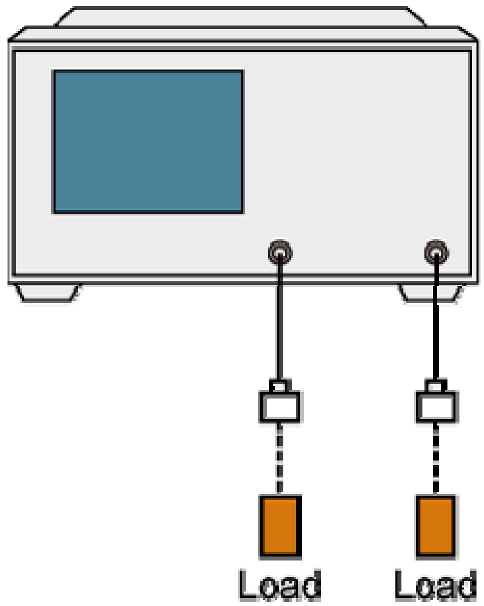
4. Select the test ports (and corresponding S parameters) upon which a THRU response calibration is to be performed.
5. Make a between the selected test ports (between the connectors to which the DUT will be connected).



6. Click **Thru** to start the calibration measurement.
    - The sweep is performed twice in the one thru measurement.
7. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.

6.

    a. Connect a LOAD standard to each of the two selected test ports (connectors to which the DUT is to be connected).



e5071c319

   b. Click **Isolation (Optional)** to start the calibration measurement.

7. Click **Done** to terminate the response calibration (and the LOAD isolation calibration) process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Enhanced Response Calibration

- Overview
- Procedure

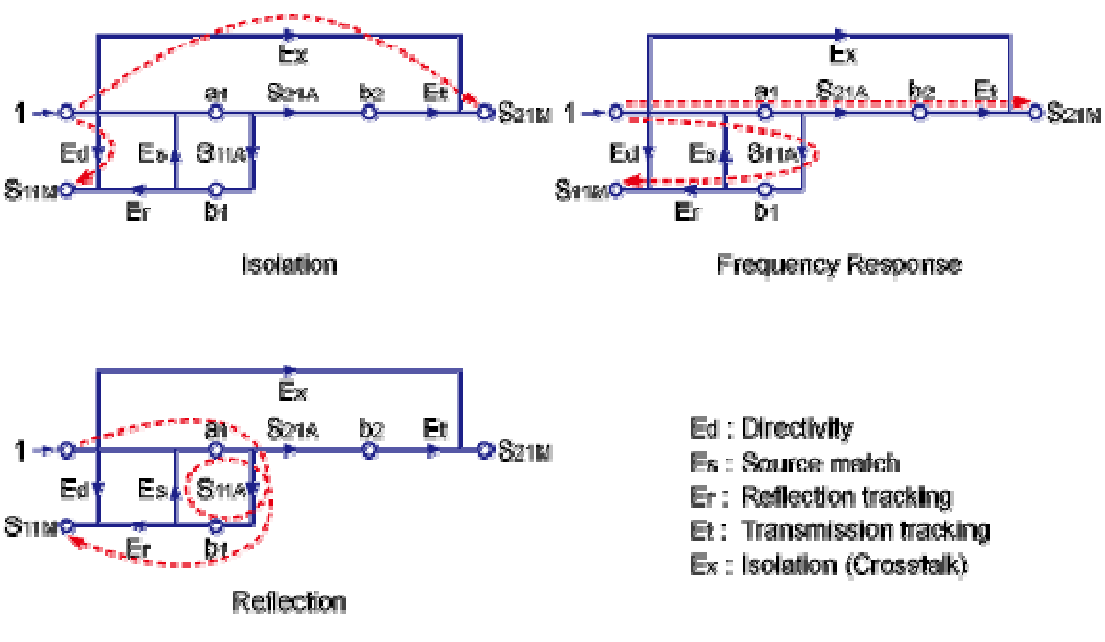
Other topics about Basic Calibration

Overview

In enhanced response calibration, calibration data are measured by connecting an OPEN standard, a SHORT standard, or a LOAD standard to the output port (or a THRU standard between two ports).This calibration effectively eliminates the directivity error, crosstalk, source match error, frequency response reflection tracking error, and frequency response transmission tracking error from the test setup in a transmission or reflection test using those ports .

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

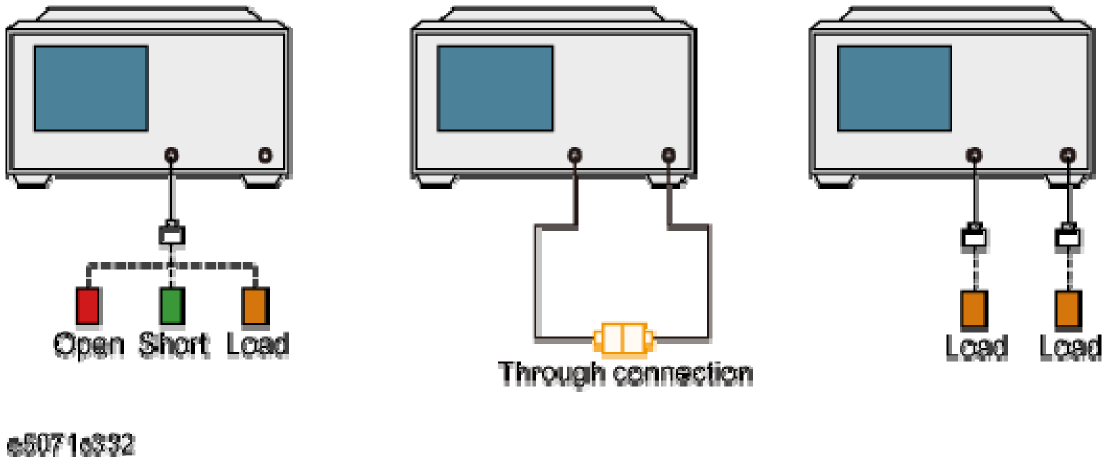
2-Port Error Model (Enhanced Response)



e5071c301

Procedure

Connecting the Standard at Enhanced Response Calibration



1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate.**
4. Click **Enhanced Response.**
5. Click **Ports** to select the test ports on which an enhanced response calibration will be performed.
6. Connect an OPEN calibration standard to the output port.
7. Click **Open** to start the calibration measurement.
   - If you select the calibration kit which has different calibration definitions for each gender, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.
8. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.
9. Click **Short** to start the calibration measurement.
10. Disconnect the SHORT calibration standard and replace it with a LOAD standard.
11. Click **Load** to start the calibration measurement.
12. Make a THRU connection between the two ports.
13. Click **Thru** to start the calibration measurement.
14. If an isolation calibration must be performed using a LOAD standard, follow the procedure below:

13.

   a. Connect a LOAD standard to the two test ports.

b. Click **Isolation (Optional)** to start the calibration measurement.

c. Click **Return**.

14. Click **Done** to terminate the enhanced response calibration process. Upon pressing the key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

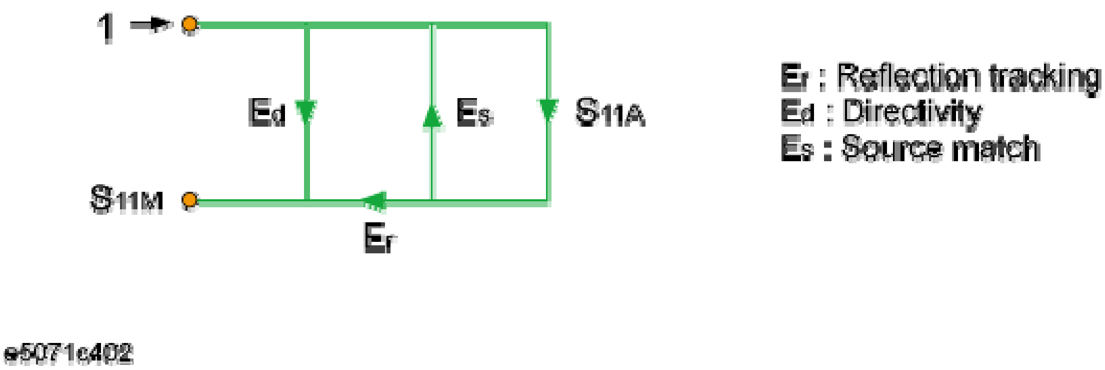
1-Port Calibration (reflection test)

- Overview
- Procedure

Other topics about Basic Calibration

Overview

In 1-port calibration, calibration data are measured by connecting an OPEN standard, a SHORT standard, and a LOAD standard to the desired test port. This calibration effectively eliminates the frequency response reflection tracking error, directivity error, and source match error from the test setup in a reflection test using that port.



1-Port error model (1-port calibration)

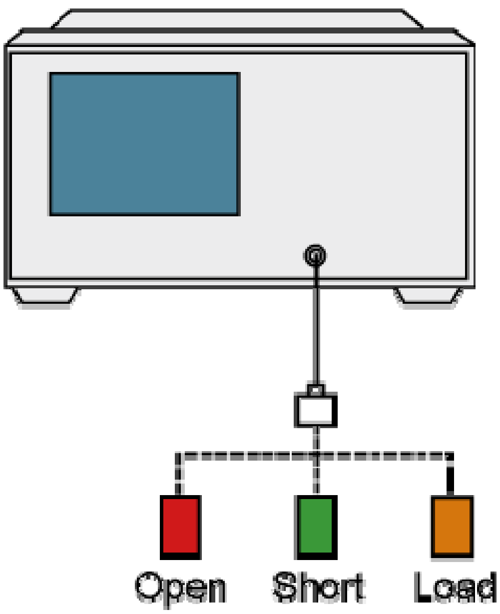
NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **1-Port Cal** > **Select Port**.
4. Select a test port (and corresponding S parameter) on which 1-port calibration will be performed.
5. Connect an OPEN calibration standard to the selected test port (connector to which the DUT is to be connected).
6. Click **Open** to start the calibration measurement.
   - If you select the calibration kit which has different calibration definitions for each gender, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.

7. Connect a SHORT calibration standard to the selected test port (connector to which the DUT is to be connected).
8. Click **Short** to start the calibration measurement.
9. Connect a LOAD calibration standard to the selected test port (connector to which the DUT is to be connected).
10. Click **Load** to start the calibration measurement.
11. Click **Done** to terminate the 1-port calibration process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Connecting the standard for 1-port calibration



e5071c300

Full 2-Port Calibration

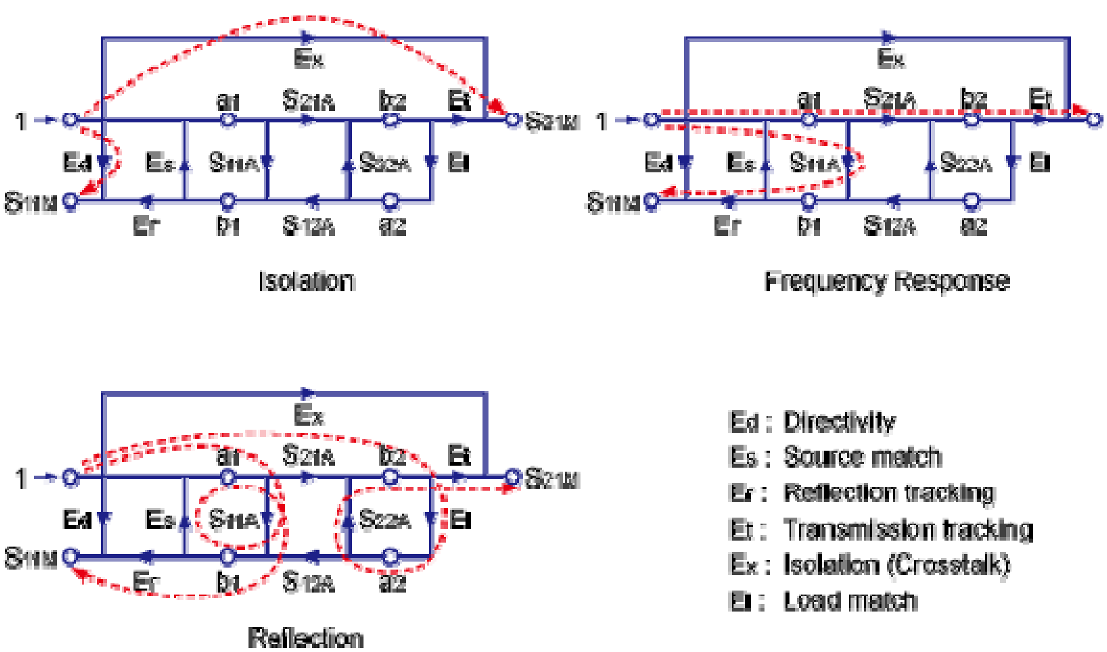
- Overview
- Procedure

Other topics about Basic Calibration

Overview

In full 2-port calibration, calibration data are measured by connecting an OPEN standard, a SHORT standard, or a LOAD standard to two desired test ports (or a THRU standard between two ports).This calibration effectively eliminates the directivity error, crosstalk, source match error, frequency response reflection tracking error, and frequency response transmission tracking error from the test setup in a transmission or reflection test using those ports. This calibration makes it possible to perform measurements with the highest possible accuracy. A total of twelve error terms, six each in the forward direction and the reverse direction, are used in a calibration.

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.



e5071c303

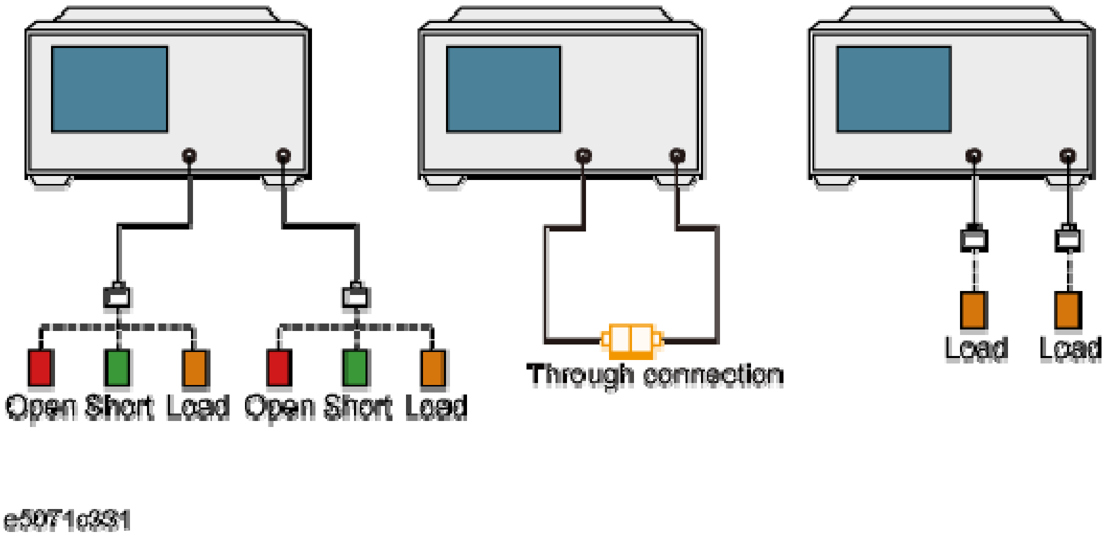
Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.

2. Press **Cal** key.
3. Click **Calibrate** > **2-Port Cal**.
4. Click **Select Ports**, then select the test ports on which you will perform full 2-port calibration. (In the procedure below, the selected test ports are denoted as x and y.)
5. Click **Reflection**.
6. Connect an OPEN calibration standard to test port x (the connector to which the DUT is to be connected).
7. Click **Port x Open** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
   - If you select the calibration kit which has different calibration definitions for each gender, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.
8. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.
9. Click **Port x Short** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
10. Disconnect the SHORT calibration standard and replace it with a LOAD standard.
11. Click **Port x Load** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
12. Repeat the above procedure for port y.
13. Click **Return**.
14. Click **Transmission**.
15. Make a THRU connection between ports x and y (between the connectors to which the DUT is to be connected).
16. Click **Port x-y Thru** to start the calibration measurement (**x** and **y** denote the test ports between which the THRU connection is being made).
17. Click **Return**.
18. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.
19. Click **Isolation (Optional)**.
20. Connect a LOAD standard to each of the two test ports (connectors to which the DUT is to be connected).

21. Click **Port x-y Isol** to start the calibration measurement (**x** and **y** denote the port numbers to which the LOAD standard is connected).
22. Click **Return**.
23. Click **Done** to terminate the full 2-port calibration process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Connecting standards in full 2-port calibration



Full 3-Port Calibration

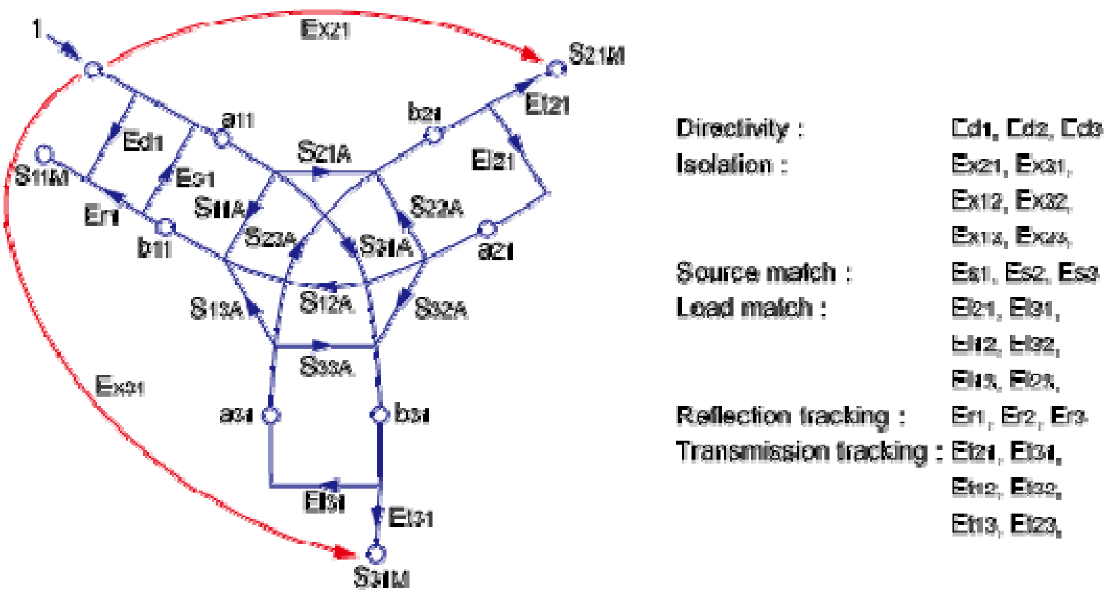
- Overview
- Procedure

Other topics about Basic Calibration

Overview

In full 3-port calibration, calibration data are measured by connecting an OPEN standard, a SHORT standard, or a LOAD standard to three desired test ports (or a THRU standard between three ports). This calibration effectively eliminates the directivity error, crosstalk, source match error, load match error, frequency response reflection tracking error, and frequency response transmission tracking error from the test setup in a transmission or reflection test using those ports. As in full 2-port calibration, this calibration method also makes it possible to perform measurements with the highest possible accuracy. There are unique error terms for directivity, source match, and reflection tracking for each stimulus test port (3 × 3 ports = 9). As for isolation, load match, and transmission tracking errors, there are unique terms for each combination between a stimulus port and a response port (3 × 6 combinations = 18). Therefore, in total, 27 error terms are involved in a full 3-port calibration.

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.



e5071c304

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **3-Port Cal**.
4. Click **Select Ports**, then select the test ports on which you will perform full 3-port calibration.

Reflection

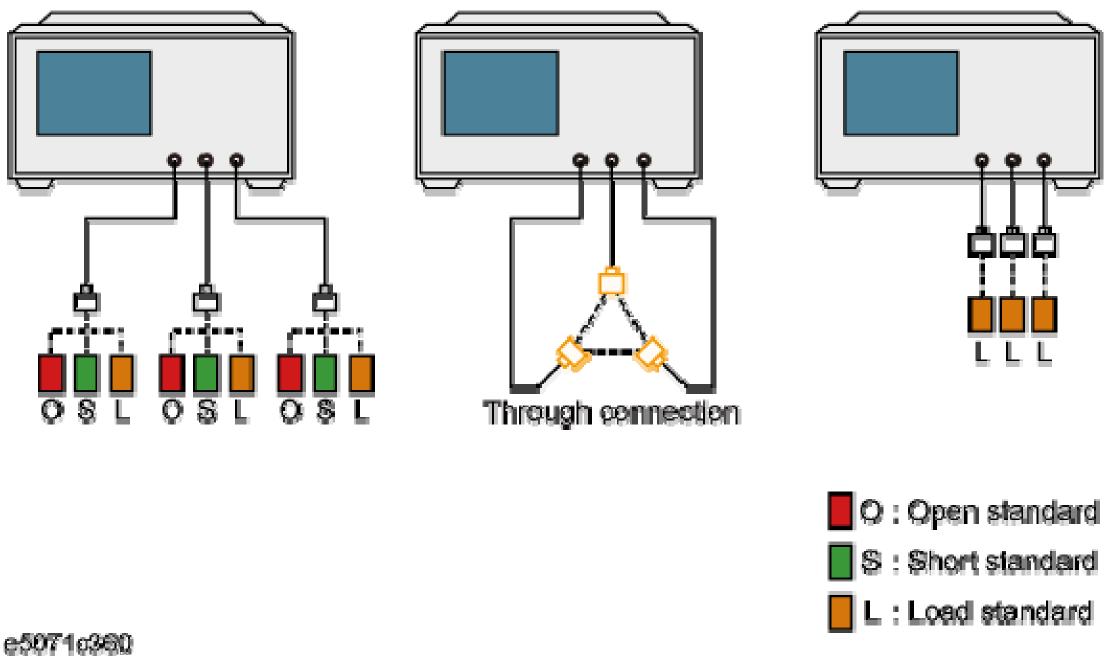
1. Click **Reflection**.
2. Connect an OPEN calibration standard to test port x (the connector to which the DUT is to be connected).
3. Click **Port x Open** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
    - If you select the calibration kit which has different calibration definitions for each gender, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.
4. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.
5. Click **Port x Short** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
6. Disconnect the SHORT calibration standard and replace it with a LOAD standard.
7. Click **Port x Load** to start the calibration measurement (**x** denotes the test port to which the standard is connected).
8. Repeat the above procedure for ports y and z.
9. Click **Return**.

Transmission

1. Click **Transmission**.
2. Make a THRU connection between ports x and y (between the connectors to which the DUT is to be connected).
3. Click **Port x-y Thru** to start the calibration measurement (**x** and **y** denote the test ports between which the THRU connection is being made).
4. Repeat above procedure for ports x and z. then for ports y and z.

5. Click **Return**.
6. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.
7. Click **Isolation (Optional)**.
8. Connect a LOAD standard to each of the three test ports (connectors to which the DUT is to be connected).
9. Click **Port x-y Isol** to start the calibration measurement (**x** and **y** denote the port numbers to which the LOAD standard is connected).
10. Click **Return**.
11. Click **Done** to terminate the full 3-port calibration process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Connecting standards in full 3-port calibration



Full 4-Port Calibration

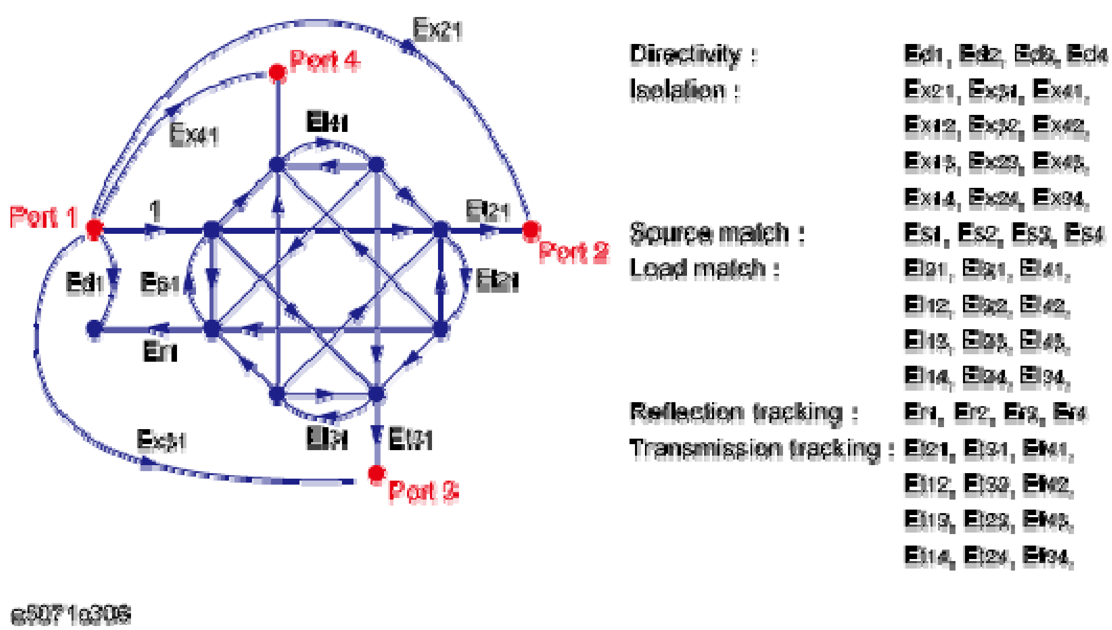
- [Overview](#)
- [Procedure](#)

Other topics about Basic Calibration

Overview

In full 4-port calibration, calibration data are measured by connecting an OPEN standard, a SHORT standard, or a LOAD standard to the four test ports (or a THRU standard between the four ports). This calibration effectively eliminates the directivity error, crosstalk, source match error, load match error, frequency response reflection tracking error, and frequency response transmission tracking error from the test setup in a transmission or reflection test using those ports. As in full 2-port calibration, this calibration method also makes it possible to perform measurements with the highest possible accuracy. There are unique error terms for directivity, source match, and reflection tracking for each stimulus test port (3 × 4 ports = 12). As for isolation, load match, and transmission tracking errors, there are unique terms for each combination between a stimulus port and a response port (3 × 12 combinations = 36). Therefore, in total, 48 error terms are involved in a full 4-port calibration.

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.



Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **4-Port Cal**.

Reflection

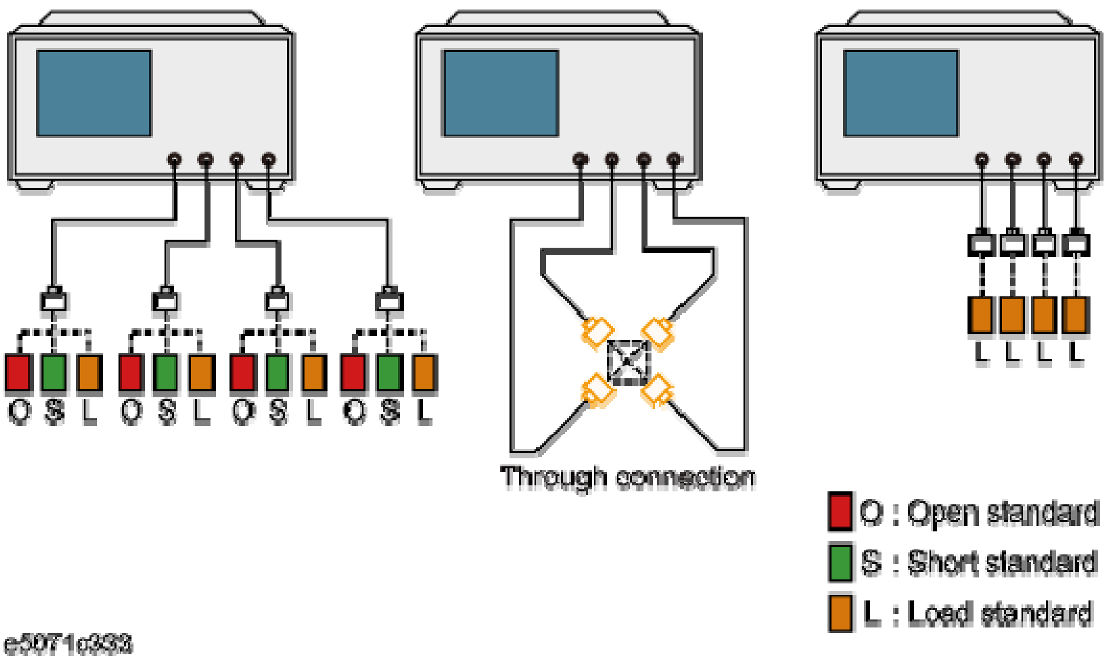
1. Click **Reflection**.
2. Connect an OPEN calibration standard to test port 1.
3. Click **Port 1 Open** to start the calibration measurement.
   - If you select the calibration kit which has different calibration definitions for each gender, (m) and (f) in the name (label) of the standard displayed in the softkey indicate male (m) and female (f) for the analyzer's connector, respectively.
4. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.
5. Click **Port 1 Short** to start the calibration measurement.
6. Disconnect the SHORT calibration standard and replace it with a LOAD standard.
7. Click **Port 1 Load** to start the calibration measurement.
8. Repeat the above procedure for ports 2, 3 and 4.
9. Click **Return**.

Transmission

1. Click **Transmission**.
2. Make a THRU connection between ports 1 and 2.
3. Click **Port 1-2 Thru** to start the calibration measurement.
4. Repeat above procedure ports 1 and 3.
5. Repeat the procedure on ports 1 and 4.
6. Repeat the procedure on ports 2 and 3.
7. Repeat the procedure on ports 2 and 4.
8. Repeat the procedure on ports 3 and 4.
9. Click **Return**.
10. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.

11. Click **Isolation (Optional)**.
12. Connect a LOAD standard to each of the four test ports (connectors to which the DUT is to be connected).
13. Click **Port 1-2 Isol** to start the calibration measurement.
14. Perform the each calibrations by clicking **Port 1-3 Isol**, **Port 1-4 Isol**, **Port 2-3 Isol** and **Port 2-4 Isol**.
15. Click **Return**.
16. Click **Done** to terminate the full 4-port calibration process. Upon pressing this key, calibration coefficients will be calculated and saved. The error correction function will also be automatically enabled.

Connecting standards in full 4-port calibration



Calibration with ECal (Electronic Calibration)

ECal (electronic calibration)

ECal is a calibration method that uses solid-state circuit technology. ECal offers the following advantages:

- Simplified calibration process.
- Shorter time required for calibration.
- Reduced chance of erroneous operation.
- Little degradation of performance due to wear because the ECal module employs PIN diodes and FET switches.
  - If the frequency sweep range exceeds the frequency range of the ECal, the calibration data for the minimum frequency or maximum frequency are used for the exceeding frequency range and extrapolation is being executed.
  - When using Ecal with the E5071C options 2D5, 4D5, 2K5 and 4K5 , wait time is inserted automatically for stable calibration results. Approximate calibration time is described in the support ECal page.

Refer the following section for ECal calibration.

- Connecting ECal to E5071C
- 1-Port Calibration Using a 2-Port ECal Module
- Calibration Using 4-port ECal
- Full 2-Port Calibration Using the 2-Port ECal Module
- Full 3-Port and Full 4-Port Calibration using 2-Port ECal
- Improving Calibration Accuracy along with ECal
- Confidence Check on Calibration Coefficients Using ECal
- Turning off ECal auto-detect function
- User-characterized ECal

Other topics about Calibration with ECal

Connecting ECal to E5071C

- ECal Driver Installation
- Connecting Single ECal
- Connecting Multiple ECal

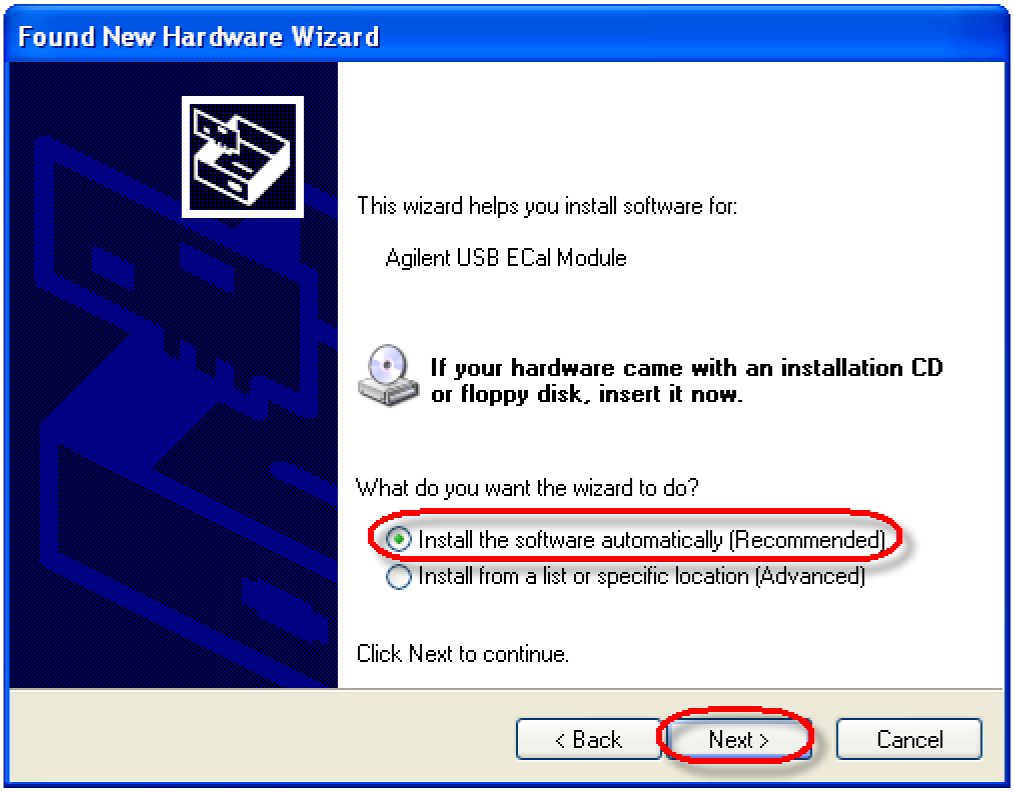
Other topics about Calibration with ECal

ECal Driver Installation

When the ECal is connected with USB ports at the first time, ECal driver installation is required.

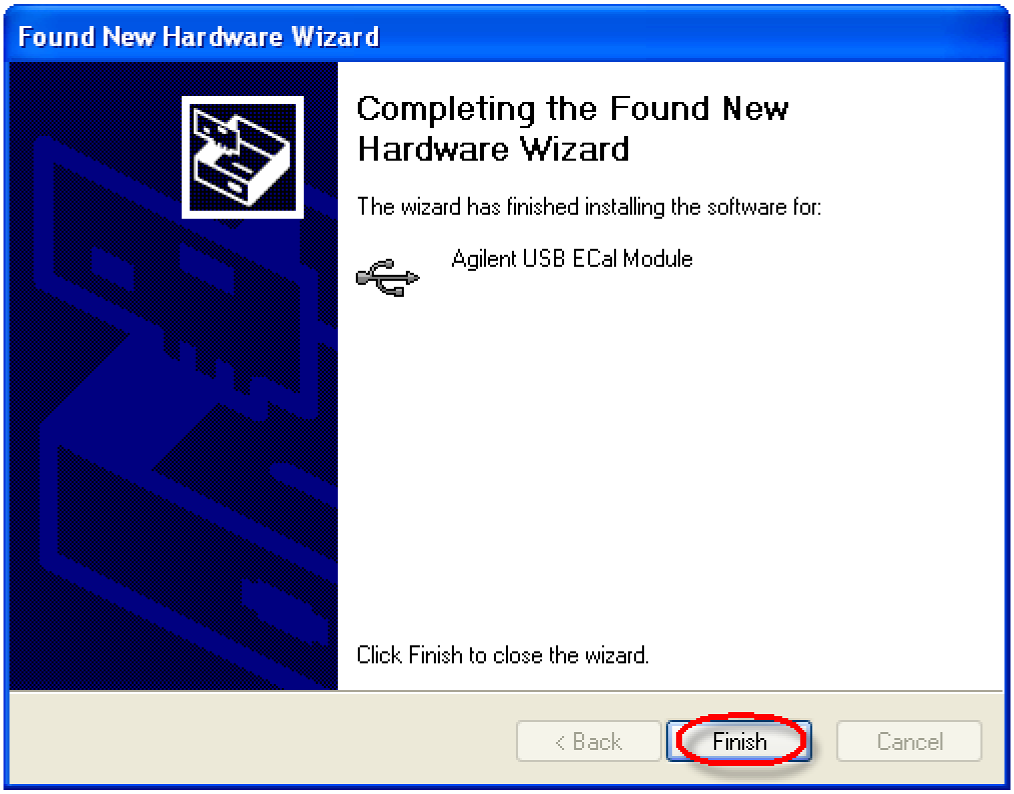
1. Connect a Ecal to the USB port of the E5071C.
2. Select **No, not this time**, then click **Next.**

3. Select **Install the software automatically (Recommended),** then click **Next.**



e5071c230

4. Click **Finish.**



e5071c231

5. Even if you install the driver on a USB port, you will be asked to install the driver again if you connect the ECal with a different USB port.

Connecting Single ECal

There are several options available in calibrating using ECal module:

- 1-Port Calibration Using a 2-Port ECal Module
- Calibration Using 4-Port ECal
- Full 2-Port Calibration Using the 2-Port ECal Module
- Full 3-Port and Full 4-Port Calibration using 2-Port ECal

Connecting Multiple ECal

The ECal module connected to E5071C is listed here: **Cal** > **ECal** > **ECal**.

If a single ECal is connected, only one ECal is listed here. When multiple ECal are connected E5071C, all the connected ECals are listed here. The first line indicates the model number of the ECal and the second line indicates its serial number, as shown below:

E5071C revision A.9.60 and above allows multiple ECal to be connected at any one time. To perform calibration using your preferred ECal module, select the desired ECal from the list and perform 1-Port, 2-Port, 3-Port or 4-Port calibration just as its performed in single ECal. Once completed, you can select another ECal connected to E5071C and repeat the procedure.

Other topics about Calibration with ECal

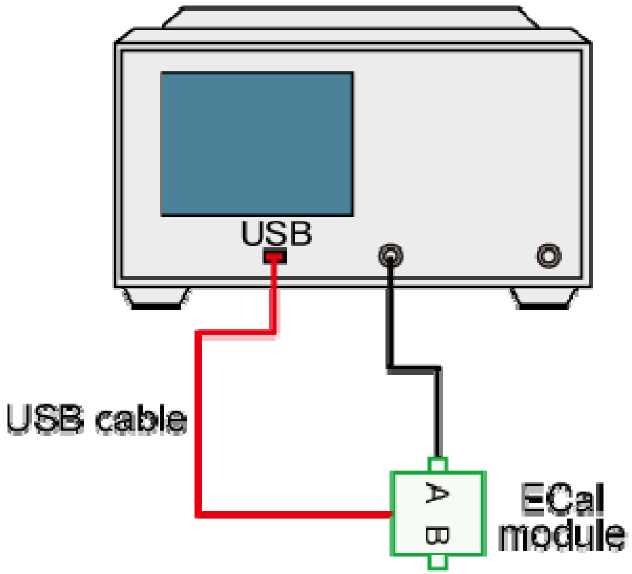
1-Port Calibration Using a 2-Port ECal Module

Follow the procedure below to perform a 1-port calibration using the 2-port ECal module.

1. Connect the USB port on the ECal module with the USB port on the E5071C via a USB cable. This connection may be made while the E5071C's power is on.
2. Allows the ECal module to warm up for 15 minutes until the module indicates from WAIT to READY.
3. Connect a port on the ECal module to the test port to be calibrated.
4. If you don't use all of the ECal module's ports, connect terminations to the unused ports.
5. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
6. Press **Cal** key.
7. Click **Ecal** > **1-Port Cal**.
8. Perform a 1-port calibration by clicking the softkey of selected port.

NOTE You can connect the ports of the ECal and the test ports of the E5071C arbitrarily. Connected ports can be manually specified although they are automatically detected before data measurement. For more information, Turning off ECal auto-detect function.

Connecting ECal module (1-port calibration)



e5071c325

Other topics about Calibration with ECal

Calibration Using 4-port ECal

- Overview
- Procedure

Other topics about Calibration with ECal

Overview

The E5071C allows you to perform calibration using the 4-port ECal module. It provides much simpler operation than when using the 2-port ECal. Especially when using a multi-port test set, calibration time and operator errors can be reduced significantly.

Procedure

1. Connect the USB cable between the USB port of the 4-port ECal module and the USB port of the E5071C. You can make this connection while the E5071C's power is on.
2. Allows the ECal module to warm up for 20 minutes until the module indicates from WAIT to READY.
3. Connect the ports of the 4-port ECal module to the test ports you want to calibrate.
4. If you don't use all of the ECal module's ports, connect terminations to the unused ports.
5. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
6. Press **Cal** key.
7. Click **Ecal**.
8. Select the **calibration type**.

Softkey	Function
1-Port ECal	Selects 1-port calibration
2-Port ECal	Selects full 2-port calibration
3-Port ECal	Selects full 3-port calibration
4-Port ECal	Selects full 4-port calibration

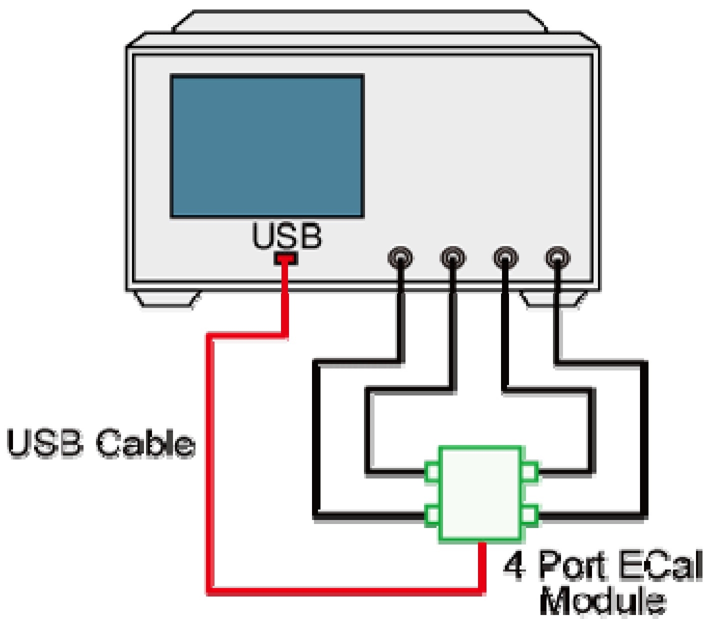
Thru ECal Selects THRU calibration

9.

10. If you must select a port, the softkey for making this selection is displayed. Select a port and start calibration. If you do not have to select a port, this step is skipped.

11. The E5071C detects the test ports connected to the ECal and then measurement starts. If the selected test ports to be calibrated is not connected to the ECal module, an error occurs.

Connecting 4-port ECal module (for full 4-port calibration)



e5071c325

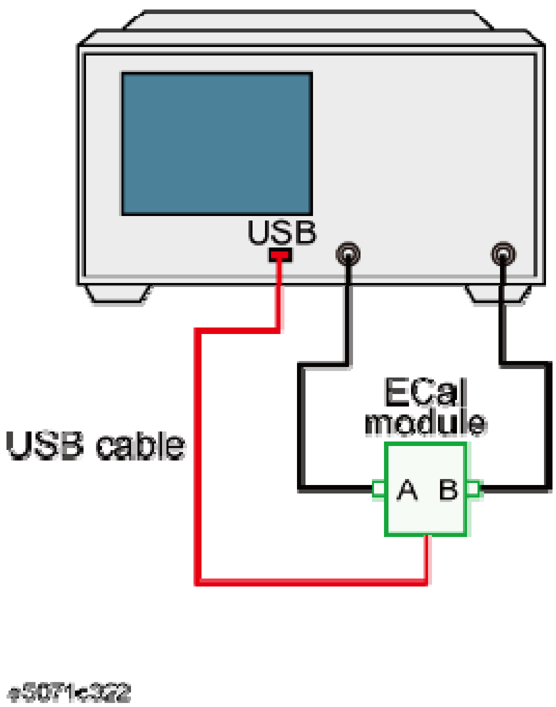
NOTE You can connect the ports of the ECal and the test ports of the E5071C arbitrarily. Connected ports can be manually specified although they are automatically detected before data measurement. For more information, Turning off ECal auto-detect function.

Full 2-Port Calibration Using the 2-Port ECal Module

Follow the procedure below to perform a full 2-port calibration using the 2-port ECal module.

1. Connect the USB port on the ECal module with the USB port on the E5071C via a USB cable. This connection may be done while the E5071C's power is on.
2. Allows the ECal module to warm up for 15 minutes until the module indicates from WAIT to READY.
3. Connect port A and port B on the ECal module to the test ports to be calibrated.
4. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
5. Press **Cal** key.
6. Click **ECal**.
7. Click **2 Port ECal**. When using a 2-port E5071C, pressing this key performs a 2-port ECal. When using a 3-port or 4-port E5071C, click one of the softkeys to start a full 2-port calibration.

Connecting ECal module (full 2-port calibration)



NOTE You can connect the ports of the ECal and the test ports of the E5071C arbitrarily. Connected ports can be manually specified although they are automatically detected before data measurement. For more information, Turning off ECal auto-detect function

Other topics about Calibration with ECal

Full 3-Port and Full 4-Port Calibration using 2-Port ECal

- Overview
- Procedure

Other topics about Calibration with ECal

Overview

A VBA macro (ECal Assistant) is pre-installed in the E5071C to carry out a full 3-port or a full 4-port calibration using the 2-port ECal.

NOTE ECal Assistant does not perform isolation calibration.

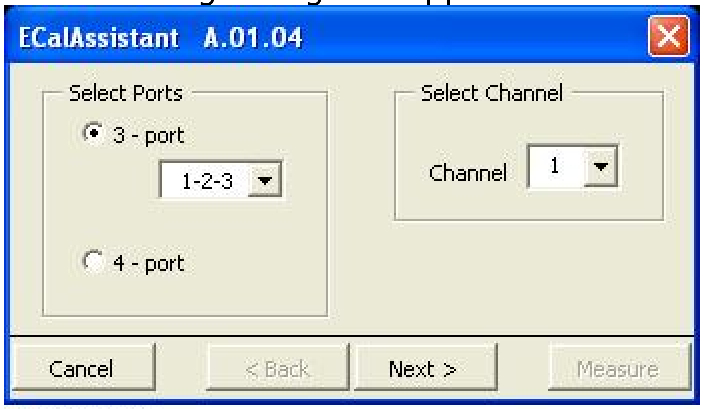
Procedure

1. Connect the USB port of the ECal module to the USB port on the E5071C with a USB cable. The connection may be made while the E5071C's power is on.
2. Allows the ECal module to warm up for 15 minutes until the module indicates from WAIT to READY.
3. Press **Macro Setup** key.
4. Click **Load Project.**
5. From the Open dialog box, select the VBA project file **D:\Agilent\ECalAssistant.vba** and click the **Open** button.
6. Press **Macro Run** key.
7. The following dialog box appears.

e5071c085

8. Click **Next.**

9. The following dialog box appears.



e5071c086

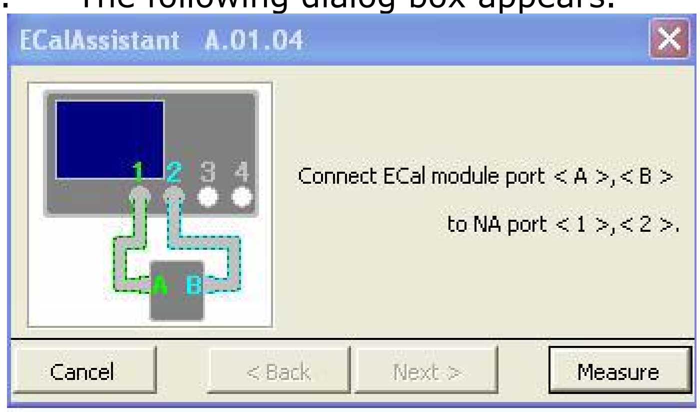
10. In the Select Ports area, click and select the **3-Port** (for a full 3-port calibration) or the **4-Port** (for a full 4-port calibration) radio button.

11. When a full 3-port calibration is carried out on an E5071C, select the test ports to be calibrated from the drop-down list box below the **3-Port** button (either **1-2-3**, **1-2-4**, **1-3-4**, or **2-3-4**).

12. In the Select Channel area, select the channel to be calibrated (one of channels **1** to **9**).

13. Click the **Next** button.

14. The following dialog box appears.

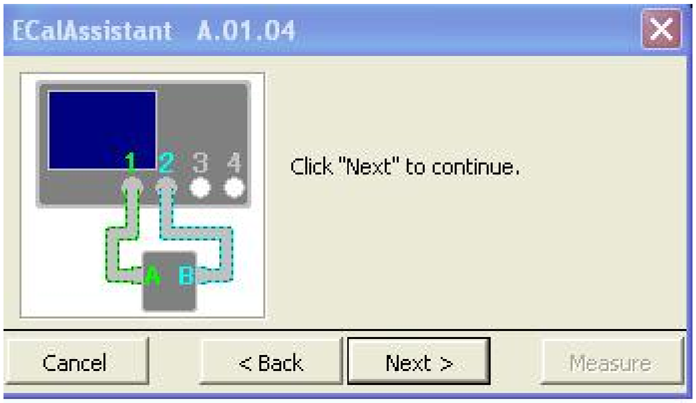


e5071c087

15. Following the connection diagram shown in the dialog box, connect ports A and B of the ECal module to test ports on the E5071C. The connection diagram shown in each dialog box that appears in each step depends on the number of test ports on the E5071C.

16. Click the **Measure** button to start the measurement of calibration data.

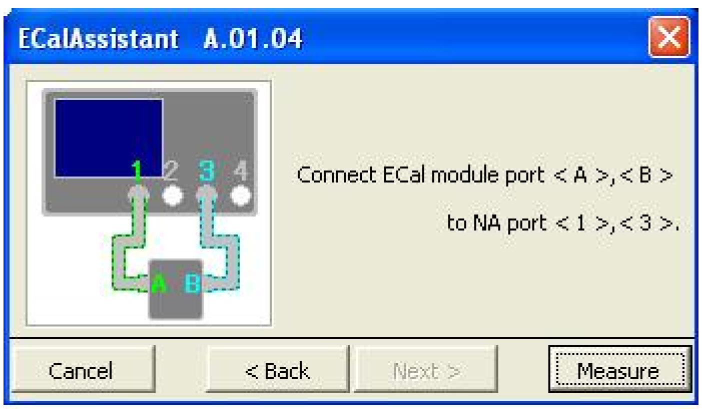
17. Upon completion of measurement, the following dialog box appears.



e5071c088

18. Click the **Next** button.

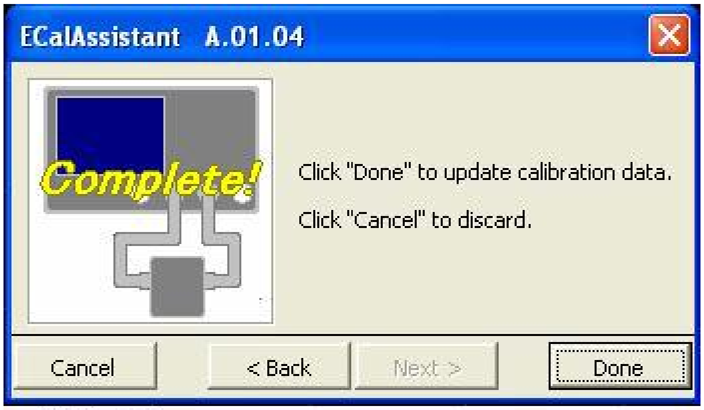
19. The following dialog box appears.



e5071c089

20. Re-connect the ECal module following the instructions given in each dialog box and continue the calibration process.

21. When all calibration data have been collected, a dialog box with the **Complete!** sign appears as shown in the following figure, click **Done** to finish the calibration. If you wish to cancel the calibration, click **Cancel**.


ECalAssistant A.01.04
Complete!
Click "Done" to update calibration data.
Click "Cancel" to discard.
Cancel
< Back
Next >
Done
e5071c099

Improving Calibration Accuracy along with ECal

Inaccuracy caused by Thru calibration in the ECal can be reduced by the using the following methods.

- Partial Overwrite
- Unknown Thru Calibration

Other topics about Calibration with ECal

Partial Overwrite

Using the partial overwrite function allows you to improve the calibration accuracy. For example, follow these steps for full 2-port calibration.

1. Execute full 2-port calibration with ECal and save the calibration coefficients.
2. Execute Partial overwrite procedure with the Thru standard of the calibration kit

Unknown Thru Calibration

E5071C allows you to perform Thru calibration of ECal as the unknown Thru calibration. In this function, the Thru calibration is performed with the Thru standard in ECal, however, the stored Thru calibration data in ECal is not used. E5071C performs the Thru calibration as an unknown Thru calibration.

1. Press **Cal** Key, then click **ECal**.
2. Click **Unknown Thru** to turn on.
3. Perform your desired ECal calibration.

Confidence Check on Calibration Coefficients Using ECal

- Overview
- Procedure

Other topics about Calibration with ECal

Overview

Using the ECal module, the E5071C lets you verify the obtained calibration coefficients to determine whether correct measurement is possible with them.

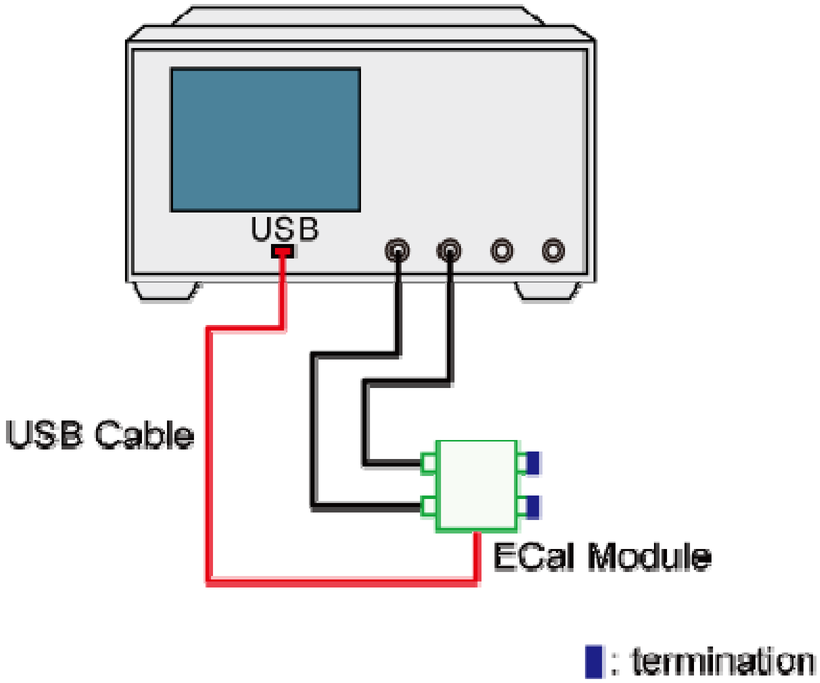
The E5071C can set ECal to the state used to verify the measurement parameters and then copy the appropriate characteristics of that verification state to the memory trace from the ECal's built-in memory. This is done according to the measurement parameters of the active trace of the active channel. While measuring ECal in this specified state, you can compare the measurement results with those of the E5071C and with the appropriate measurement results stored in ECal in several different ways. These include simultaneously displaying the data and memory traces or displaying the math operation results between the data and memory traces. This enables you to verify the correctness of measurement for each measurement parameter when the obtained calibration coefficients are used.

Procedure

1. Connect the USB cable between the USB port of the ECal module and that of the E5071C. You can make this connection while the E5071C's power is ON.
2. Allows the ECal module to warm up for 15 to 20 minutes until the module indicates from WAIT to READY.
3. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the verification.
4. Press **Meas** key.
5. Select the S-parameter you want to verify. You cannot verify the mixed mode S-parameter.
6. Connect the test ports of the E5071C corresponding to the selected S-parameter (for example, ports 1 and 2 when the S-parameter is S21) and the ports of the ECal module.
7. If you don't use all of the ECal module's ports, connect terminations to the unused ports.
8. Press **Cal** key.
9. Click **ECal**.

10. When using an adapter to the ECal, click **Characterization** and then press the softkey corresponding to the characterization for the adapter you are using.
11. Click **Confidence Check**.
12. Compare the data trace and the memory trace and verify whether measurement is correct.
13. The following is the procedure for comparison when simultaneously displaying the data trace and the memory trace.
    a. Press **Display** key.
    b. Click **Display** > **Data & Mem**.
    c. Press **Scale** key.
    d. Click **Auto Scale**.
    e. Determine whether the difference between the traces is acceptable. The difference should be read in terms of linear values instead of dB error. If we consider the magnitude of the linear error as compared to the dB delta, the value is very small. So to evaluate the difference between the traces, a linear error scale should be used for comparison instead of the dB error scale.
14. For all of the parameters you want to verify, repeat the procedure.

Connecting ECal module (for verification of S21)



e5071c328

Turning off ECal auto-detect function

The ECal module automatically detects the connection between E5071C's test ports and ECal module's ports. You can turn off this function to set ports manually.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to turn off the auto-detect function.
2. Press **Cal** key.
3. Click **ECal** > **Orientation**, and select **Manual**.
4. Specify a test port of the E5071C.
5. Specify an Ecal port for the port of the E5071C you specified.

NOTE Even if the connection is wrong with the auto-detect function turned off, no error is displayed.

Other topics about Calibration with ECal

User-characterized ECal

- Overview
- Precautions to take in using VBA macros
- Storing user characteristics to the ECal module
- Backup/recovery of ECal module's built in flash memory
- Executing User characterized ECal

Other topics about Calibration with ECal

Overview

The E5071C allows you to execute ECal calibration with user-defined characteristics instead of the ECal characteristics defined as the factory default. This feature is called User-characterized ECal, and it is used to execute ECal calibration when an adapter is connected to the ECal module.

Before executing the User-characterized ECal, you have to measure data, such as characteristics when the adapter is connected to the ECal module, and store them to the built-in flash memory of the ECal module as the user characteristics.

Use the following VBA macro to acquire user characteristics and store them to the ECal module's built-in memory.

Storage folder	VBA macro name (project name)
D:\Agilent	**EcalCharacterization.vba**

Precautions to take in using VBA macros

- Never connect/disconnect the USB cable while executing the VBA macro.

  CAUTION In particular, the above precaution must always be observed while the VBA macro is storing data to the ECal module's built-in flash memory; disconnecting the USB cable at this time may damage the ECal module.

- Back up the flash memory contents.
  The VBA macro provides a feature to back up the contents of the ECal module's built-in flash memory. Before storing user characteristics to the ECal module, be sure to use this feature to back up the flash memory's current contents.

Storing user characteristics to the ECal module

Follow these steps to measure characteristics while an adapter is connected to the ECal module and then to store them to the ECal module's built-in flash memory as user characteristics.

NOTE With the 2-port E5071C, you cannot measure the user characteristics of a 4-port ECal module and store them into the memory by using this VBA macro.

1. Connecting ECal Module

Connect the USB cable between the USB port of the ECal module and that of the E5071C. You can make this connection while the E5071C's power is ON.

2. Setting Stimulus Condition

Set the stimulus condition of the channel for which you want to measure the user characteristics. For optimal accuracy, set the IF bandwidth to 1 kHz or less.

3. Executing Calibration

For the channel for which you have set the stimulus condition, execute full 2/4-port calibration with a mechanical calibration kit when characterizing 2/4-port ECal. Define the calibration surface as the connector surface connected to each port of the ECal module in the state used to measure characteristics.

4. Starting the VBA MACRO

a. Press **Macro Setup** key.
b. Click **Load Project.**
c. The Open dialog box appears. Specify the file name "**D:\Agilent\EcalCharacterization.vba**" and click **Open**.
d. Press **Macro Run** key to start the macro. When more than 1601 measurement points is set for 1 channel and 4 traces, the E5071C VBA macro function may take more time to operate. A warning is also displayed if the user tries to increase the number of points to

more than 1601 with 1 channel and 4 traces. Click **OK**.

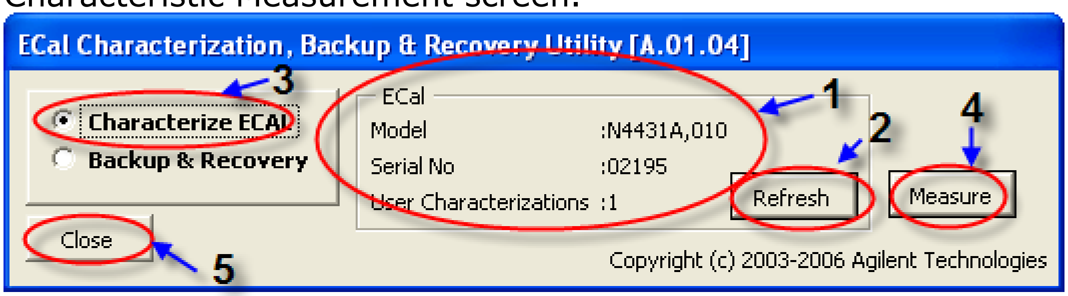
e5071c136

e. The ECal part (1 in the figure below) displays the information of the ECal module connected to the E5071C.

f. Click **Refresh** (2 in the figure below) to update the information if you have connected another ECal module after the macro has been started.

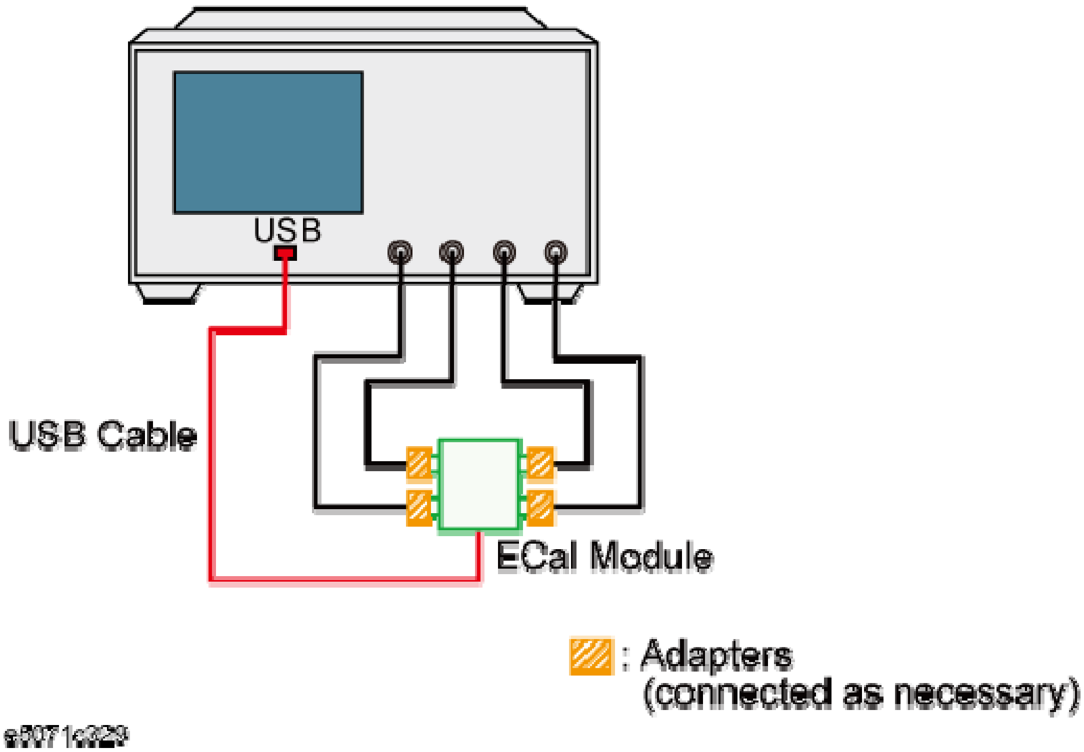
5. Measuring User Characteristics

a. Select Characterize ECAL (3 in the figure below) to display the User Characteristic Measurement screen.



e5071c232

b. After connecting the adapter to the ECal module as necessary, connect each port of the ECal module and the test port of the E5071C.



c. Click **Measure** (4 in the figure) to start measurement.

   a. You can select any port of the ECal module and any test port of the E5071C for connection; the E5071C automatically recognizes the connected ports before measurement.

6. Storing the User Characteristics to the Memory

a. When the measurement is complete, the User Characterization Info screen appears.

b. Enter the following information.

Designation	Category	Description
1	Number	Specify a user number (a location number in the memory where you want to store the user characteristics) . If the specified location number is not used for storage, the parts Characterization, Connectors, and Adapter Description are left blank; if already used, the stored contents are displayed.

2	Characterization	Enter the information (operator, used analyzer, and so on) when measuring user characteristics as necessary.
3	Connectors	Select the connector types of the adapters for the ECal module's test ports. Male and female in the list of connected types indicate male and female adapter, respectively. Select "No adapter" if no adapter is used on a port.
4	Adapter Description	Enter the detailed information on the adapters connected to each port as necessary.

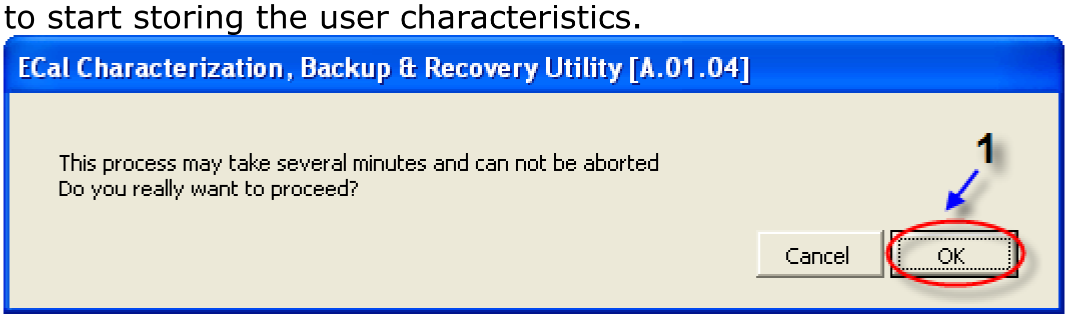
The information you have entered is displayed when checking the user characteristics information by using the key strokes: **Cal** > **Ecal** > **Characterization Info**.

d. Click **Write**.

e. At this time, if user characteristics are already stored for the specified user number, a dialog appears to confirm overwriting. Click **OK**.

a. Although the maximum number of user characteristics stored to the ECal's memory is usually five, this number may be limited by memory size because the size of user-characteristics data is not fixed and increases in proportion to the number of measurement points. An error occurs when the **Write** button is pressed if the total size added the new user characteristics exceeds this limitation due to memory size.

f. The following dialog box is displayed to confirm execution. Click **OK** to start storing the user characteristics.

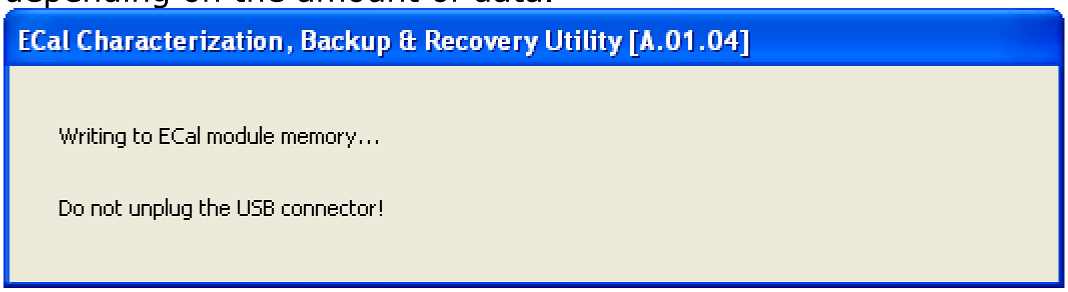


e5071c234

a. Do not disconnect the USB cable or terminate the VBA macro by force while the VBA macro is storing data to the

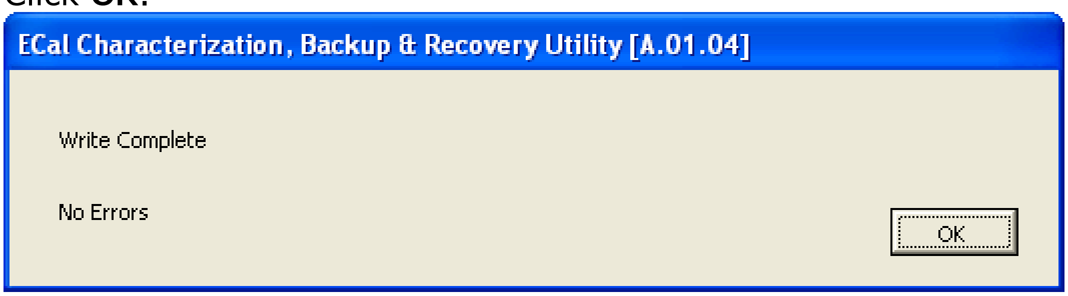
ECal's built-in flash memory. Doing so may damage the ECal module.

g. The following dialog box appears while the VBA macro is storing data to memory. Storing the user characteristics takes a few minutes depending on the amount of data.



e5071c236

h. Another dialog box is displayed to notify completion of data storage. Click **OK**.



e5071c237

8. Closing the VBA macro

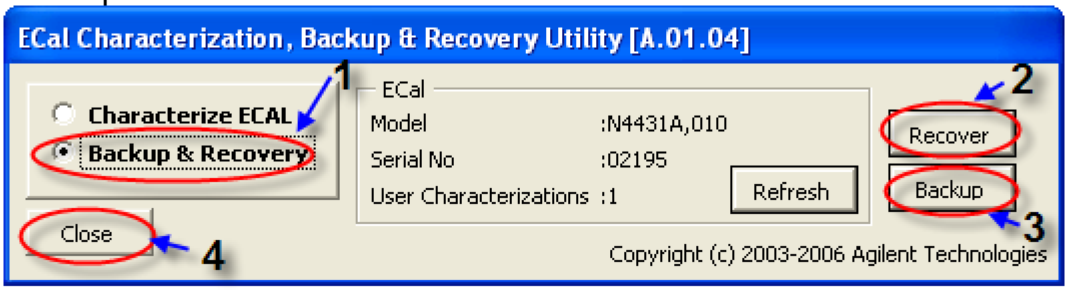
a. Click **Close** (No. 5 in Figure) to close the macro.

Backup and recovery of ECal module's built-in flash memory

Follow these steps to back up the contents of the ECal module's built-in flash memory.

1. Connect the USB cable between the USB port of the ECal module and that of the E5071C. You can make this connection while the E5071C's power is ON.
2. Start the VBA macro according to Starting the VBA MACRO

3. Select **Backup Flash ROM** (No. 1 in the following figure) to display the Backup screen.



e5071c235

Recovery

1. Click **Recover** (No. 2 in the figure above.).
2. The Open dialog box appears. Enter the file name of the contents you want to recover and Click **Open**. If the serial number information stored in the file does not match that of the ECal module connected to the E5071C, a confirmation dialog box appears. Click **OK** to continue the recovery only if a mismatch between these serial numbers is allowed.
3. The dialog box is displayed to confirm execution. Click **OK** to start the recovery of the flash memory. The dialog box appears while the VBA macro is storing data to the memory. The recovery of the flash memory takes a few minutes depending on the amount of data.
4. The Completion screen appears. Click **OK**.

Backup

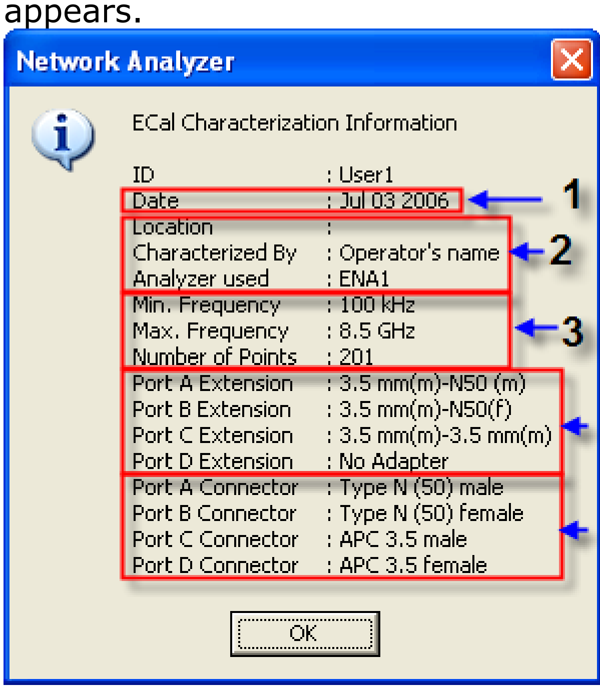
1. Click **backup**.
2. The Save As dialog box appears. Enter the name of the file you want to save and press **Save**.
3. Click **Close** to close the macro.

Executing User-characterized ECal

The execution procedure for the User-characterized ECal is the same as for normal ECal except that it requires the user characteristics to be selected in advance.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to execute calibration.
2. Press **Cal** key.
3. Click **ECal** > **Characterization**.

4. Select a user characteristic which is specified by User Characterization Info screen.
5. To check the information on the user characteristics you have selected, click **Characterization Info**. The following dialog box appears.



1:	The date when the user characteristics were measured
2:	The information you entered in Characterization (1 of User Characterization Info screen)
3:	The stimulus conditions when the user characteristics were measured
4:	The information you entered in Adapter Description (4 of User Characterization Info screen)
5:	The information you entered in Connectors (3 of User Characterization Info screen)

Advanced Calibrations

Setting the trigger source for calibration

- Overview of Trigger Source
- Setting Trigger Source

Other topics about Advanced Calibration

Overview of Trigger Source

You can set the trigger source for calibration before executing calibration.

You can select it from "Internal" or "System." Setting it to "System" allows you to use the same trigger source setting for calibration and measurement.

You can set the trigger source for calibration (which is usually set to "Internal") to "System," which allows you to generate triggers at any timing from a PC for external control or from the front panel for calibration in the same way as for measurement.

The setting of the point trigger and averaging trigger is also applied to the trigger for calibration. When the trigger source for calibration is set to "System" and the trigger source for measurement is set to "External" or "Manual" with the point trigger function set to ON, a trigger is required for each measurement point during calibration. When the averaging trigger function is set to ON, the sweep is performed the number of times specified by the averaging factor for a single trigger during calibration.

NOTE For the following types of calibration, the setting of the trigger source does not take effect. Those calibration operations are controlled by the internal trigger.

- Calibration using ECal
- Power calibration
- Receiver calibration
- Mixer converter calibration

Setting Trigger Source

Follow these steps to set the trigger source for calibration:

1. Press **Cal** Key.
2. Click **Cal Trig Source**, then select the trigger source you want to use.

Softkey	Function
Internal	Selects "Internal."

System	Selects "System."

Sliding and Offset Load Calibration

- Supported Calibration
- Sliding Load Calibration
- Offset Load Calibration

Other topics about Advanced Calibration

Supported Calibration

The following table shows the supported calibration for the sliding load and offset load.

Calibration	SOLT	Response	Enhanced Response	TRL-Line	Adapter Removal	Scalar Mixer	Vector Mixer (VBA)
Sliding Load	Yes	Yes	Yes	Yes	Yes	No	No
Offset Load	Yes	No	Yes	No	No	No	No

Sliding Load Calibration

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 calibration can be very accurate when performed perfectly. It can also be very inaccurate when not using proper technique. For accurate results, follow the users manual instructions closely.

The following calibration kits are supported:

- 85050B 7 mm Calibration Kit
- 85052B 3.5 mm Calibration Kit
- 85054B 50 Ohm Type-N Calibration Kit
- 85056A 2.4 mm Calibration Kit

Procedure

Follow these steps to perform calibration using sliding load. The example demonstrates re-calibration for full 1-port calibration.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **Cal Kit.**
4. Select the calibration kit with sliding load, for example, **85052B.**
5. Press **Cal** key.
6. Click **Calibrate** > **1-Port Cal** > **Select Ports.**
7. Select the test ports for which you want to perform re-calibration for full 1-port calibration.
8. Make a OPEN connection on the selected test port.
9. Click **Open.**
10. Make a SHORT connection on the selected test port.
11. Click **Short.**
12. Make a LOAD connection on the selected test port with a sliding load.
13. Click **Calibrate** > **1-Port Cal** > **Load** > **Load 2 Sliding** for female load OR **Calibrate** > **1-Port Cal** > **Load** > **Load 5** for male load. The assignment of female and male loads varies from one calibration kit to another. In the case of 85052B, load 2 is a female and load 5 is a male load.
14. Slide the load to the desired position and click **Sliding Load.** Repeat this step for at least 5 times (till **Sliding Load 5**), then click **Done.** This step can be repeated at most 10 times only.
15. Click **Return.**
16. Now, all **Short**, **Open** and **Load** should have check mark on their left.
17. Click **Done.**

The sweep range setting should match the frequency range of the sliding load. Else, the load process is not executed, hence no check mark appear on the **Load** softkey. Similarly, if the frequency range is extended, the load process is also not executed. Refer to redefining a calibration kit to learn more about it.

Offset Load Calibration

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.

The following calibration kits are supported:

- X11644A X Band 90 Calibration Kit
- P11644A P Band WG 62 Calibration Kit
- K11644A K Band WG 42 Calibration Kit

NOTE The K11644A K Band WG 42 Calibration Kit is not verified.

Procedure

Follow these steps to perform calibration using offset load. The example demonstrates re-calibration for full 1-port calibration.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **Cal Kit.**
4. Select the calibration kit with sliding load, for example, **X11644A.**
5. Press **Cal** key.
6. Click **Calibrate** > **1-Port Cal** > **Select Ports**.
7. Select the test ports for which you want to perform re-calibration for full 1-port calibration.
8. Make a OPEN connection on the selected test port.
9. Click **Open**.
10. Make a SHORT connection on the selected test port.
11. Click **Short.**
12. Click **Calibrate** > **1-Port Cal** > **Load** > **Load 2 Offset Load**.
13. Make a LOAD connection on the selected test port with a offset load (Thru).
14. Click **Offset Line 1 Thru**.

15. Make a LOAD connection on the selected test port with a offset load (X-band Delay line).
16. Click **X-band Delay Line**.
17. Click **Done.**
18. Click **Return** twice.
19. Now, all **Short**, **Open** and **Load** should have check mark on their left.
20. Click **Done**.

The sweep range setting and Thru Offset frequency range should match the frequency range of the offset load. Else, the load process is not executed, hence no check mark appear on the **Load** softkey. Similarly, if the frequency range is extended, the load process is also not executed. Refer to redefining a calibration kit to learn more about it.

Modifying Calibration Kit Definition

Other topics about Advanced Calibration

In most measurements, the user can use pre-defined calibration kits as they are. However, it may be necessary to change the definition of a calibration kit (or create a new one) when changing the pre-defined connector between male and female (e.g. from OPEN (f) to OPEN (m)) or when a special standard is used or a high degree of accuracy is demanded. When it is necessary to change the definition of a calibration kit that contains a calibration device but no calibration kit model, the user must fully understand error correction and the system error model.

A user-defined calibration kit may be used in the following circumstances.

- When the user wants to use connectors other than those pre-defined in the calibration kits for the E5071C (e.g., a SMA connector).
- When the user wants to use different standards in place of one or more standards pre-defined in the E5071C. For example, when three offset SHORT standards are used instead of OPEN, SHORT, and LOAD standards.
- When the user wants to modify the standard model of a pre-defined calibration kit and turn it into a more accurate model. It is possible to perform better calibration if the performance of the actual standard is better reflected in the standard model. For example, you may need to define the 7-mm LOAD standard as 50.4 ohm instead of 50.0 ohm.

Definition of Terms

The terms used in this section are defined as follows:

Standard

An accurate physical device, for which the model is clearly defined, used to determine system errors. With the E5071C, the user may define up to 21 standards per calibration kit. Each standard is numbered from 1 through 21. For example, standard 1 for the 85033E 3.5-mm calibration kit is a SHORT standard.

Standard type

The type of standard used to classify a standard model based on its form and construction. Six standard types are available: SHORT, OPEN, LOAD, delay/THRU, Unknown Through, and arbitrary impedance.

Standard coefficient

The numeric characteristics of the standard used in the selected model. For example, the offset delay (32 ps) of the SHORT standard in the 3.5-mm calibration kit is a standard coefficient.

Standard class

A group of standards used in a calibration process. For each class, the user must select the standards to use from the 21 available standards.

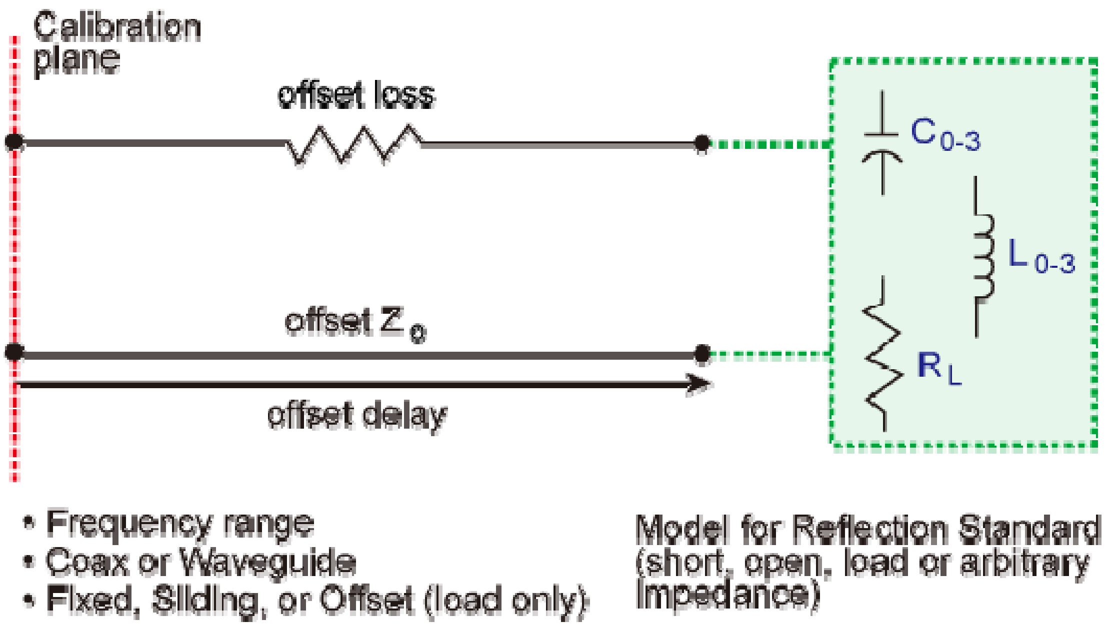
Subclass

You can register up to 8 standard types. This capability lets you specify a different standard for each frequency range. You must assign standards to subclasses you use.

Defining Parameters for Standards

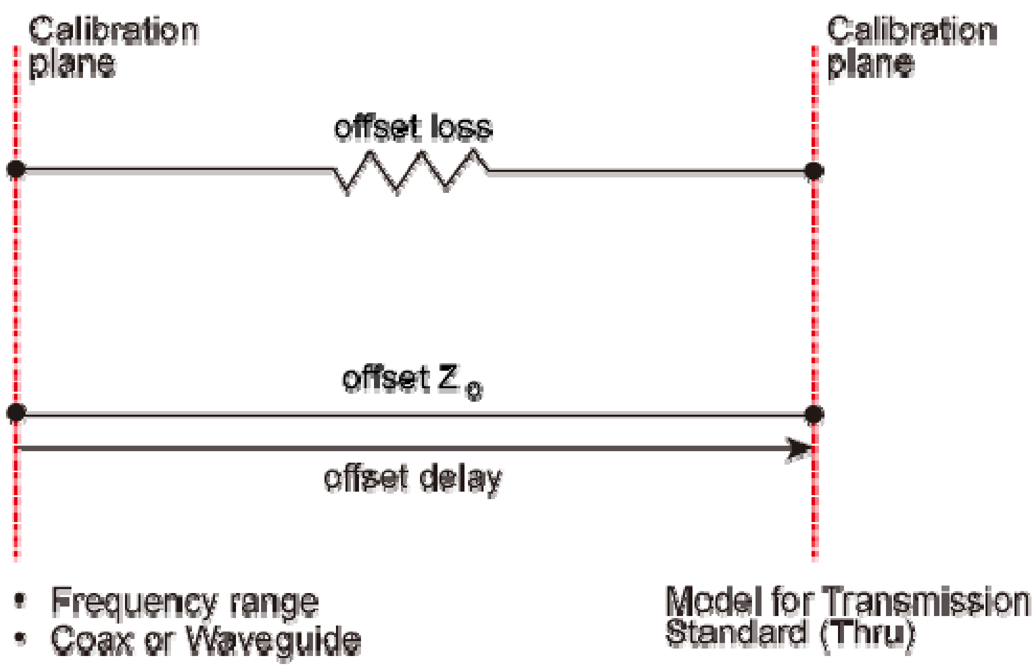
The following figures show the parameters used in defining standards.

Reflection Standard Model (SHORT, OPEN, or LOAD)



e5071c494

Transmission Standard Model (THRU)



e5071c495

Offset Z0

The offset impedance between the standard to be defined and the actual measurement plane. Normally, this is set to the system's characteristic impedance.

Offset Delay

The delay that occurs depending on the length of the transmission line between the standard to be defined and the actual measurement plane. In an OPEN, SHORT, or LOAD standard, the delay is defined as one-way propagation time (sec.) from the measurement plane to the standard. In a THRU standard, it is defined as one-way propagation time (sec.) from one measurement plane to the other. The delay can be determined through measurement or by dividing the exact physical length of the standard by the velocity coefficient.

Offset Loss

This is used to determine the energy loss caused by the skin effect along the length (one-way) of the coaxial cable. Loss is defined using the unit of ohm/s at 1 GHz. If you select waveguide as the media type, this parameter is ignored. In many applications, using the value 0 for the loss should not result in significant error. The loss of a standard is determined by measuring the delay (sec.) and the loss at 1 GHz and then substituting them in the formula below.

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

C0, C1, C2, C3

It is extremely rare for an OPEN standard to have perfect reflection characteristics at high frequencies. This is because the fringe capacitance of the standard causes a phase shift that varies along with the frequency. For internal calculation of the analyzer, an OPEN capacitance model is used. This model is described as a function of frequency, which is a polynomial of the third degree. Coefficients in the polynomial may be defined by the user. The formula for the capacitance model is shown below:

$$C = (C0) + (C1 \times F) + (C2 \times F^2) + (C3 \times F^3)$$

F: measurement frequency
C0 unit: (Farads) (constant in the polynomial)
C1 unit: (Farads/Hz)

C2 unit: (Farads/Hz^2)
C3 unit: (Farads/Hz^3)

L0, L1, L2, L3

It is extremely rare for a SHORT standard to have perfect reflection characteristics at high frequencies. This is because the residual inductance of the standard causes a phase shift that varies along with the frequency. It is not possible to eliminate this effect. For internal calculation of the analyzer, a short-circuit inductance model is used. This model is described as a function of frequency, which is a polynomial of the third degree. Coefficients in the polynomial may be defined by the user. The formula for the inductance model is shown below.

$$L = (L0) + (L1 \times F) + (L2 \times F^2) + (L3 \times F^3)$$

F: Measurement frequency
L0 unit: [Henry] (the constant in the polynomial)
L1 unit: [Henry/Hz]
L2 unit: [Henry/Hz^2]
L3 unit: [Henry/Hz^3]

In most existing calibration kits, THRU standards are defined as "zero-length THRU," i.e., the delay and loss are both "0". Such a THRU standard does not exist, however. Calibration must be done with two test ports interconnected directly.

NOTE The measurement accuracy depends on the conformity of the calibration standard to its definition. If the calibration standard has been damaged or worn out, the accuracy will decrease.

Redefining a Calibration Kit

This section provides the procedure to change the definition of a calibration kit.

- Procedure to select and define a calibration kit
- Procedure to select the standard type and define standard coefficient
- Procedure to define standard class

Procedure to select and define a calibration kit

1. Press **Cal** key.
2. Click **Cal Kit**, then select the calibration kit to be redefined. If the names (labels) of calibration kits were changed prior to operation, the new names will appear on the respective softkeys.

3. Click **Modify Kit.** To change the pre-defined connector type (e.g. OPEN(f) to OPEN (m)) skip to the specifty classes.
4. Click **Define STDs.**
5. Select the standard to be redefined from among standards numbered 1 through 21.
6. Click **No Name** > **Label** and input a new label for the standard using the keypad displayed on the screen.

Procedure to select the standard type and define standard coefficient

1. Press **Cal** key.
2. Click **Modify Kit.**
3. Click **Define STDs** and select the standard to be redefined from among standards numbered 1 through 21.
4. Click **STD Type**, then select the **type of standard**.

Softkey	Function
Open	Selects the OPEN standard
Short	Selects the SHORT standard
Load	Selects the LOAD standard
Delay/Thru	Selects the delay/THRU standard
Unknown Thru	Selects the Unknown THRU standard
Arbitrary	Selects the arbitrary impedance
None	Selects no standard type

5.
6. Set the **standard coefficient**.

Softkey	Function
C0	Sets C0

C1	Sets C1
C2	Sets C2
C3	Sets C3
L0	Sets L0
L1	Sets L1
L2	Sets L2
L3	Sets L3
Offset Delay	Sets the offset delay
Offset Z0	Sets the offset Z0. Specify 1 ohm when Waveguide is set for the media type.
Offset Loss	Sets the offset loss
Arb. Impedance	Sets an arbitrary impedance
Min. Frequency	Sets a start frequency. Specify cutoff frequency when Waveguide is set for the media type.
Max. Frequency	Sets a stop frequency. Specify cutoff frequency when Waveguide is set for the media type.
Media	Selects a media type, Coaxial or Waveguide.
Length Type	When the standard type is Load, this softkey can be selected from Fixed, Sliding or Offset.

7.

8. Repeat step 1 to step 5 to redefine all standards for which changes are necessary, then click **Return**.

Procedure to define standard class

1. Press **Cal** key.
2. Click **Modify Kit**.
3. Click **Specify CLSs**, then select the **class**.

Softkey	Function
Sub Class	Selects a subclass you want to use.
Open	Selects the OPEN class
Short	Selects the SHORT class
Load	Selects the LOAD class
Thru	Selects the THRU class
TRL Reflect	Selects TRL Reflect.
TRL Line/Match	Selcts TRL Line/Match.

4.
5. For TRL Thru, select **ports**. Select **Set All** to use the same standards for all test ports.

Softkey	Function
Port 1-2	Selects port 1 and port 2.
Port 1-3	Selects port 1 and port 3.
Port 1-4	Selects port 1 and port 4.
Port 2-3	Selects port 2 and port 3.

Port 2-4 Selects port 2 and port 4.

Port 3-4 Selects port 3 and port 4.

6.
7. Select the test port. Select **Set All** to use the same standards for all test ports.
8. Select the standards to be registered in the class from among standards numbered 1 through 21. To change the connectors between male and female (e.g. OPEN (f) to OPEN (m)), select the appropriately labeled standards here. For more information on changing the gender, refer to Defining Calibration Kit Gender.
9. Repeat the procedure until classes are defined for all test ports that need to be redefined, then click **Return**.
10. Repeat the procedure to redefine all classes that need to be modified, then click **Return**.
11. Click **Label Kit** and input a new label for the calibration kit by using the keypad displayed on the screen.

You can save the definition. See Saving Definition File of Calibration Kit.

Example of defining the TRL calibration kit

You need to enter the definition of the TRL calibration kit to perform TRL calibration. Follow these steps to define the following calibration kit given as an example.

- THRU (Delay 0 ps, Offset Loss 1.3 Gohm/s)
- REFLECT (SHORT, Delay 0 ps)
- MATCH (@0 to 2GHz)
- LINE1 (50-ohm transmission line, Delay 54.0 ps @ 2G to 7GHz)
- LINE2 (50-ohm transmission line, Delay 13.0 ps @ 7G to 32GHz)

Procedure to define the name of the calibration kit

1. Press **Cal** key.
2. Click **Cal Kit**, then select a User kit that has not been registered.
3. Click **Modify Kit** > **Label Kit [User]**, then type in a name you want.

Procedure to define Thru and Reflect

1. Click **Define STDs**.
2. Click **1:No Name** > **Label**, then type in **THRU**.

3. Select **STD Type** > **Delay/Thru**.
4. Set **Offset Loss** to 1.3Gohm/s, and **Offset Delay** to 0.
5. Click **Return** to return to the Define STD menu.
6. In the same way, repeat the procedure to enter the definition of REFLECT to No. 2.
7. Select **SHORT** for STD Type.

Procedure to define Match

1. Click **3:No Name** > **Label**.
2. Type in **MATCH <2G**.
3. Click **STD Type** > **Load**.
4. Set **Max Frequency** to 2GHz.
5. Click **Return** to return to the Define Std menu.

Procedure to define Line 1/2

1. Click **4:No Name - Label**.
2. Type in **LINE <7G**.
3. Click **STD Type - Thru.**
4. Set **Offset Delay** to 54 ps.
5. Set **Min Frequency** to 2 GHz.
6. Set **Max Frequency** to 7 GHz.
7. Click **Return** to return to the Define Std menu.
8. Click **5:No Name - Label**.
9. Type in **LINE >7G**.
10. Click **STD Type** > **Thru**.
11. Set **Offset Delay** to 13 ps.
12. Set **Min Frequency** to 7 GHz.
13. Click **Return** to return to the Define Std menu.
14. Click **Return** to return to the Modify Cal Kit menu.

Procedure to register Thru/Reflect/Match to sub class 1

1. Click **Specify CLSs** > **Sub Class.**
2. Select **Sub Class1**.
3. Click **TRL Thru** > **Set All** > **THRU** > **Return.**
4. Click **TRL Reflect** > **REFLECT**.
5. Click **TRL Line/Match** > **Set All** > **MATCH <2G** > **Return.**

Procedure to register Line 1/2 to subclass 2/3

1. Click **Sub Class2**.
2. Click **TRL Line/Match** > **Set All** > **LINE <7G** > **Return.**
3. Click **Sub Class3**.
4. Click **TRL Line/Match** > **Set All** > **LINE >7G** > **Return.**
5. Press **Cal** key and check that the name you specified is selected as Cal Kit.

By assigning Match and Line 1/2 to subclass 1/2/3 respectively, you can calibrate 3 standards with different frequency bands for TRL line/match calibration.

Setting Options for TRL Calibration

This section describes how to set the reference impedance and measurement data of the standard used for calculating the calibration plane for TRL calibration.

1. Press **Cal** key.
2. Click **Cal Kit**, then select a calibration kit.
3. Click **Modify Cal Kit** > **TRL Option.**
4. Click **Impedance** to select the **reference impedance**.

Softkey	Function
Line	Calculates the calibration coefficient using the characteristic impedance of the line standard as the reference impedance. The impedance of the line standard is used as the center of the Smith chart, and the calibration coefficient is calculated using the line standard's as the direction.
System	Calculates the calibration coefficient using the system impedance as the reference impedance. Select this when the line impedance and the

test port impedance differ.

5. Click **Reference Plane** to select the **measurement data of the standard** used for calculating the reference plane.

Softkey	Function
Thru	Uses the length of the thru/line standard to calculate the calibration plane.
Reflect	Uses the reflection coefficient of the reflection standard to calculate the calibration plane.

6.

Setting a media type for the calibration kit

You can set a media type for the standard you use.

1. Press **Cal** key.
2. Click **Cal Kit**, then select a calibration kit.
3. Click **Modify Cal Kit** > **Modify Cal Kit.**
4. Click **Define STDs**, and select a standard.
5. Click **Media**, and select a **media type**.

Softkey	Function
Coaxial	Selects coaxial as the media type.
Waveguide	Selects waveguide as the media type.

6.

    7. If you select waveguide as the media type, set the system impedance and the characteristic impedance to 1ohm .
    8. Depending on the media type you use, the calculation method of the electrical delay, which is required to correct the phase delay, differs.

You can save the definition. See Saving Definition File of Calibration Kit.

Defining Calibration Kit Gender (m/f)

The gender of a calibration can be set using the subclass function in the ENA. For example, subclass 1 can be used as "male" and subclass 2 can be used as "female" or vice-versa. The display label, for example "**Open (f)**" or "**Open (m)**", is just a display label and can be edited using **Cal** > **Modify Kit** > **Define STDs** and hence cannot be used to change the gender of the calibration kit. To change the gender of the calibration kit, the E5071C subclass function must be used.

For example, the gender of the default calibration kit (85032F) can be checked using the following procedure:

1. Press **Cal** key.
2. Click **Cal Kit** and select **85032F**.
3. Select **Cal** > **Modify Cal Kit** > **Specify CLSs** > **Sub Class**.

The subclass gender is set from the E5071C side and not from the Calibration kit side.

If both male/female are set using subclass in advance, the gender which is used for the calibration can also be changed programmatically using SCPI.SENSe(Ch).CORRection.COLLect.ACQuire.SUBClass

- The gender setting option using subclass is defined from the E5071C connector type and not from the standard calibration kit gender type.

Saving Definition File of Calibration Kit

You can save the definition file of the calibration kit for each standard into a file on a storage device, then recall it later to reproduce the definition.

1. Press **Cal** key.
2. Click **Cal Kit**, then select a calibration kit.
3. Click **Modify Cal Kit** > **Export Cal Kit...** to open the dialog box.
4. Specify a folder, enter a file name, and click **Save**.

NOTE The file is saved with the (.ckx) extension. Do not open and modify it directly.

CAUTION Never modify the contents (folders and files) of drives except for drive D. Modifying the contents of drives other than drive D may give serous damage to the functions and performance of the analyzer.

Loading Definition File of Calibration Kit

1. Press **Cal** key.
2. Click **Cal Kit**, then select a calibration kit.

3. Click **Modify Cal Kit** > **Import Cal Kit...** to open the dialog box.
4. Specify a folder, enter a file name, and click **Open**.

Restoring Definition File of Calibration Kit

1. Press **Cal** key.
2. Click **Cal Kit**, then select a calibration kit.
3. Click **Modify Cal Kit** > **Restore Cal Kit...**.
4. Click **OK** to restore the definition of the calibration kit selected by **Cal Kit** to factory default settings.

Specifying Different Standards for Each Frequency

- Overview
- Defining Standards for Each Subclass
- Disabling Standards Defined for a Subclass
- Possible Cases of Frequency Ranges Using Subclasses

Other topics about Advanced Calibration

Overview

This section demonstrates the procedure to define a different open standard for each frequency band, based on the following information:

Standard	Label name	Frequencies to be defined
Open	Open 3G	1 GHz - 3 GHz
Open	Open 6G	3 GHz - 6 GHz

1. Press **Cal** key.
2. Click **Cal Kit.**
3. Select a calibration kit you want to define.
4. Click **Modify Kit** > **Define STDs.**
5. Select #1, click **Label**, and enter **Open 3G** using the character input pad that appears on the screen.
6. Click **Open** as the standard type (**STD Type**).
7. Set a necessary standard coefficient.
8. Enter **1G** as the minimum frequency for **Min Frequency.**
9. Enter **3G** as the maximum frequency for **Max Frequency.**
10. Click **Return.**
11. In the same way, select #2, press **Label**, and enter **Open 6G** using the character input pad that appears on the screen.
12. Select **Open** as the standard type (**STD Type**).
13. Set a necessary standard coefficient.
14. Enter **3G** as the minimum frequency for **Min Frequency.**
15. Enter **6G** as the maximum frequency for **Max Frequency.**
16. Click **Return.**

Defining Standards for Each Subclass

This section demonstrates the procedure to specify a different OPEN standard for each frequency band, using subclasses #1 and #2. In this example, the standards created in Defining different standard for each frequency band are used.

1. Press **Cal** Key.
2. Click **Cal Kit.**
3. Select a calibration kit you want to use.
4. Click **Modify Kit** > **Specify CLSs**.
5. Click **Sub Class** > **Sub Class 1**.
6. Select the standard type **Open**, press **Set All**, make settings for all ports.
7. Select #1 (**Open 3G**).
8. Click **Return.**
9. Then, press **Sub Class**, and select **Sub Class 2**.
10. Select the standard type **Open**, press **Set All**, make settings for all ports.
11. Select #2 (**Open 6G**).
12. Click **Return**.

Disabling Standards Defined for a Subclass

The following procedure shows how to disable a standard defined for a subclass. You cannot disable subclass 1 because at least one standard must exist. The following procedure shows how to disable an OPEN standard of subclass 2.

1. Press **Cal** key.
2. Click **Cal Kit.**
3. Select a calibration kit you want to use.
4. Click **Modify Kit** > **Specify CLSs**.
5. Click **Sub Class** > **Sub Class 2**.
6. Select the standard type **Open**, click **Set All**, make settings for all ports.
7. Select **None**.
8. Click **Return**.

Possible Cases of Frequency Ranges Using Subclasses

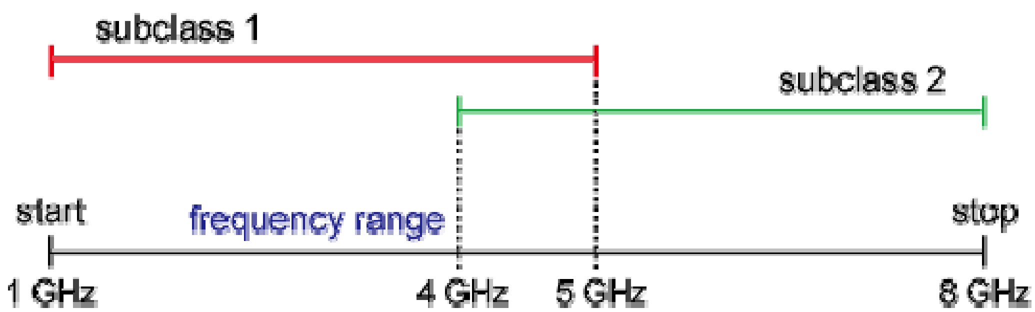
By using several subclasses, you can set a different standard for each frequency, but note the following on frequency ranges. The following table shows possible cases and whether calibration is possible.

Possible cases of frequency ranges	Execution of calibration

Example 1, When frequency ranges specified with subclasses overlap	Possible
Example 2, When frequency ranges specified with subclasses do not cover a part of a measurement frequency range	Impossible
Example 3, When a frequency range specified with a subclass lies outside the measurement frequency range	Possible
Example 4, When a part of a frequency range specified with a subclass lies outside the measurement frequency range	Possible

Example 1, When frequency ranges specified with subclasses overlap

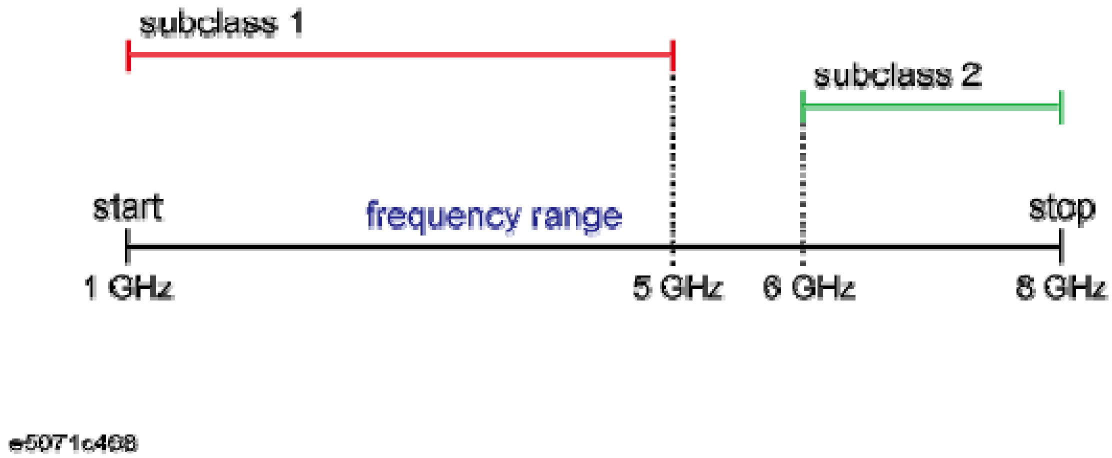
For the measurement frequency range (1 GHz - 8 GHz), if you execute calibration with a standard of subclass 1 (1 GHz-5 GHz) and then with a standard of subclass 2 (4 GHz-8 GHz), the standard last executed is applied for the overlapping portion (4 GHz-5 GHz).



e5071c470

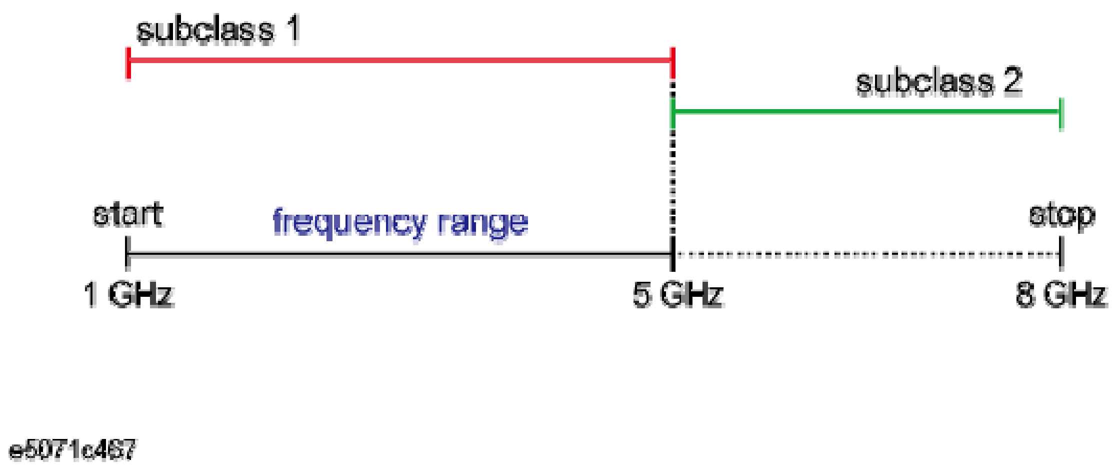
Example 2, When frequency ranges specified with subclasses do not cover a part of a measurement frequency range

For a measurement frequency range (1 GHz to 8 GHz), if you execute calibration with a standard of subclass 1 (1 GHz to 5 GHz) and then with a standard of subclass 2 (6 GHz to 8 GHz), **Done** is NOT available for the undefined portion (5 GHz-6 GHz).



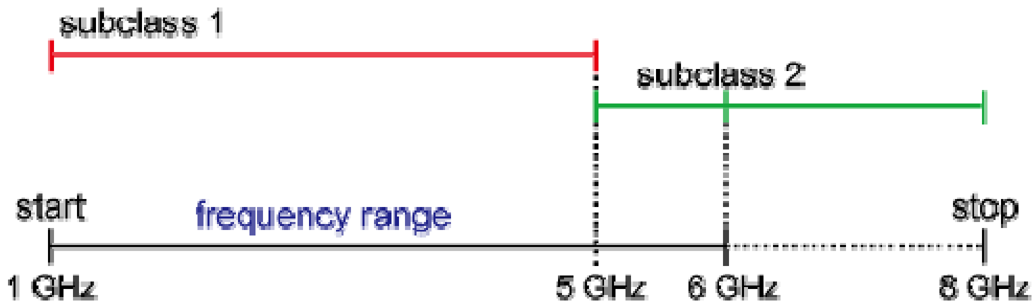
Example 3, When a frequency range specified with a subclass lies outside the measurement frequency range

For a measurement frequency range (1 GHz to 5 GHz), if you define a standard of subclass 1 (1 GHz to 5 GHz) and a standard of subclass 2 (5 GHz to 8 GHz), calibration possible for subclass 2 although it is out of the measurement frequency range. The calibration coefficients, however, are disregarded. (There is no check mark on the softkey.) Note that, because the standard of subclass 1 covers the measurement frequency range, **Done** is available.



Example 4, When a part of a frequency range specified with a subclass lies outside the measurement frequency range

For a measurement frequency range (1 GHz to 6 GHz), if you define a standard of subclass 1 (1 GHz - 5 GHz) and a standard of subclass 2 (5 GHz to 8 GHz), a part of subclass 2 (6 GHz to 8 GHz) lies outside the measurement frequency range, but the other part (5 GHz to 6 GHz) lies within it, so calibration is possible.


subclass 1
subclass 2
start
frequency range
stop
1 GHz
5 GHz
6 GHz
8 GHz

e5071c469

Receiver Calibration

- Overview
- Turning Receiver Error Correction ON/OFF
- Selecting Target Port for Error Correction
- Measuring Calibration Data

Other topics about Advanced Calibration

Overview

The E5071C has a function to calibrate the gain of the individual receivers in absolute value measurement.

The receiver calibration function calibrates the gain of the receiver by inputting the output power of a stimulus port that has been assigned a correct value by the power calibration to the port of the receiver you need to calibrate.

NOTE Receiver calibration is valid only for the parameters of absolute value measurement.

Turning Receiver Error Correction ON/OFF

The receiver calibration data are acquired for each channel/test port, so the gain error correction can be turned ON/OFF independently for any channel or port.

The status of the gain error correction for each channel is indicated by one of the symbols in the channel status bar in the lower part of the window, as shown in the following table.

Symbol	Status of receiver error correction
RC (displayed in blue)	Error correction is performed for all of the receiver ports.
RC (displayed in grey)	Error correction is performed for some of the receiver ports.
RC? (displayed in blue)	Error correction is performed for all of the receiver ports. Interpolated calibration data is used.
RC? (displayed in grey)	Error correction is performed for some of the receiver ports. Interpolated calibration data is used.
RC! (displayed in blue)	Error correction is performed for all of the receiver ports. Extrapolated calibration data is used.
RC! (displayed in grey)	Error correction is performed for some of the receiver ports. Extrapolated calibration data is used.
--- (displayed in gray)	Error correction is not performed.

	(At least one receiver port is turned on for error correction, but valid calibration data are not available.)
None	Error correction is not performed. (Error correction is turned off for all of the receiver ports.)

Turning receiver error correction ON/OFF

The error correction of the receiver is automatically turned on when the calibration data are measured. You can also turn this ON/OFF explicitly by following the steps below.

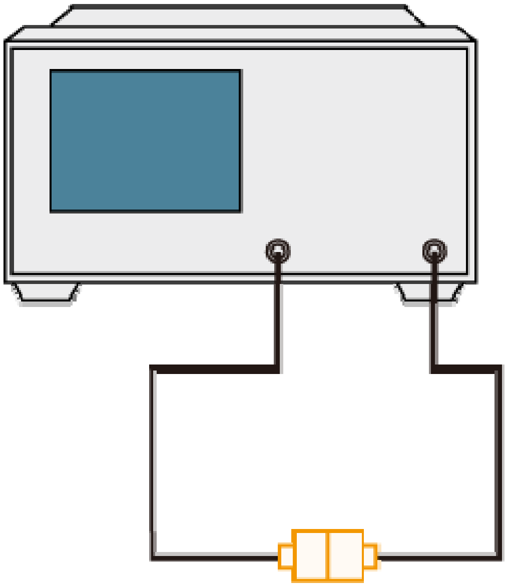
1. Press **Cal** key.
2. Press **Receiver Calibration**.
3. Select a port (see Selecting target port for error correction).
4. Click **Correction**. Each time the key is pressed, ON and OFF switches over alternately.

Selecting Target Port for Error Correction

The error correction of receiver ports is performed for each channel/port, and you can set the following items for them:

- Turning on or off error correction
- Calibration data

1. Press **Cal** key.
2. Press **Receiver Calibration**.
3. Press **Channel Next**/**Channel Prev** keys to select a channel.
4. Click **Select Port**.



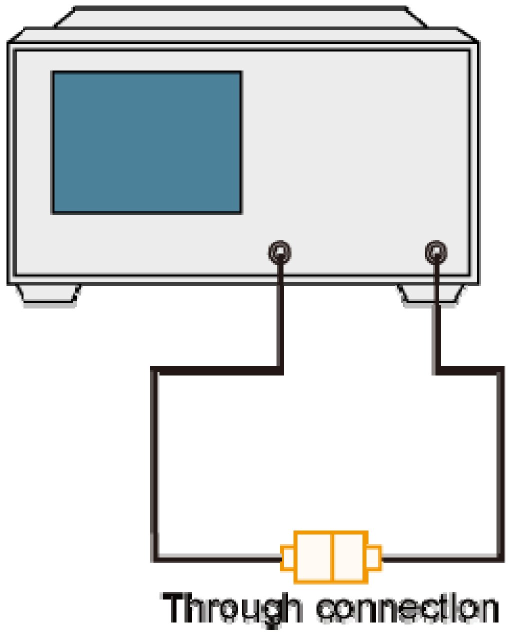
e5071c821

5. Click the softkey corresponding to the port you want to calibrate.

Measuring Calibration Data

NOTE The power calibration information on both the receiver port and source port is used to calculate calibration coefficients. The accuracy of receiver calibration will increase if power calibration is implemented for both the receiver port and the source port before starting receiver calibration. For information on power calibration, refer to Power Calibration.

1. Press **Cal** key.
2. Press **Receiver Calibration**.
3. Select a port (see Selecting target port for error correction).
4. Click **Source Port.**
5. Select the stimulus port for which you want to perform power calibration.
6. Connect the selected stimulus port by cable to any port you want to calibrate.



e5071c921

7. Click **Receiver Calibration** > **Power Offset** to enter the offset value. Receiver calibration is done at the condition of (source power)+(Specified offset).

   For example, if the source Power is -10 dBm, then:

- If specified offset is 0 dBm, then Source Power Reading after calibration would be -10 dBm.
- If specified offset is 10 dBm, then Source Power Reading after calibration would be 0 dBm.
- If specified offset is -10 dBm, then Source Power Reading after calibration would be -20 dBm.

NOTE The value of offset is independent for each channel.

8. The E5071C provides an option to calibrate both the Receiver and Transmitter end, or only the Receiver end, or only the Transmitter end:
   - Click **Receiver Calibration** > **Calibrate Both** to start measurement of calibration data for both receiver and transmitter end. This step allows the gain of the receiver to be calibrated, turning on the error correction function automatically.
9. From Firmware revision 9.2, **Take Cal Sweep** softkey is renamed as the **Calibrate Both** softkey.

- Click **Receiver Calibration** > **Calibrate R** to calibrate the receiver end only.

NOTE When **Calibrate R** is selected, the **Select Port** setting is used and the **Source Port** setting is ignored.

- Click **Receiver Calibration** > **Calibrate T** to calibrate the transmitter end only.

NOTE When **Calibrate Both** or **Calibrate T** is selected, the source port should be different from measurement port.

Power Calibration

Other topics about Advanced Calibration

Overview

The E5071C has a calibration feature for power level output that uses the power meter (power calibration).

The power calibration function outputs a stimulus signal with a more accurate power level (closer to the set value) by measuring calibration data (power level) in advance with the power meter and sensor. Then this function performs error correction of the power level by using the calibration data.

Turning Power Level Error Correction ON/OFF

Power calibration data are acquired for each channel/test port, and you can turn ON/OFF the power level error correction independently for any channel or test port.

The status of the power level error correction of each channel is indicated by one of the symbols shown in table below in the channel status bar in the lower part of the window.

Symbol	Status of power level error correction
PC (displayed in blue)	Error correction is performed for all stimulus ports
PC (displayed in gray)	Error correction is performed for some stimulus ports
PC? (displayed in blue)	Error correction is performed for all stimulus ports; interpolated calibration data is used. If you turn on the error correction function when the stimulus setting is different from that when the power calibration data were acquired, power level error correction is performed with interpolated calibration data only when calibration data can be interpolated (extrapolation is not performed for calibration data).
PC? (displayed in gray)	Error correction is performed for some stimulus ports; interpolated calibration data is used. If you turn on the

	error correction function when the stimulus setting is different from that when the power calibration data were acquired, power level error correction is performed with interpolated calibration data only when calibration data can be interpolated (extrapolation is not performed for calibration data).
--- (displayed in gray)	Error correction is not performed (some stimulus ports have error correction set to on, but there is no valid calibration data)
None	Error correction is not performed (error correction is off for all stimulus ports)

Procedure to turn ON/OFF power level error correction

Power level error correction is automatically turned on when you execute the measurement of calibration data. You can turn it on or off as necessary by using the following procedure.

1. Press **Cal** key.
2. Click **Power Calibration.**
3. Select a port.
4. Click **Correction.** Each press toggles this selection between ON/OFF.

Preparing Power Meter and Sensor

To execute power calibration, you need to prepare the power meter and power sensor used to acquire power calibration data.

NOTE Power calibration may not be executed when the specified frequency and level is out of range of the power sensor specification.

NOTE Power meter is controlled via USB/GPIB connector or USB (for USB enabled Power Meters)

The table below shows available power meters and recommended power sensors for power calibration.

Product Description	Model Number	Supported ENA Firmware Revision
Power meters	Agilent 437B, 438A, E4418A/B, E4419A/B, N1911A/N1912A (w/ USB/GPIB interface)	All
	HP EPM-441A/442A, EPM-P E4416A/E4417A (w/ USB/GPIB interface)	

	Agilent N1911A/N1912A (USB to USB direct connection)	A.09.2x and above
	Agilent N1913A/N1914A EPM Series	A.11.0x and above
Power sensors	Agilent 8482A, E4412A, N1921A	All
	HP ECP-E18A	
	Agilent N1922A, E4413A, E9300A, E9301A, E9304A, E9304A-H18/H20, E9300H, E9300H-H19, E9301H, E9300B, E9301B	A.09.1x and above
	Agilent U2000A/B/H, U2001A/B/H, U2002A/H, U2004A	A.09.2x and above

Preparing to control the power meter

When acquiring power calibration data, the power meter is controlled via GPIB or USB (for USB enabled Power Meters) from the E5071C.

To control the power meter through GPIB from the E5071C, connect the USB port of the E5071C and the GPIB/USB connector of the power meter through the USB/GPIB interface and set the GPIB address of the connected power meter with the E5071C.

NOTE The USB/GPIB interface and USB enabled power sensor must be ready to use. When you connect them for the first time, new hardware wizard runs. For the procedure, refer to Setting system controller (USB/GPIB interface). The hardware wizard procedure for USB enabled power sensor is same. At the last step in the procedure, Assign USB device alias dialog box will be displayed. Select "When a new USB device is plugged in" or "Never show this dialog" instead of "Each time a USB devices is plugged in", or ENA sometimes fail to recognize the USB power sensor.

For detailed information, refer to "Hardware Installation and Configuration", page 15 to 19 of USB Power sensor operation manual, U2000-90405 November 14, 2008 3rd edition.

NOTE Check whether the USB power sensor is properly installed by referring to *"Hardware Installation and Configuration"* of USB Power sensor operation manual, U2000-90405. If you face trouble while connecting the USB power sensor for the first time, or just after reconnecting, try to restart the ENA firmware and re-connect the USB power sensor again.

Selecting the power meter

Use the following procedure to select the type of GPIB or USB enabled power meter:

1. Press **System** key.
2. Click **Misc Setup** > **Power Meter Setup** > **Select type** > **GPIB|USB**
   3. From Firmware revision 9.2, E5071C supports USB enabled power sensors.

Using GPIB controlled power meter

Use the following procedure to set the GPIB address of the power meter:

1. Press **System** key.
2. Click **Misc Setup** > **GPIB Setup** > **Power Meter Address**.
3. Enter the GPIB address of the power meter you are using.
   4. You can also set the GPIB address of power meter by pressing **System** > **Misc Setup** > **Power Meter Setup** > **GPIB Address**

Setting power sensor calibration factor table

NOTE Before using the power sensor calibration factor table of the E5071C, set the calibration factor to 100% and then calibrate the power sensor.

When you use the 437B or 438A as the power meter, you need to set the power sensor calibration factor table with the E5071C.

If you use a power meter other than the 437B or 438A, refer to the following table.

Power sensor	Setting of calibration factor table
8482A	Set the calibration factor table with the E5071C only when you do not set the calibration factor table with the power meter. If you set the calibration factor table with both the power meter and the E5071C, calibration is executed by both of them and you cannot obtain correct measurement results.
E4412A ECP-E18A	You do not need to set the calibration factor table with the E5071C. Even if you set the calibration factor table with the E5071C, this setting would be ignored.

NOTE If you use the E4418A, E4419A, EPM-441A, or EPM-442A with Firmware revision Ax.02.00 or earlier and you set the calibration factor table with the E5071C, calibration is executed by both the power meter and the E5071C due to this Firmware revision, and thus you cannot obtain correct measurement results. Therefore, never set the calibration factor table under this condition with the E5071C.

Use the following procedure to set the power sensor's calibration factor table:

1. Press **Cal** key.
2. Click **Power Calibration**.
3. Click **Sensor A Settings** (for power sensor connected to channel A) or **Sensor B Settings** (for power sensor connected to channel B).
4. Click **Ref Cal Factor**.
5. Enter the reference calibration factor (the calibration factor at 50 MHz).
6. According to the calibration factor data attached to the power sensor, set the frequency (**Frequency**) and the coefficient (**Factor**) of the calibration factor table by using the hardkeys and softkeys.
    7. Pressing **Preset** > **OK** does not affect the current setting of the reference calibration factor and the calibration factor table.
    8. For a frequency other than one set in the table, a value obtained by linear interpolation of the calibration factors at the 2 points adjacent to the frequency is used. If a frequency is lower than the lowest frequency in the table, the calibration factor at the lowest frequency is used; if larger than the highest frequency in the table, the calibration factor at the highest frequency is used.
    9. When setting the table by using the front panel keys or the keyboard, you need to first set the focus on (select) the operation target (table or softkey). You can change the focus by pressing **Focus** key in the ENTRY block. When the focus is placed on the table, the window frame of the table is displayed as bright as the window frame of the active channel. When the focus is placed on the softkey menu, the softkey menu title area is displayed in blue.

Saving power sensor calibration factor table

You can save the power sensor calibration factor table as a CSV (Comma Separated Value) format file.

1. Press **Cal** key.
2. Click **Power Calibration**.
3. Click **Sensor A Settings** or **Sensor B Settings**.

4. Click **Export to CSV File** to open the Save As dialog box. At this time, CSV File (extension *.csv) is selected as the file type.
5. Enter a file name in the **File Name** box and press the **Save** button to save the power sensor calibration factor table.

Recalling power sensor calibration factor table

By recalling a power sensor calibration factor table saved in the CSV format according to saving power sensor calibration factor table, you can set the power sensor calibration factor table.

1. Press **Cal** key.
2. Click **Power Calibration**.
3. Click **Sensor A Settings** or **Sensor B Settings**.
4. Click **Import from CSV File** to open the Open dialog box. At this time, CSV File (extension*.csv) is selected as the file type.
5. Select the CSV format file you want to import and press the **Open** button to recall the power sensor calibration factor table.
    6. This operation is not guaranteed under the following two conditions:
    1) you imported a CSV format file created/edited on a spreadsheet program, or
    2) you imported a CSV format file that had been exported according to Saving power sensor calibration factor table but then modified.

Using USB power sensor

Use the following procedure to select the USB power meter:

1. Press **System** key.
2. Click **Misc Setup** > **Power Meter Setup** > **USB**.
3. Select desired USB power sensor.
    4. When USB enabled power sensor is plugged to ENA, it will take about 30 seconds to be ready. If you perform calibration or zeroing before the ENA is ready, an error will occur.

Performing Zeroing/Calibration

Zeroing a power sensor is performed in order to reduce zero measurement offset and noise impact to improve the accuracy of RF power measurement. The U2000 Series USB power sensors have two types of zeroing INTernal zeroing and EXTernal zeroing. For more information, refer to http://cp.literature.agilent.com/litweb/pdf/U2000-90405.pdf.

Use the following procedure to perform zeroing/calibration:

1. Press **Cal** key.
2. Click **Power Calibration** > **Sensor A/B Settings** > **Zero Type** > **INT|EXT** to select from internal or external zeroing.
3. Click **Power Calibration** > **Sensor A/B Settings** > **Zero/Calibrate Sensor**
    4. For U2000 Series USB power sensors, Sensor B setting is disabled.

Selecting Target Port of Error Correction

The power level error correction is executed for each channel/test port. You can set the following items for each channel/test port:

- ON/OFF of error correction
- Setting of loss compensation
- Selection of the power sensor
- Number of power level measurements at one measurement point
- Calibration data

Follow the steps below to select the test port for which you want to set/execute power level error correction.

1. Press **Cal** key.
2. Click **Power Calibration.**
3. Press **Channel Next**/**Channel Prev** keys to select the channel.
4. Click **Select Port.**
5. Click the softkey corresponding to the test port you want to select.

Setting Loss Compensation

If you need to apply a signal with a more accurate power level to the DUT, you can use the loss compensation feature to obtain calibration data for correcting the difference in power loss due to the difference in connection method (cable, adapter, etc.) between the time when the power calibration data are measured and when the actual DUT is measured.

The loss compensation feature corrects the power measurement result based on the preset power loss data. More specifically, if you turn on the loss compensation, the measurement result of the power level obtained in the power calibration data measurement is a value obtained by adding the loss value set in the loss compensation table to the measured power value.

Turning ON/OFF loss compensation

Follow the steps below to turn ON/OFF the loss compensation.

1. Press **Cal** key.
2. Click **Power Calibration**.

3. Select a port (refer to Selecting target port of error correction).
4. Click **Loss Compen** > **Compensation**. Each press toggles the selection between ON/OFF.

Creating loss compensation table

Follow the steps below to set the loss compensation table.

1. Press **Cal** key.
2. Click **Power Calibration.**
3. Select a port (refer to Selecting target port of error correction).
4. Click **Loss Compen**
5. Set the frequency (**Frequency**) and the loss (**Loss**) of the loss compensation table by using the hardkeys and softkeys.
    1. Pressing **Preset** > **OK** does not affect the current setting of the reference calibration factor and the calibration factor table.
    2. For a frequency other than one set in the table, a value obtained by linear interpolation of the calibration factors at the 2 points adjacent to the frequency is used. If a frequency is lower than the lowest frequency in the table, the calibration factor at the lowest frequency is used; if larger than the highest frequency in the table, the calibration factor at the highest frequency is used.
    3. When setting the table by using the front panel keys or the keyboard, you need to first set the focus on (select) the operation target (table or softkey). You can change the focus by pressing **Focus** key in the ENTRY block. When the focus is placed on the table, the window frame of the table is displayed as bright as the window frame of the active channel. When the focus is placed on the softkey menu, the softkey menu title area is displayed in blue.

Saving loss compensation table

You can save the loss compensation table as a CSV (Comma Separated Value) format file.

1. Press **Cal** key.
2. Click **Power Calibration.**
3. Select a port (refer to Selecting target port of error correction).
4. Click **Loss Compen**.

5. Click **Export to CSV File** to open the Save As dialog box. At this time, CSV File (extension*.csv) is selected as the file type.
6. Enter a file name in the **File Name** box and press the **Save** button to save the loss compensation table.

Recalling loss compensation table

By recalling a loss compensation table saved in the CSV format according to the Saving loss compensation table, you can set this loss compensation table.

    1. It is possible to recall a file from a different channel/port from where it was saved.

1. Press **Cal** key.
2. Click **Power Calibration.**
3. Select a port (refer to Selecting target port of error correction).
4. Click **Loss Compen**.
5. Click **Import from CSV File** to open the Open dialog box. At this time, CSV File (extension*.csv) is selected as the file type.
6. Select the CSV format file you want to import and press the **Open** button to recall the loss compensation table.

    7. This operation is not guaranteed under the following two conditions: 1) you imported a CSV format file created/edited on a spreadsheet program, or 2) you imported a CSV format file that had been exported according to Saving loss compensation table but then modified.

Setting Tolerance for Power Calibration

The tolerance value sets the target power tolerance for the power calibration. Follow the steps below to set the tolerance for power calibration:

1. Press **Cal** key.
2. Click **Power Calibration** > **Tolerance**.
3. Enter the tolerance value for power calibration.

Setting Max Iteration

When you want to make a power calibration with higher accuracy, use the Iteration function. This function repeats the cycle of updating the power calibration coefficient and measuring the power level until the Max Iteration number at each stimulus point is reached.

On the other hand, Num of Readings just repeats the power measurement and averages the measurement results without updating the power calibration coefficient. This capability should be used only when the measurement of the power sensor is unstable due to some circumstances.

- When the max iteration is set to 0, no iteration is performed. The output power level is measured and the power calibration coefficient is set by comparing the targeted power level and the actually measured power level at each stimulus point. If the measured power level is out of the tolerance, the power calibration process is aborted with an error message.
- When the max iteration is 1 and above, the power measurement is re-performed after the initial power measurement and updating the power calibration coefficient. And if the measured power level is within the tolerance, the analyzer goes to the next stimulus point to be calibrated. If the measured power level is out of the tolerance, the analyzer repeats the cycle of updating the calibration coefficient and measuring the power level until the max iteration number. If the power measurement value is out of the tolerance even after the max iteration, the power calibration process is aborted with an error message.
- Max Iteration is independent of measurement type. It corrects the E5071C source regardless of which receivers are being used in a measurement. Therefore, it can be used with both ratio or non-ratio measurements.
- Applies only to those measurements on the selected channel that use the test port specified as the source for the calibration. For example, if you specify Channel 1 and Port 1 as the source to be calibrated, only those measurements on channel 1 that use port 1 as the source will be corrected.
- Can be used in conjunction with other measurement calibrations, such as a full 2-port calibration. For highest accuracy, perform the measurement calibration after the source calibration.
- Can be used with Power Sweep type. Source Power Cal corrects the power at all power levels across the power sweep.
- Can also be used with Port Power Uncoupled.

To set the maximum iteration, follow the steps below:

1. Press **Cal** > **Power Calibration** > **Maximum Iteration**, then set the maximum iteration value.

Measuring Calibration Data

NOTE Before measuring calibration data, you need to execute the zero adjustment and calibration of the power sensor. For information on how to execute these operations, see the manual of the power meter you are using.

When using the power sensor calibration factor table of the E5071C, set the calibration factor to 100% and execute the calibration of the power sensor.

1. Press **Cal** key.
2. Click **Power Calibration**.
3. Select a port (refer to Selecting target port of error correction).
4. Click **Use Sensor**. Each press toggles between channel A and channel B. If you use a power meter with one channel, channel A is always selected.
5. Click **Tolerance** and enter the tolerance value (for example, 0.1 dB).
6. Click **Max Iteration** and enter the max iteration number (for example, 5).
7. Connect the power sensor for the selected channel to the selected port.
8. Click **Take Cal Sweep** to start the measurement of calibration data.
9. You can abort the measurement by pressing **Abort** during measurement.
10. When the measurement is complete, the power level error correction is automatically turned on.
11. If the power meter GPIB address is not set correctly or if the power sensor is not connected to the specified channel, an error occurs and calibration data are not measured.

NOTE If appropriate calibration data cannot be obtained, an error may occur for each sweep after the measurement of calibration data is complete. In this event, turn off the power level error correction, check the connection and setting, and then measure the calibration data again.

Partial Overwrite Calibration

- Overview
- Procedure

Other topics about Advanced Calibration

Overview

The partial overwrite function is used to perform partial measurement after the execution of calibration, and it overwrites the calibration coefficients.

There are three types of calibration coefficients: Er, Es, Ed for reflection, Et for transmission, and Ex for isolation. If some of them do not provide satisfactory calibration, you can use this function to re-calculate the calibration coefficients by measuring an applicable standard only instead of measuring all standards again.

> NOTE When the calibration coefficients become inappropriate over time or the status on the E5071C side from the calibration surface changes due to replacement of a cable or connector, you need to also perform thru measurement when partial overwrite is required for reflection or isolation measurement.

> NOTE The adapter removal and partial overwrite function is only available when calibration status is [Corr] and not for [C?] or [C!].
>
> Partial overwrite is not available if no calibration has been done. You cannot append calibration coefficients to previous calibrations. For example, you cannot realize full 4-port calibration by performing additional calibration for 1 port after the execution of full 3-port calibration. The partial overwrite function is used to make measurements for previous calibration coefficients and overwrite them.

Procedure

Follow these steps to execute the partial overwrite function. The example demonstrates re-calibration for full 2-port thru calibration.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **2-Port Cal** > **Select Ports**.
4. Select the test ports for which you want to perform re-calibration for full 2-port calibration. (In the following procedure, the test ports you select are expressed as test ports x and y.)

5. Click **Transmission.**
6. Make a thru connection between selected test ports x and y (between the connectors to which the DUT is connected).
7. Click **Port x-y Thru** (**x** and **y** are the thru-connected port numbers) to start the re-measurement of the calibration standard.
8. Click **Return**.
9. Click **Overwrite** to finish the re-calibration for the full 2-port calibration. At this point, the calibration coefficients are re-calculated and saved.

.

Simplified Calibration

- Overview
- Simplified Full 3/4-port Calibration
- Simplified 3/4-port TRL Calibration

Other topics about Advanced Calibration

Overview

The simplified calibration calculates the calibration coefficients by skipping part of thru measurement (and line measurement for TRL measurement) that is necessary for the full 3/4-port calibration and the 3/4-port TRL calibration.

Simplified Full 3/4-port Calibration

In the simplified full 3/4-port calibration, since the calibration coefficients are calculated while omitting part of the thru measurement data, the effect of errors when acquiring calibration data becomes larger than in the normal full 3/4-port calibration. Therefore, you must pay special attention to the following points when measuring data for the simplified full 3/4-port calibration.

- The standard used for measurement must match its definition value.
    - Use a standard that provides good repeatability (stability).
    - Do not omit the length of the thru when defining the standard.
    - When using a user-created standard, verify the definition value.
    - For the N connector, note the discrimination between male and female.
- Realize high reliability and repeatability for measurement.
    - Reduce the difference in external environment (such as temperature difference) between conditions of measuring calibration data and conditions of measuring actual data.
    - Set the power level of the stimulus signal to a value that does not generate compression.
    - Narrow the IF bandwidth.
    - Increase the averaging factor.
    - Use a cable whose change in amplitude/phase characteristics when bent is small.
    - Use high-precision connectors.

The types of thru measurement you can omit are determined by the ports you select. Some types of thru measurement cannot be omitted. The table below shows the types of thru measurement that can be omitted.

Omissible Thru Type at Full 3-port Calibration

Ports used	Types of thru measurement that can be omitted
Ports 1, 2, 3	2-3
Ports 1, 2, 4	2-4
Ports 1, 3, 4	1-4
Ports 2, 3, 4	2-4

For the simplified full 3-port calibration, (Optional) is displayed on the softkey of the type of thru measurement that can be omitted. For example, when the omissible thru measurement is 2-3, **2-3 (Optional)** is displayed. The display after the execution of the omissible thru measurement is the same as that for a required thru measurement.

Omissible Thru Type at Full 4-port Calibration

Required thru measurements	Types of thru measurement that can be omitted
1-2, 1-3, 3-4	1-4, 2-3, 2-4

For the simplified full 4-port calibration, (Recommended) is displayed on the softkey of the type of thru measurement that can be omitted. For example, when the omissible thru measurement is 2-3, **2-3 (Recommended)** is displayed. The display after the execution of the omissible thru measurement is the same as that for a required thru measurement, and the display for the remaining omissible thru measurement is **2-3 (Optional)**.

For the simplified full 4-port calibration, up to three thru measurements can be omitted, but it is recommended to perform two or more types of calibration because omitting all of them results in poor accuracy.

Simplified 3/4-port TRL Calibration

In the simplified 3/4-port TRL calibration, because the calibration coefficients are calculated while omitting part of the thru/line measurement data and line/match measurement data, the effect of errors when acquiring calibration data becomes larger than in the normal 3/4-port TRL calibration. Therefore, you must give more attention to data measurement for the simplified full 3/4-port calibration than for the ordinary 3/4-port TRL calibration.

The types of thru/line measurement and line/match measurement you can omit are determined by the ports you select. Some types of measurement cannot be omitted. You cannot omit thru/line measurement only or line/match measurement only, even for an omissible measurement path.

Omissible Thru type at 3-port TRL calibration

Ports used	Omissible measurement
Ports 1, 2, 3	2-3
Ports 1, 2, 4	2-4
Ports 1, 3, 4	1-4
Ports 2, 3, 4	2-4

For the simplified 3-port TRL calibration, (Optional) is displayed on the softkey of the omissible thru/line measurement. For example, when the omissible thru/line measurement is 2-3, **2-3 (Optional)** is displayed. The display after the execution of the omissible thru/line measurement is **2-3 Thru/Line**.

For the softkey of an omissible line/match measurement, (Optional) is displayed as in the case of the omissible thru/line measurement. For example, when the omissible line/match measurement is 2-3, **2-3 (Optional)** is displayed. The display after the execution of the omissible line/match measurement is **2-3 Line/Match**.

The softkeys (Fwd and Rvs) for forward measurement and reverse measurement for the omissible line/match measurement do not change.

Omissible measurements at 4-port TRL calibration

Required measurements	Omissible measurements
1-2, 1-3, 3-4	1-4, 2-3, 2-4

For the simplified 4-port TRL calibration, (Recommended) is displayed on the softkey of the omissible thru/line measurement. For example, when the omissible thru/line measurement is 2-3, **2-3 (Recommended)** is displayed. The display after the execution of the omissible thru/line measurement is **2-3 Thru/Line**, and the display for the remaining omissible thru/line measurement is **2-3 (Optional)**.

For the softkey of an omissible line/match measurement, (Recommended) is displayed as in the case of the omissible thru/line measurement. For example, when the omissible line/match measurement is 2-3, **2-3**

(Recommended) is displayed. The display after the execution of the omissible line/match measurement is **2-3 Line/Match**, and the display for the remaining omissible line/match measurement is **2-3 (Optional).**

The softkeys (Fwd and Rvs) for forward measurement and reverse measurement for the omissible line/match measurement do not change.

For the simplified 4-port TRL calibration, up to three thru/line measurements and line/match measurements can be omitted, but omitting all of them results in poor accuracy.

Adapter Removal-Insertion

- About Adapter Removal
- About Adapter Insertion
- About Adapter Waveguide & Rotate
- Performing Adapter Waveguide Calibration
- Procedure for Adapter Removal/Insertion
    - Adapter Removal
    - Adapter Insertion
- Difference between Traditional Network Analyzer & E5071C Adapter Calibration

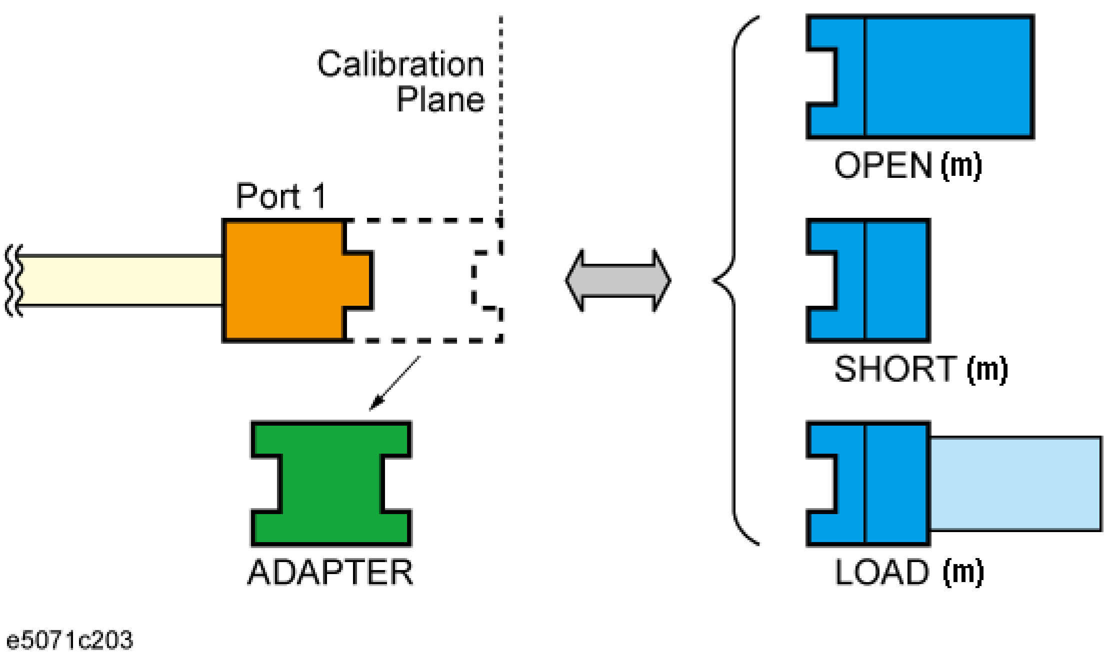
Other topics about Advanced Calibration

About Adapter Removal

Adapter Removal is a technique used to remove any adapter characteristics from the calibration plane. The E5071C uses the following adapter removal process to remove adapter characteristics:

1. Perform calibration with the adapter in use.
2. Remove the adapter from the port and measure Open, Short, and Load values to determine the adapter's characteristics.
3. Remove the obtained adapter characteristics from the error coefficients in a de-embedding fashion.

Open, Short, and Load values measured with the adapter removed



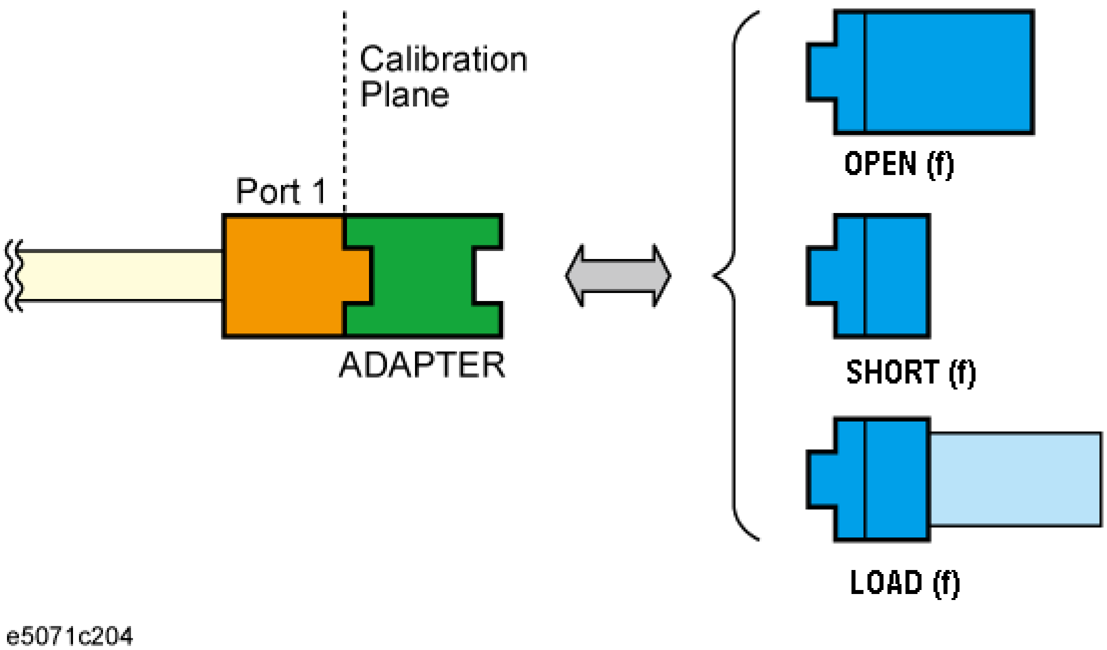
NOTE The adapter removal and partial overwrite function is only available when the calibration status is [**Corr**] and not for [**C?**] or [**C!**].

About Adapter Insertion

This above described method also makes it possible to add adapter characteristics to a port with n-port full calibration. This allows you to make a measurement with the adapter. E5071C uses the following adapter insertion process to insert adapter characteristics:

1. Perform calibration without adapter in use.
2. Insert the adapter to the port and measure Open, Short, and Load values to determine the adapter's characteristics.
3. Insert the obtained adapter characteristics to the error coefficients in an embedding fashion.

Open, Short, and Load values measured with adapter attached



In order to determine the adapter characteristics (with four unknown parameters) by making three measurements (Open, Short, and Load), the adapter must satisfy the following requirements:

1. Adapter must be Reciprocal (with S21 and S12 equal) in nature. It should have a consistent behavior, independent of the direction from which it is used.
2. The electrical length of Adapter should be known at an accuracy of ±1 /4 of wavelength.

About Adapter Waveguide & Rotate

From firmware A.09.10, functionality related to adapter waveguide and adapter rotation is added. Adapter waveguide length and Cut off frequency can be set using the following commands:

SCPI.SENSe(Ch).CORRection.COLLect.ADAPter(Pt).WAVeguide.LENGth

SCPI.SENSe(Ch).CORRection.COLLect.ADAPter(Pt).WAVeguide.CUToff

Coaxial length can be set using **Cal** > **Calibrate** > **Adapter Removal** > **Coaxial Length**

Rotate command executes Adapter Removal/Insertion along with moving the phase of adapter (which is removed or inserted) to 180 degrees. This command is useful in cases where auto judgement of phase fails. This command can be executed several times while the calibration remains valid.

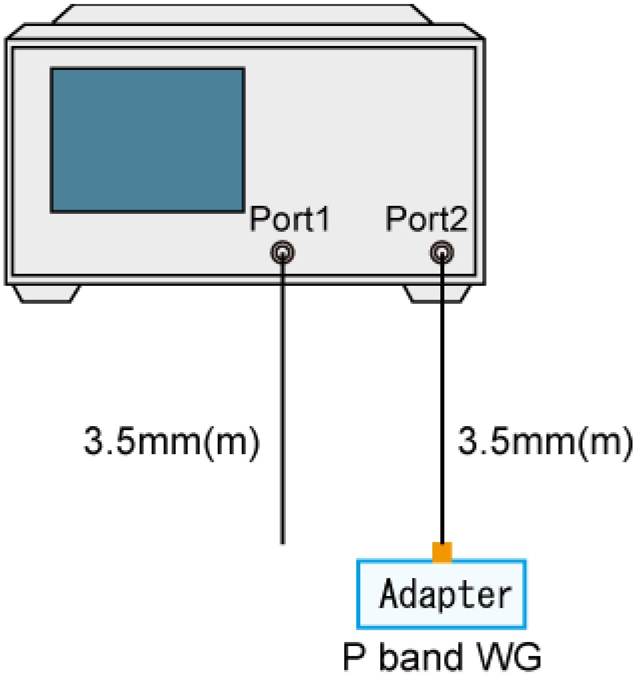
SCPI.SENSe(Ch).CORRection.COLLect.ADAPter(Pt).ROTate

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

NOTE Some calibration kits such as the waveguide calibration kit have operational frequency range defined by Minimum and Maximum frequency. When the E5071C stimulus setting is out of the operational frequency range, you cannot click **Done** key or finish the calibration by remote control. In this case, use a calibration kit that has proper frequency range, or change the E5071C stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5071C with **Cal** > **Modify Cal Kit** > **Define STDs** for the Maximum & Minimum frequency of the calibration kit.

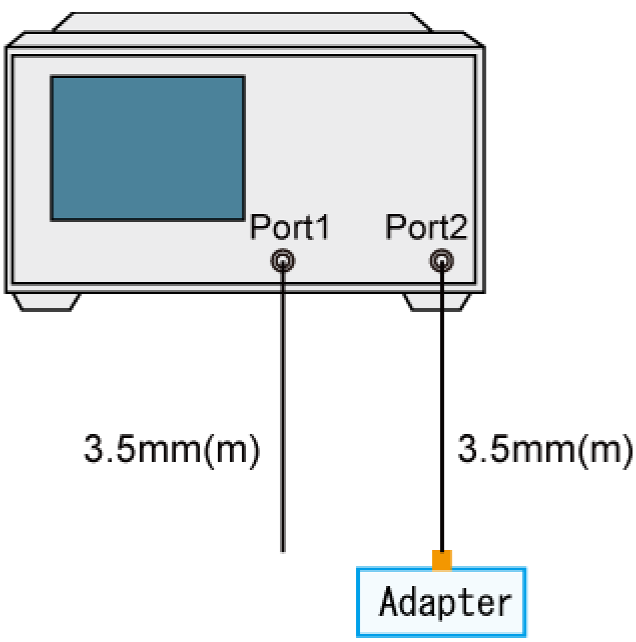
Performing Adapter Waveguide Calibration

This section provides an example of performing adapter waveguide calibration to perform a Full 2-port calibration plane between Port 1 (3.5mm) & Port 2 (P-band wave guide).



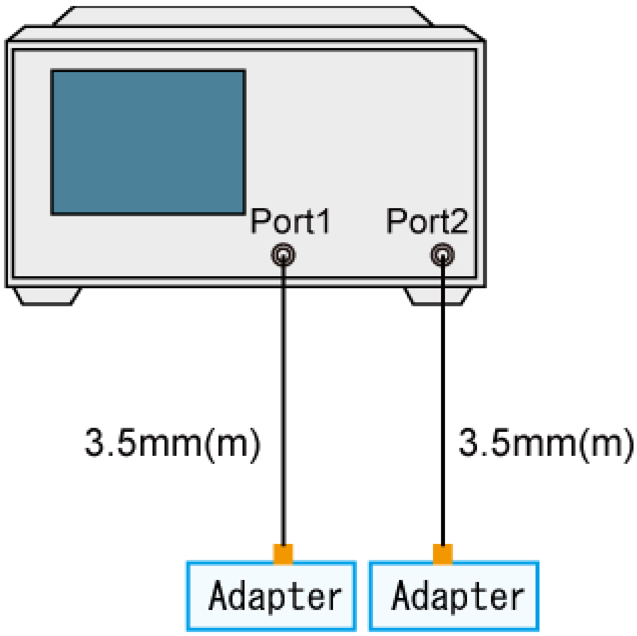
e5071c488

1. Using Adapter Insertion:
    a. Perform a SOLT F2P Calibration at Port 1 (3.5mm) using Z0=50 ohm
    b. Perform Adapter insertion at Port 2 (Z0=1) using P11644A



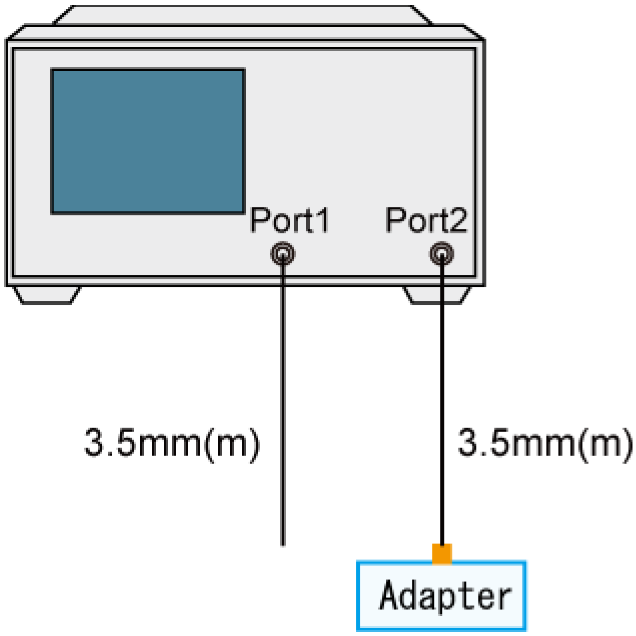
e5071c489

2. Using Adaptor Removal:
    a. Perform a SOLT/TRL F2P Calibration at P band WG (Z0 = 1)
    b. Perform Adapter removal at port 1 (Z0=50) using 85052D



e5071c490

3. Using Port Extension:
    a. Perform a SOLT F2P calibration at Port 1 (3.5mm) with Z0 = 50
    b. Perform Port extension calibration at Port 2 (manual with Coax length, WG length, Cut off). System Z0 can be either 1 oh or 50 ohm; both are fine to use.



e5071c491

Procedure for Adapter Removal/Insertion

Example scenario where adapter removal/insertion is required: When the DUT is an SMA (female) -to-Type N (male) whereas the test ports are two SMA (male) cables. In such cases, an adapter of Type N (female)-to-SMA (female) is required to perform adapter removal/insertion.

The S parameter of a reciprocal adapter can be determined when the following data is available:

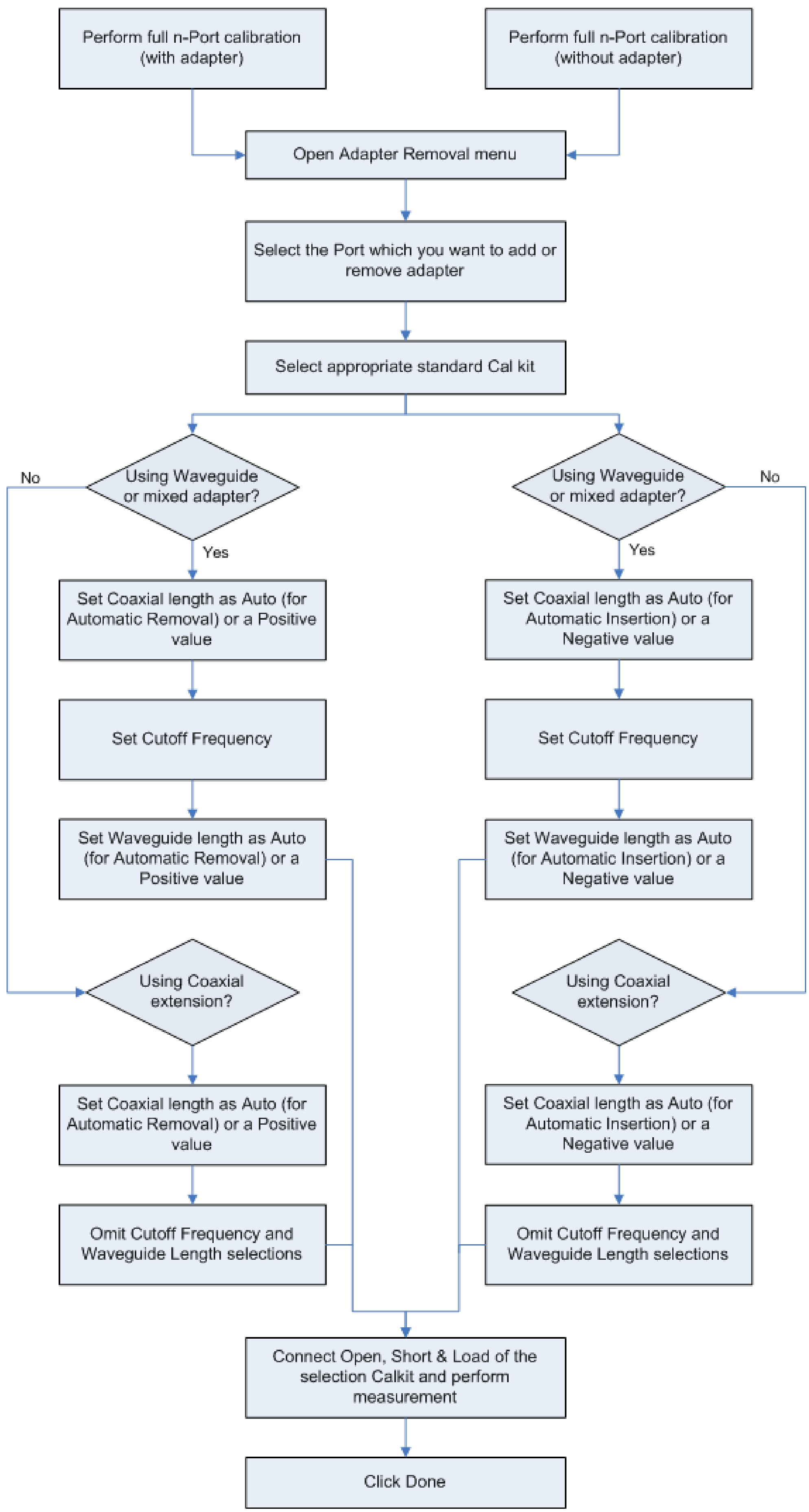
1. Open, Short, and Load measurements.
2. Actual values derived from the **CalKit** definitions.
3. An approximate length of the adapter.
4. Nature of the intended operation: removal or insertion.

NOTE If the frequency setting is not Zero Span, an approximate length of the adapter should be provided otherwise an error might be generated in the measurement.

NOTE For Adapter Removal, the adapter length is (**+**) Positive. For Adapter Insertion, the Adapter length is (-) Negative.

NOTE To change gender of the calibration kit, refer to Defining Calibration Kit Gender.

The diagram below shows summary of the adapter removal and insertion procedure:


Perform full n-Port calibration (with adapter)
Perform full n-Port calibration (without adapter)
Open Adapter Removal menu
Select the Port which you want to add or remove adapter
Select appropriate standard Cal kit
Using Waveguide or mixed adapter?
No
Yes
Using Waveguide or mixed adapter?
No
Yes
Set Coaxial length as Auto (for Automatic Removal) or a Positive value
Set Coaxial length as Auto (for Automatic Insertion) or a Negative value
Set Cutoff Frequency
Set Cutoff Frequency
Set Waveguide length as Auto (for Automatic Removal) or a Positive value
Set Waveguide length as Auto (for Automatic Insertion) or a Negative value
Using Coaxial extension?
Using Coaxial extension?
Set Coaxial length as Auto (for Automatic Removal) or a Positive value
Set Coaxial length as Auto (for Automatic Insertion) or a Negative value
Omit Cutoff Frequency and Waveguide Length selections
Omit Cutoff Frequency and Waveguide Length selections
Connect Open, Short & Load of the selection Calkit and perform measurement
Click Done

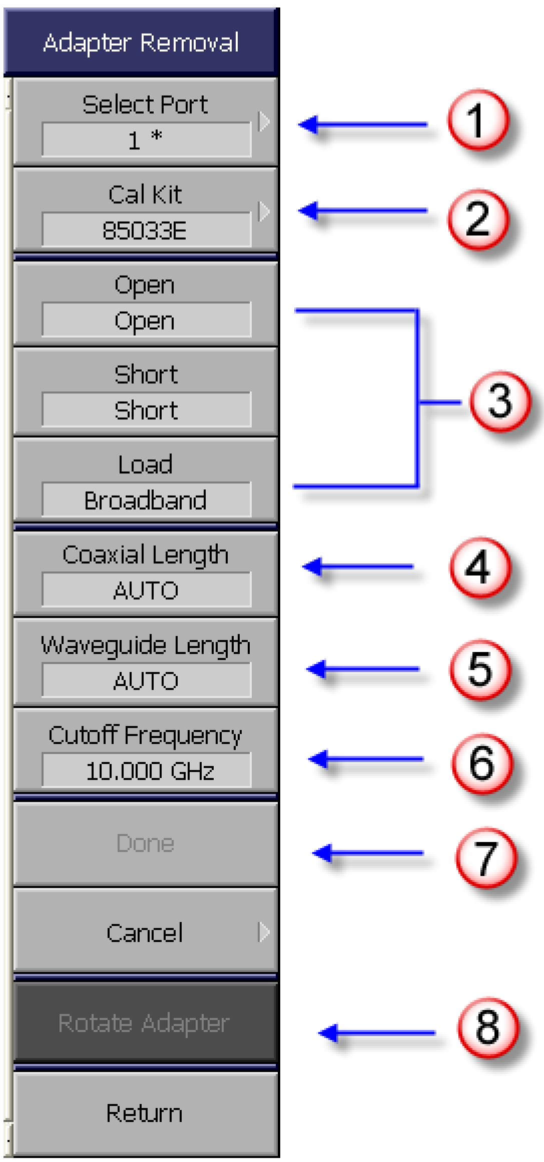
To use Adapter Removal:

1. Perform a full n-Port calibration using your calibration kit so that the port used to conduct adapter removal is calibrated.

   NOTE When you need to remove an adapter from the calibration plane (adapter removal) to connect your DUT, perform the calibration with an adapter so that you can make a calibration with your calibration kit.

2. Open the Adapter Removal menu:

   Cal > **Calibrate** > **Adapter Removal**



e5071c194

3. Select the Port from which you want to remove Adapter characteristics from. **Select Port** (e.g. 1) (**1** in the above figure). A * sign appears in front of the port indicates that this is a valid port to conduct adapter removal (as well as insertion) as the full n-port calibration has been performed on the port.
4. Select the proper standard Calkit you need to use from the options available at **CalKit** (e.g. 85033E) (**2** in the above figure) to characterize the adapter. The calibration kit is used at the plane from which adapter is removed.
5. If you are using a Waveguide or a mixed adapter:
   a. Set the **Coaxial length** as **Auto** for Automatic Removal or with a (+) Positive value.
   b. Select the **Cutoff Frequency** of the waveguide you want to enter (**6** in the above figure). The Cutoff Frequency is defined by the waveguide calibration kit in use.

      NOTE Without entering the value of **Cutoff Frequency**, the **Waveguide Length** cannot be entered.

   c. Set the **Waveguide Length** as **Auto** for Automatic Removal or if you want to input a manual Waveguide Length, type a value with a (**+**) Positive value, which indicates removal.

      NOTE When using waveguide calibration kit and measuring waveguide devices, ensure that you change the system Z0 to 1 ohm prior to calibration.

If you are using Coaxial extension:

   a. Set the **Coaxial length** as **Auto** for Automatic Removal or with a (+) Positive value.
   b. Omit the **Cutoff Frequency** and **Waveguide Length** selection.

6. Connect Open, Short, and Load of the selected **Calkit** (e.g. 85033E) with the selected port respectively and select **Open**, **Short**, and **Load** respectively. E5071C measures the cal kit standard, calculates the adapter characteristics, and then conducts adapter removal.

   NOTE A checkmark appears on the (**Open**, **Short**, and **Load**) menu after each type of calibration is completed, respectively.

7. Click **Done** to complete the process (**7** in the above figure).

   NOTE Some calibration kits such as the waveguide calibration kit have operational frequency range defined by Minimum and Maximum frequency. When the E5071C stimulus setting is out of the operational frequency range, you cannot click **Done** or finish the calibration by remote control. In this case, use a calibration kit that has proper frequency range, or change the E5071C stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5071C with **Cal** > **Modify Cal Kit** > **Define STDs** for the Maximum & Minimum frequency of the calibration kit.

NOTE In cases where auto judgement of phase fails, select **Rotate Adapter** to move the phase of adapter (which is removed) to 180 degrees.

To use Adapter Insertion:

1. Perform a full n-Port calibration using your calibration kit so that the port used to conduct adapter insertion is calibrated.

   NOTE When you need to add an adapter into the calibration plane (adapter insertion) to connect your DUT, perform the calibration without an adapter.

2. Open the Adapter Removal menu:

   Cal > **Calibrate** > **Adapter Removal**

3. Select the Port in which you want to insert the Adapter characteristics from **Select Port** (e.g. 1) (**1** in the above figure. A * sign appears in front of the port is a valid port to conduct adapter insertion (as well as removal) as the full n-port calibration has been performed on the port.
4. Select the proper standard Calkit you need to use from the options available at **CalKit** (e.g. 85033E) (**2** in the above figure) to characterize the adapter. The calibration kit is used at the plane in which adapter is inserted.
5. If you are using a Waveguide or a mixed adapter:

   a. Set the **Coaxial length** as **Auto** for Automatic Insertion or with a (-) Negative value.
   b. Select the **Cutoff Frequency** of the waveguide you want to enter (**6** in the above figure). The Cutoff Frequency is defined by the waveguide calibration kit in use.

      NOTE Without entering the value of **Cutoff Frequency**, the **Waveguide Length** cannot be entered.

   c. Set the **Waveguide Length** as **Auto** for Automatic Insertion or if you want to input a manual Waveguide Length, type a value with a (**-**) Negative value, which indicates insertion.

NOTE When using waveguide calibration kit and measuring waveguide devices, ensure that you change the system Z0 to 1 ohm prior to calibration.

If you are using Coaxial extension:

a. Set the **Coaxial length** as **Auto** for Automatic Insertion or with a (-) Negative value.

b. Omit the **Cutoff Frequency** and **Waveguide Length** selection.

6. Connect Open, Short, and Load of the selected **Calkit** (e.g. 85033E)with the selected port respectively and select **Open**, **Short**, and **Load** respectively. ENA measures the cal kit standard, calculates the adapter characteristics, and then conducts adapter insertion.

   NOTE A checkmark appears in (**Open**, **Short**, and **Load**) menu after each type of calibration is completed, respectively.

7. Click **Done** to complete the process (**7** in the above figure).

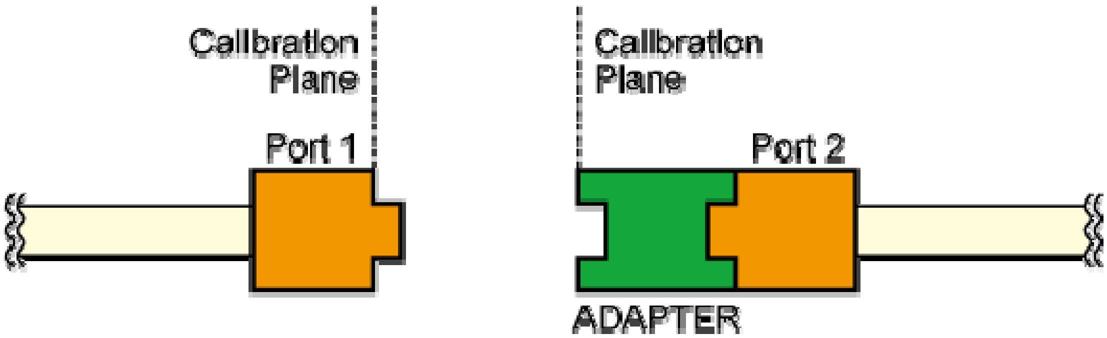
   NOTE Some calibration kits such as the waveguide calibration kit have operational frequency range defined by Minimum and Maximum frequency. When the E5071C stimulus setting is out of the operational frequency range, you cannot click **Done** or finish the calibration by remote control. In this case, use a calibration kit that has proper frequency range, or change the E5071C stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5071C with **Cal** > **Modify Cal Kit** > **Define STDs** for the Maximum & Minimum frequency of the calibration kit.

   NOTE In cases where auto judgement of phase fails, select **Rotate Adapter** to move the phase of adapter (which is inserted) to 180 degrees.

Difference between Traditional Network Analyzer & E5071C Adapter Calibration

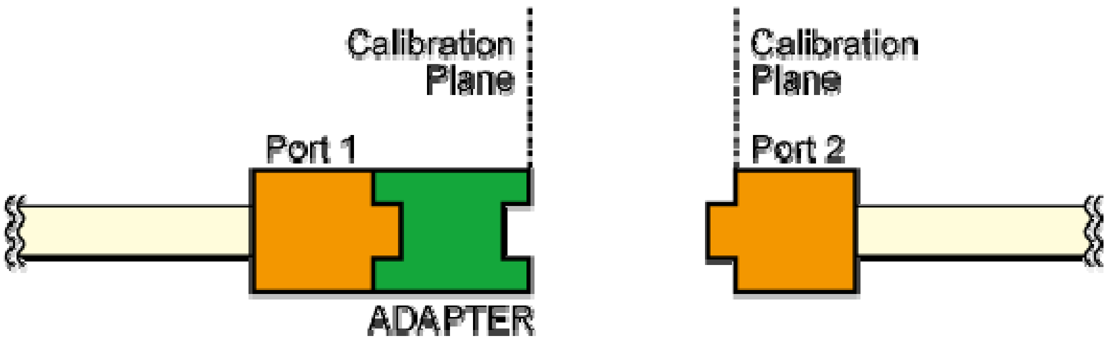
Usually, two-port network analyzers removes adapter characteristics by performing two sets of Full 2 Port Calibration as shown below:

Calibration performed with the adapter connected to Port2



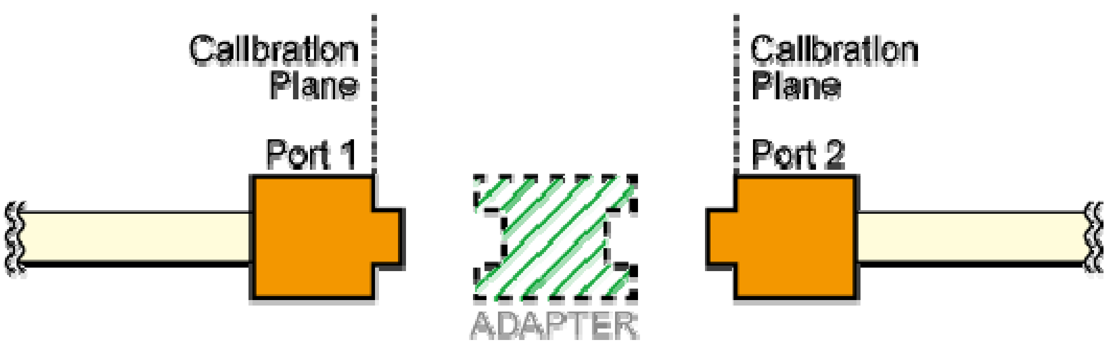
e5071c200

Calibration performed with the adapter connected to Port1



e5071c201

Removing adapter characteristics using two (above) sets of calibration



e5071c202

However, this method is not suitable for a multi-port network analyzer because it will require as many sets of Full 2 Port Calibration as twice the number of port combinations. Therefore, the E5071C uses an advanced method to remove Adapter characteristics described in About Adapter Insertion.

Unknown Thru Calibration

- Overview
- Procedure

Other topics about Advanced Calibration

Overview

Unknown Thru Cal is the preferred THRU method of calibrating the ENA to measure a non-insertable device. The major benefits of using a unknown Thru calibration are:

- It is easy to perform.
- Provides better accuracy than Defined Thru and is usually better than Adapter Removal.
- Does not rely on the 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 treated as the THRU standard.

The Thru must satisfy the following requirements:

1. Thru must be Reciprocal (with S21 and S12 equal) in nature. It should have a consistent behavior, independent of the direction from which it is used.
2. The electrical length of the Thru should be known at an accuracy of ±1 /4 of the wavelength.
3. Unknown Thru is NOT supported during the TRL calibration.

Procedure

To measure the nominal electrical delay of the Unknown Thru standard:

1. With port 1 configured, perform a simple S11 response calibration at port 1 by using a short circuit

   Press **Cal** > **Calibrate** > **Response (Short)** > **Short**
2. Connect the Unknown Thru standard to port 1 and terminate the standard with a short circuit.
3. Measure the electrical delay of the Unknown Thru standard

   Press **Cal** > **Port Extensions** > **Auto Port Extension** > **Measure SHORT** > **Port 1**

   The electrical delay is automatically measured and can be confirmed.

Press **Cal** > **Port Extensions** > **Extension Port 1** {Value}

4. If there is an offset in the short circuit used to terminate the Unknown Thru standard, remember to subtract its electrical length from the measurement result.

To perform the unknown Thru Calibration, follow the procedure below:

1. Press **Cal** key, then click **Calkit.**
2. Select the desired calibration kit (e.g. 85033D or User Kit)
3. Press **Cal** > **Modify Calkit** > **Define STDs** > **No Name** > **Label**
4. Type a name for unknown Thru Cal standard e.g. {Unknown Thru1}
   - By changing the label to a different name (e.g. Unknown Thru1), the 'No Name' standard will adopt to the specified new name.
5. Press **Cal** > **Modify Calkit** > **Define STDs** > **{Defined Name}** > **STD Type** > **Unknown Thru**
6. Click **Offset Delay**, then enter the approximate offset delay value of the unknown Thru standard.
7. Press **Cal** > **Modify Calkit** > **Specify CLSs** > **Thru** > **Set All** > **{Defined Name}**

6. Different Unknown Thru can be used for different ports.
7. Press **Cal** > **Calibrate** > **n-Port Cal** (n>1) > **Transmission** > **{Defined Name}** to execute mechanical Unknown Thru calibration.
8. Complete Reflection, and Isolation (optional) calibration for n full port Cal.

Performing 8 term Calibration using External PC

- Overview
- Procedure
- Example

Other topics about Advanced Calibration

Overview

To perform 8-term error model calibration using an external PC (measuring raw measurement data using the E5071C, calculating error term using external PC and then performing correction with calculated error terms and measured raw data), the procedure described in this section must be followed. When you use this method, the measurement specification is not satisfied.

NOTE In most cases, users do NOT need to follow the below procedure as these considerations have already been performed on 8-term error model calibration in E5071C.

NOTE When users perform 12-term error model calibration using external computer such as SOLT calibration, users do NOT need to take following procedure.

Procedure

Turning off the "System Calibration"

In its default setting, the E5071C applies "system calibration" to the measured S-parameter data. Because the "system calibration" is applied only to the S-parameter measurement data and NOT to the absolute value measurement data, it interferes with 8-term error model calibration when calculating error coefficients on external computer. You need to turn OFF this state, all the time for calibration and measurement.

The system calibration can be turned off by executing SCPI.SYSTem.CORRection.STATe

NOTE Once the E5071C receives this command, it will keep this setting until preset. This state is saved in the state file.

Turning off the "Virtual Bridge" (3 GHz, 4.5 GHz, 6.5 GHz and 8.5 GHz option only)

There is a software conversion function called "virtual bridge" on E5071C. The default setting of this state is ON, and this state interferes with 8-term error model calibration when calculating error coefficients on external computer. You need to turn OFF this state, all the time for the calibration and the measurement.

NOTE This function only exist on frequency 3 GHz, 4.5 GHz, 6.5 GHz and 8.5 GHz options. It's not necessary to consider "virtual bridge" on E5070A/71A/70B/71B, and E5071C frequency 14 GHz and 20 GHz options.

The virtual bridge can be turned off by the command: SERVice:ADJust:VIRTual:BRIDge[:STATe] {ON|OFF|1|0}.

NOTE Since this command is under service menu, the "SVC" indicator is turned on when the ENA sets virtual bridge OFF.

NOTE Once the ENA receives this command, it will keep this setting until preset.

NOTE The state of "virtual bridge" is NOT saved in the state file, and the default state of the "virtual bridge" is ON. So you need to send the command to turn OFF the state, every time you recall the state file.

Set the "RF ranging mode" to MANual (3 GHz, 4.5 GHz, 6.5 GHz and 8.5 GHz option only)

There is a hardware function called "RF ranging mode" on E5071C. The default setting of this state is AUTO, and this state interferes with 8-term error model calibration when calculating error coefficients on external computer. You need to turn off the "RF ranging Auto mode".

The "RF ranging mode" can be changed by the command: SERVice:ADJust:SENSe[Ch]:SWEep:RF:RANGe:PORT[Pt]:AUTO {ON|OFF}. This command applies to each channel and measurement port.

NOTE This function only exist on frequency 3 GHz, 4.5 GHz, 6.5 GHz and 8.5 GHz options. It's not necessary to consider "virtual bridge" on E5070A/71A/70B/71B, and E5071C frequency 14 GHz and 20 GHz options.

NOTE Since this command is under service menu, the "SVC" indicator is turned on when the ENA sets RF range auto mode OFF. Once the ENA receives this command, it will keep this setting until preset.

NOTE The state of "RF ranging mode" is NOT saved in the state file, and the default state of the "RF ranging mode" is AUTO. So you need to send the command to set state to AUTO OFF, every time you recall the state file.

You need to keep the state of "RF range Auto Mode" as OFF, all the time for calibration and measurement using this method

NOTE Once the ENA receives above command, it will keep this setting until preset.

NOTE Since the "RF ranging" is hold during the measurement, the system dynamic range of the E5071C will degrade by 6dB, as a result of the 8-term calibration performed externally. However, there is NO dynamic range degrade by 8-term error model calibration with internal calibration of the ENA.

Example

Assume you perform 4-port TRL calibration on channel1, you need to send following command before calibration. Also, you need to keep this state all the time for calibration and measurement.

SYSTem:CORRection:STATe OFF 'system correction off

SERVice:ADJust:VIRTual:BRIDge:STATe OFF 'virtual bridge off

SERVice:ADJust:SENSe1:SWEep:RF:RANGe:PORT1:AUTO OFF 'RF ranging mode to manual on port1

SERVice:ADJust:SENSe1:SWEep:RF:RANGe:PORT2:AUTO OFF 'RF ranging mode to manual on port2

SERVice:ADJust:SENSe1:SWEep:RF:RANGe:PORT3:AUTO OFF 'RF ranging mode to manual on port3

SERVice:ADJust:SENSe1:SWEep:RF:RANGe:PORT4:AUTO OFF 'RF ranging mode to manual on port4

NOTE Once you preset the system, you need to resend all of the above commands.

NOTE Once you save and recall the state using this method, you need to resend all of the above commands except for system correction off.

Mixer Calibration

Scalar-Mixer Calibration

Other topics about Mixer Calibration

Overview

The E5071C has a scalar-mixer calibration function for measuring frequency conversion devices.

Scalar-mixer calibration allows you to measure the magnitude value and reflection parameter of the mixer's conversion loss with very high accuracy by performing calibration using calibration standards (OPEN/SHORT/LOAD) in combination as well as a power meter.

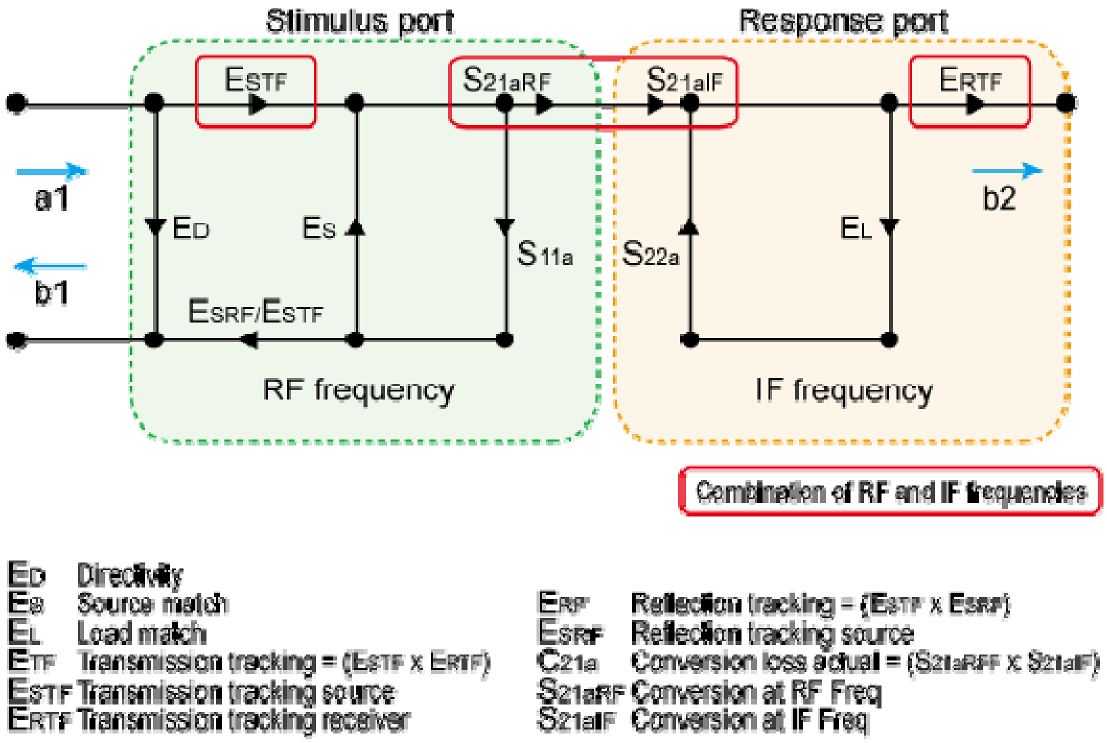
For measurement of the conversion loss in a frequency conversion device, normal full 2-port calibration is not available because of the frequency difference between the stimulus port and the response port. Therefore, scalar-mixer calibration allows you to correct the error term that resides in a full 2-port error model by using the error model and an expression based on a new concept.

You can correct the following error elements by using the scalar-mixer calibration.

- The reflection between the output port of the network analyzer and the input port of the measured mixer (vector error correction).
- The reflection between the output port of the measured mixer and the input port of the network analyzer (vector error correction).
- Transmission frequency characteristics at different frequencies (scalar error correction).

The frequency-offset error model can be described by the flow graph in the figure below. From the flow graph it can be seen that the model is divided in two halves: the stimulus port and the response port. By conceptualizing the error model in these two halves, each error term can be isolated to either of the model's halves depending on the frequency at which it was generated. The majority of the signals will only affect measurements by causing errors at the same frequency.

Two error signals that are functions of both halves of the frequency-offset model are isolation (EXF) and transmission tracking (ETF). EXF is set to zero because it is cannot be detected by the frequency change. ETF must be divided into two types of errors: one associated with the stimulus side at the input frequencies and the other associated with the response side at the output frequencies. As previously discussed, calculating the transmission tracking (ETF) based on both stimulus and response sides is the key to using error correction during frequency-offset measurements.



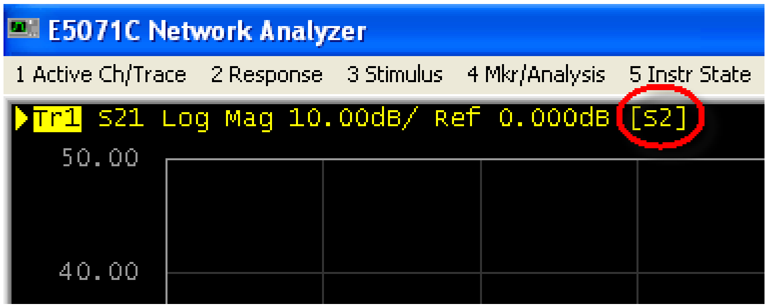
e5071c457

In scalar-mixer calibration, use the power meter to measure the transmission tracking error (ESTF) of the signal source in both the output and input frequencies in order to calculate the transmission tracking in the frequency-offset (ETF). The transmission tracking of the receiver in the output frequency (ERTF) can be determined by dividing the transmission tracking (ETF) calculated based on the output frequency in full 2-port calibration by the receiver's transmission tracking (ESTF). Multiplying the signal source transmission tracking (ESTF) by the receiver transmission tracking (ERTF) provides the transmission tracking in the frequency-offset (ETF). Since ESTF and ERTF are products of two power measurements, the resulting correction coefficient is a scalar parameter.

Confirming Calibration Status

Error correction status of each channel

You can confirm the progress of error correction for each channel by viewing the error calibration status. For the traces in which the scalar-mixer is valid, the status [S2] is added.

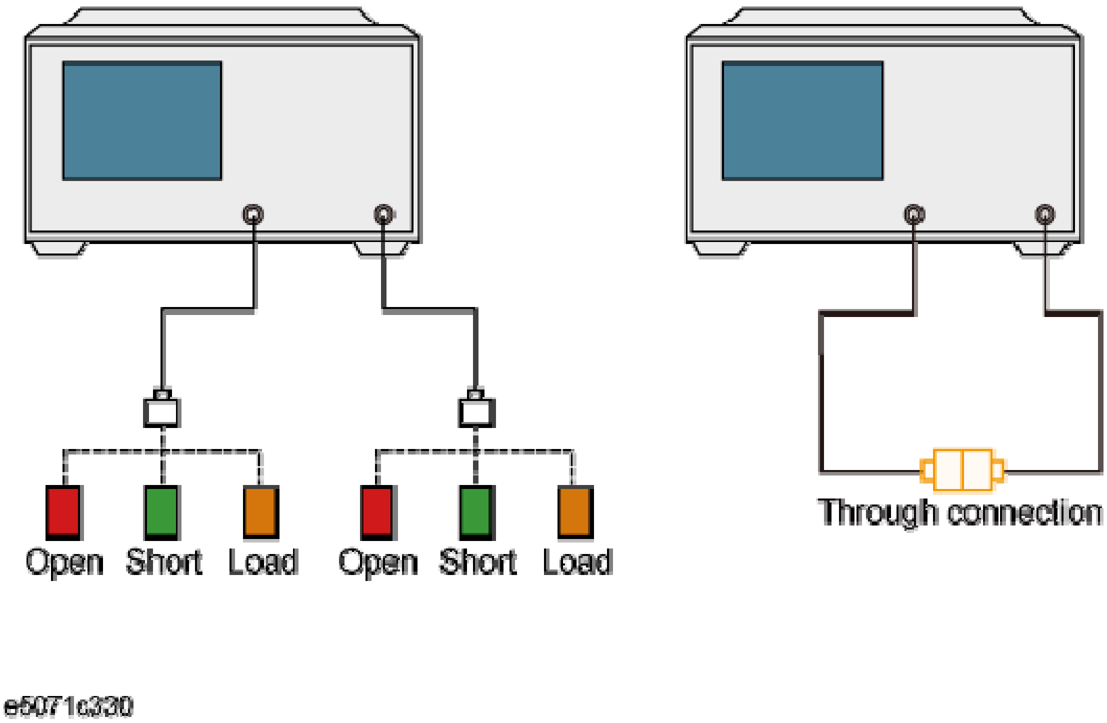


e5071c207

NOTE In scalar-mixer calibration, the normal calibration coefficient is invalid while the frequency-offset sweep is in progress; in this case the scalar-mixer calibration coefficient is used instead. Turning the frequency-offset sweep off switches over to the normal calibration coefficient; however, the information on the scalar-mixer calibration coefficient is retained.

Procedure Using Mechanical Calibration Kit

1. Before starting scalar-mixer calibration, verify that the frequency-offset is activated by clicking **Sweep Setup** key > **Frequency Offset**. If the frequency-offset is not valid, you cannot start scalar-mixer calibration.
2. Press **Channel Next**/**Channel Prev** keys to select the channel you want to calibrate.
3. Press **Cal** key.
4. Click **Mixer/Converter Calibration** > **Scalar Cal (Manual)** > **Select Ports**.
5. Select the test port for starting scalar-mixer calibration.



NOTE For the scalar-mixer calibration, only one direction with 2-ports calibration is available. "One direction" means the forward or reverse direction. Two directions may be used among the same ports, but this would not be full 2-port scalar-mixer calibration but simply the simultaneous operation of single-direction scalar-mixer calibrations.. .

Reflection

Both **x** and **y** in the following steps are provided for the case where you have selected in the test port selection.

6. Click **Reflection**.
7. Connect the OPEN calibration standard to the test port **x** (connector for the DUT).
8. Click **Port x @Freq x Open** to start measurement of the calibration standard.
9. Click **Port x @Freq y Open** to start measurement of the calibration standard.
10. Disconnect the OPEN calibration standard, then connect SHORT calibration standard in its place.

11. Click **Port x@Freq x Short** to start measurement of the calibration standard.
12. Click **Port x@Freq y Short** to start measurement of the calibration standard.
13. Disconnect the SHORT calibration standard, then connect LOAD calibration standard in its place.
14. Click **Port x@Freq x Load** to start measurement of the calibration standard.
15. Click **Port x@Freq y Load** to start measurement of the calibration standard.
16. Repeat the procedure for port y.
17. Click **Return**.

Transmission

18. Click **Transmission**.
19. Connect a THRU between the test ports x and y (between the connectors for the DUT).
20. Click **Port y-x@Freq y Thru** to start measurement of the calibration standard.
21. Click **Port x-y@Freq x Thru** to start measurement of the calibration standard.
22. Click **Return**.

Power Meter

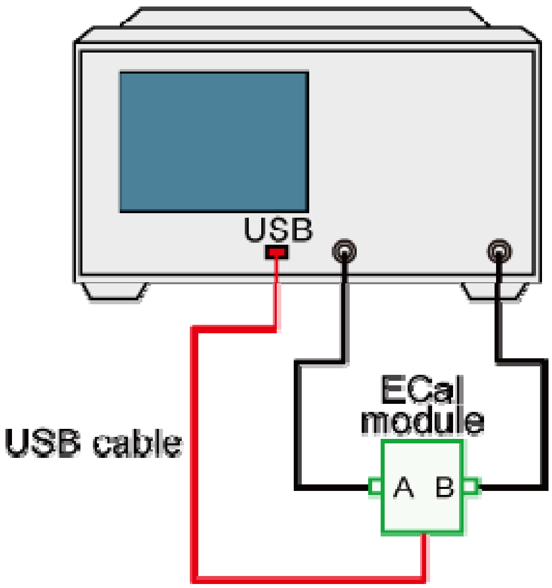
23. Click **Power Meter**. See Preparing to control the power meter for setup the power meter.
24. Click **Use Sensor**. Each time the key is pressed, Channels A and B switch over alternately. If you are using a one-channel power meter, select Channel A.
25. Connect the power sensor for the selected channel to the selected port.
26. Click **Port x@Freq x**.
27. Click **Port x@Freq y**.
28. Click **Port y@Freq x**.
29. Click **Port y@Freq y**.
30. Click **Return**.

Error Correction Done

31. Click **Done** to exit the scalar-mixer calibration. This step allows the calibration coefficient to be calculated, turning on the error correction function automatically.

Procedure Using ECal Kit

1. Before starting scalar-mixer calibration, verify that the frequency-offset is activated by clicking **Sweep Setup** key > **Frequency Offset**. If the frequency-offset is not valid, you cannot start scalar-mixer calibration.
2. Press **Channel Next**/**Channel Prev** keys to select the channel you want to calibrate.
3. Press **Cal** key.
4. Click **Mixer/Converter Calibration** > **Scalar Cal (ECal)** > **Select Ports**.
5. Select the test port for starting scalar-mixer calibration.
6. Click **Power Meter**. See Preparing to control the power meter for setup the power meter.
7. Click **Use Sensor**. Each time the key is pressed, Channels A and B switch over alternately. If you are using a one-channel power meter, select Channel A.
8. Connect the power sensor for the selected channel to the selected port.
9. Click **Port x@Freq x**.
10. Click **Port x@Freq y**.
11. Click **Port y@Freq x**.
12. Click **Port y@Freq y**.
13. Click **Return**.
14. Connect the test port you selected to the ECal module.



e5071c322

15. Click **ECal & Done**. The calibration coefficient will be calculated, turning on the error correction function automatically.

NOTE If the ECal module is not connected to the E5071C or power calibration have not been done yet, the **ECal & Done** menu item will not be not available. In addition, if the ECal module is not connected to the test port of the calibration target, an error may occur.

Vector-Mixer Calibration

Other topics about Mixer Calibration

Overview of Vector-Mixer Calibration

The E5071C has a vector-mixer calibration function for use in measuring frequency conversion devices.

The vector-mixer calibration allows you to measure the magnitude, phase and group delay of the mixer's conversion loss by using in combination calibration standards (OPEN/SHORT/LOAD) and calibration mixer with an IF filter, as well as the network de-embedding function incorporated in the E5071C.

NOTE For Fixed RF measurement (RF: Fixed, LO and IF: Swept), it is NOT possible to perform vector-mixer calibration because touchstone file, which defined with fixed frequency, can not be imported to ENA.

NOTE For Fixed IF measurement (RF and LO: Swept, IF: Fixed), it is possible to perform vector-mixer calibration and to measure conversion loss and return loss, but NOT possible to measure group delay because IF frequency is fixed.

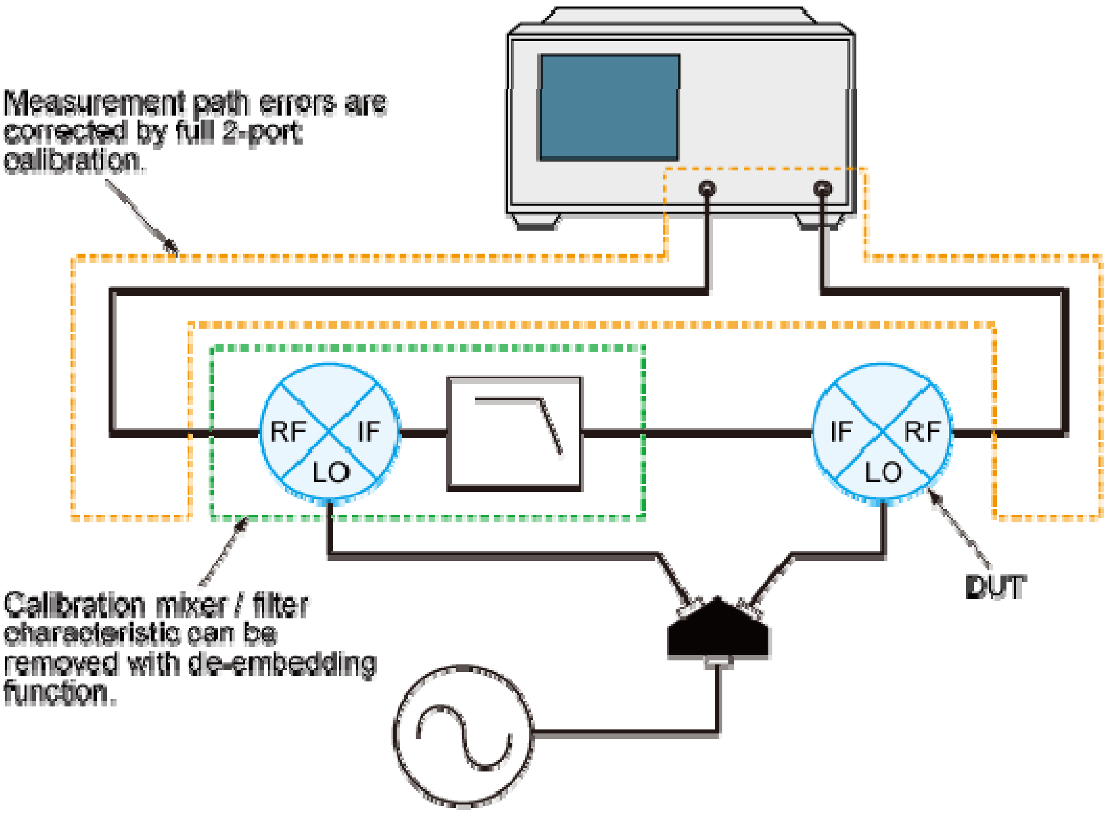
NOTE For Swept IF measurement (RF and IF: Swept, LO: Fixed), it is possible to perform vector-mixer calibration and to measure conversion loss, return loss and group delay.

You can also perform balanced mixer measurements by using two calibration mixers that each has an IF filter.

Vector-mixer calibration is implemented by eliminating the characteristics of the calibration mixer and IF filter by using the network de-embedding function after full 2-port calibration has been completed. Using the up/down conversion method allows you to specify the same sweep measurement frequency for the input and output ports, thus enabling full 2-port calibration at the end of the target port. Consequently, only the characteristics of the measured mixer (DUT) can be obtained by using the network de-embedding function, after eliminating the characteristics of the calibration mixer with an IF filter from all measurement results.

NOTE Since the up/down conversion method is used in vector-mixer calibration, the frequency-offset function is not used. But, option 008 is required to use the VMC macro, SG control, and etc.

Overview of Vector-Mixer Calibration



e5071c340

The vector-mixer calibration requires the characteristics data for the calibration mixer with IF filter.

Measured mixer

A measured mixer (DUT) signifies an unknown target mixer of measurement. However, a measured mixer meeting the requirements for a calibration mixer can be used as a calibration mixer.

Calibration mixer (with IF filter)

The calibration mixer is required for supporting the measurement system of the up/down conversion. You must also evaluate in advance the frequency response characteristics of the calibration mixer. The vector-mixer calibration method obtains the characteristics of the measured mixer alone by using the network de-embedding function to eliminate the characteristics of the calibration mixer from the measurement result. You can use the IF filter to select any required frequency conversion component such as RF+LO, RF-LO, and LO-RF. The calibration mixer and IF filter can be seen as a part of the test system setup, just like the network analyzer and the test cable; they are connected at the same location during the entire calibration or measurement.

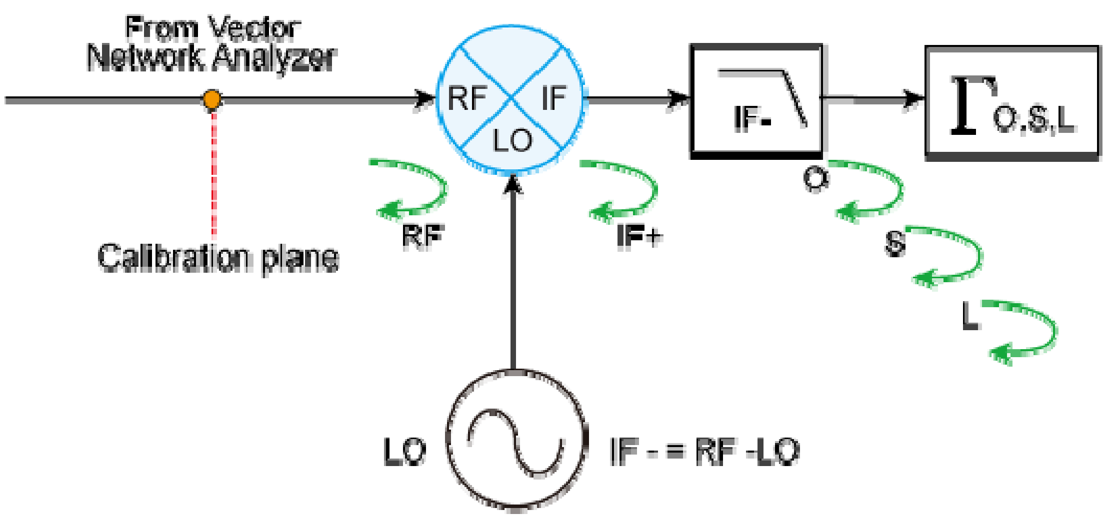
NOTE The frequency range must be equal to or greater than that of the measured mixer. If you want to test multiple mixers with a single setup, select a wide range of calibration mixers that can cover all frequencies of the target test devices.

Characterizing Calibration Mixer (with IF filter)

In vector-mixer calibration, you must characterize the calibration mixer with the IF filter. As shown in the following figure, connect the target mixer (with IF filter) to the port of the network analyzer on which vector calibration has been performed and then connect an OPEN, SHORT or LOAD standard to the end of the IF filter to start reflection measurement. The signals measured at the test port include the reflection signal from the mixer's RF port, the IF signal (IF+) converted by the mixer and then reflected by the IF filter, and the IF signal (IF-) passing through the IF filer and then reflected by the calibration standard.

The characteristics of the calibration mixer can be described in a 1-port error model, and each error item can be determined from Γ_O, Γ_S, and Γ_L, which are obtained in the reflection measurement of individual standards.

Characteristics evaluation of calibration mixer (with IF filter)



e5071c458

NOTE The calibration mixer must be reciprocal. The term "reciprocal" means the magnitude and phase of the conversion loss are equal both in the forward and reverse directions. The forward conversion loss occurs during the measurement of the output signal at the IF port while inputting measurement signals into the RF port. In contrast, the reverse conversion loss occurs during measurement of the output signal at the RF port while inputting measurement signals into the IF port.

NOTE For precise calibration, the conversion loss in each direction must be less than 10 dB using a calibration mixer and IF filter in combination. Exceeding 15 dB of the conversion loss in any direction may deteriorate the calibration accuracy significantly.

Characterizing procedure for calibration mixer (with IF filter)

The E5071C has a pre-installed VBA macro (Vector Mixer Characterization) for characterizing the calibration mixer with an IF filter.

Storage Folder	VBA Macro (Project Name)
D:\Agilent	**MixerCharacterization.vba**

Executing Characterization of Calibration Mixer

1. Setting Stimulus Conditions

Set the stimulus conditions for the channel you want to calibrate. For the necessary steps, refer to Setting Stimulus Conditions.
You must also set the external signal source in advance. For the necessary steps, refer to Setting External Signal Source.

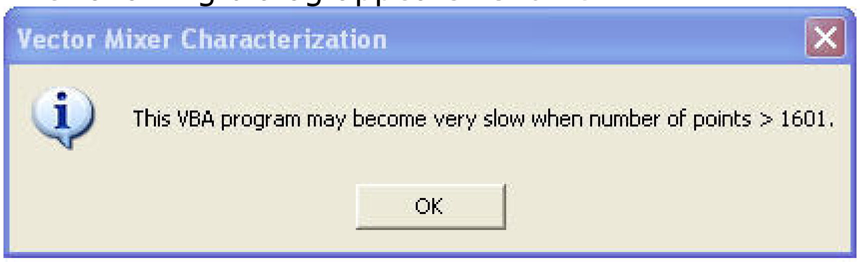
2. Running VBA Macro

NOTE Full 2-port calibration is recommended for characterizing the calibration mixer with the IF filter, although 1-port calibration is also available. This is because using full 2-port calibration simplifies the evaluation procedures, which are described in Overview of vector-mixer calibration . For detailed information on full 2-port calibration, see Full 2-Port Calibration.

Load the VBA project and then run it.

1. Press **Macro Setup**.
2. Click **Load & Project**.
3. Specify **D:\Agilent\ MixerCharacterization.vba** for the file name in the Open dialog box and then press the **Open** button.

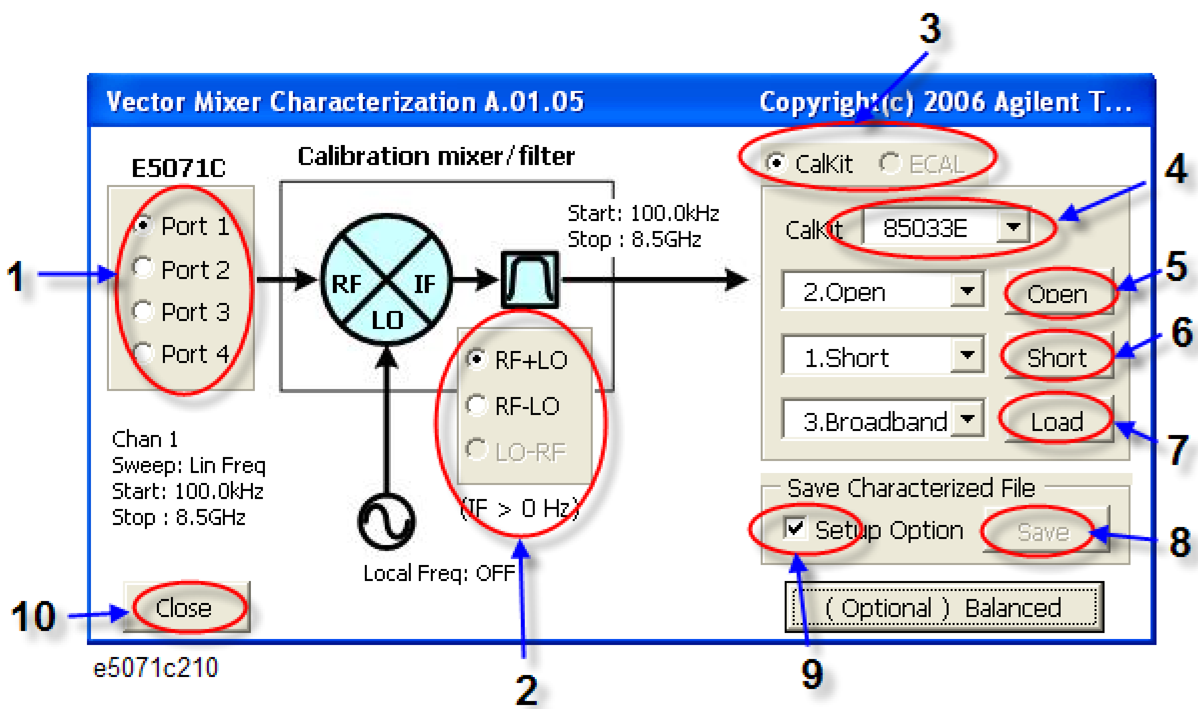
4. Press **Macro Run** to run the macro.
5. The following dialog appears. Click **OK**.



e5071c136

When more than 1601 measurement points is set for 1 channel and 4 traces, the E5071C VBA macro function may take more time to operate.

Vector Mixer Characterization Macro



e5071c210

3. Selecting Measurement Port

Select the 1-port calibration port (1 in menu).

NOTE Select any port if full 2-port calibration is set.

4. Setting IF Frequency

Select IF frequency from RF+LO, RF-LO and LO-RF (2 in menu), depending on the IF frequency of the calibration mixer.

NOTE The number displayed in the Vector Mixer Characterization macro is the frequency set in the E5071C and read from it. You must also set the minimum IF frequency at more than 0 kHz.
IF BW must be set to much smaller value than IF frequency.

5. Selecting a Calibration Kit

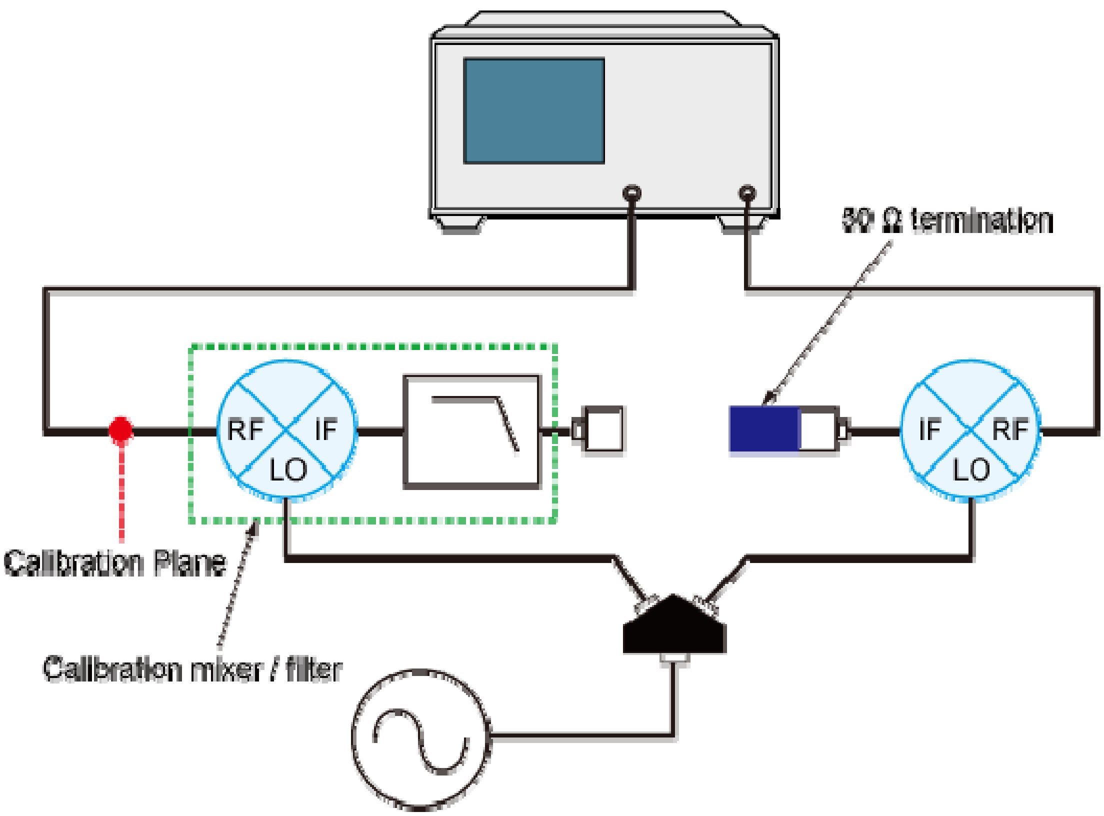
Select a calibration kit (3 in menu).

NOTE The mechanical calibration kit displayed in the Vector Mixer Characterization macro is the frequency registered in the E5071C and read from it. If an ECal module is connected to the E5071C, ECal will be selected automatically.

6. Measuring Calibration Mixer with IF Filter (when using calibration kit)

Connect the calibration mixer to one of the test ports on which 1-port calibration has been done, as shown in the following figure.

Connection of calibration mixer (with IF filter)



e5071c341

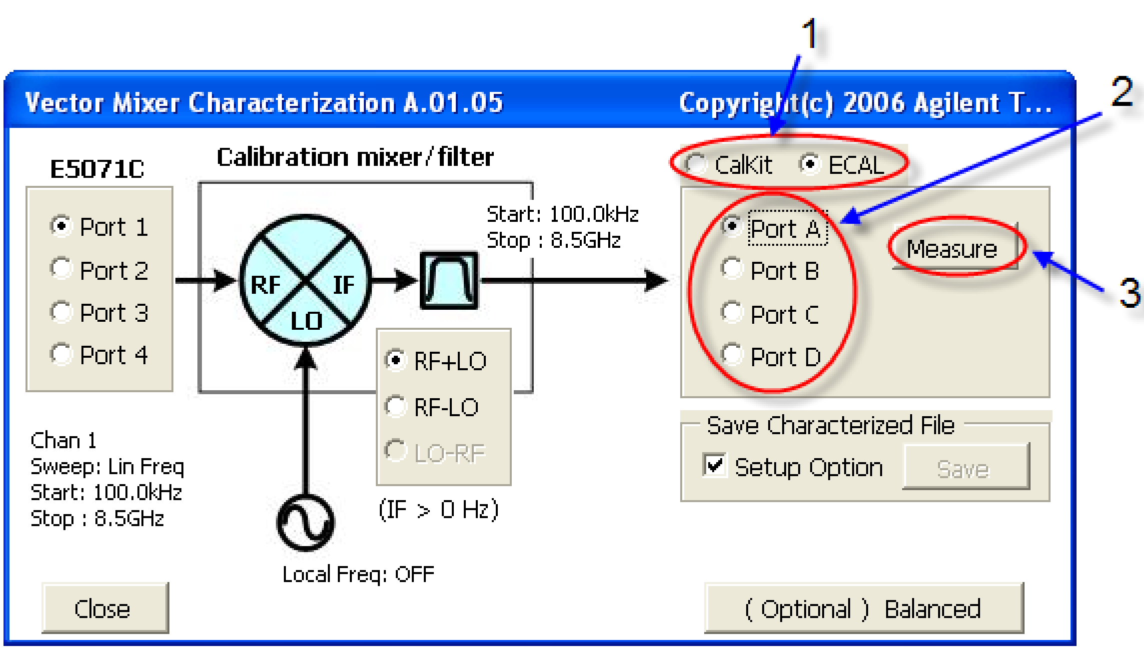
NOTE Select any port if full 2-port calibration is set.

NOTE We recommend that you characterize the calibration mixer with an IF filter when the power splitter for distributing the LO signal is connected to the measured mixer. In vector-mixer calibration, where the up/down conversion method is used, the power of the LO signal is distributed to the calibration mixer and the measured mixer through the power splitter. During a characteristics evaluation of the calibration mixer, the LO power level used by the drive of the calibration mixer must be equal to the LO power level with the measured mixer connected. This is because the mixer's conversion loss and reflection coefficient are significantly affected by the power level of the LO signal.

1. Select **CalKit** (3 in menu).
2. Select the type number of the calibration kit from **CalKit** menu (4 in menu).
3. Click the **Open** button (5 in menu) to start measurement in OPEN.
4. Click the **Short** button (6 in menu) to start measurement in SHORT.
5. Click the **Load** button (7 in menu) to start measurement in LOAD.

7. Measuring Calibration Mixer with IF Filter (when using ECal module)

Characterizing calibration mixer using ECal



e5071c211

1. Select **ECal** (1 in menu).
2. Select the port used for the ECal module (2 in menu).

3. Click the **Measure** button (3 in menu) to start measurement.

8. Saving Characteristic Data and Closing VBA Macro

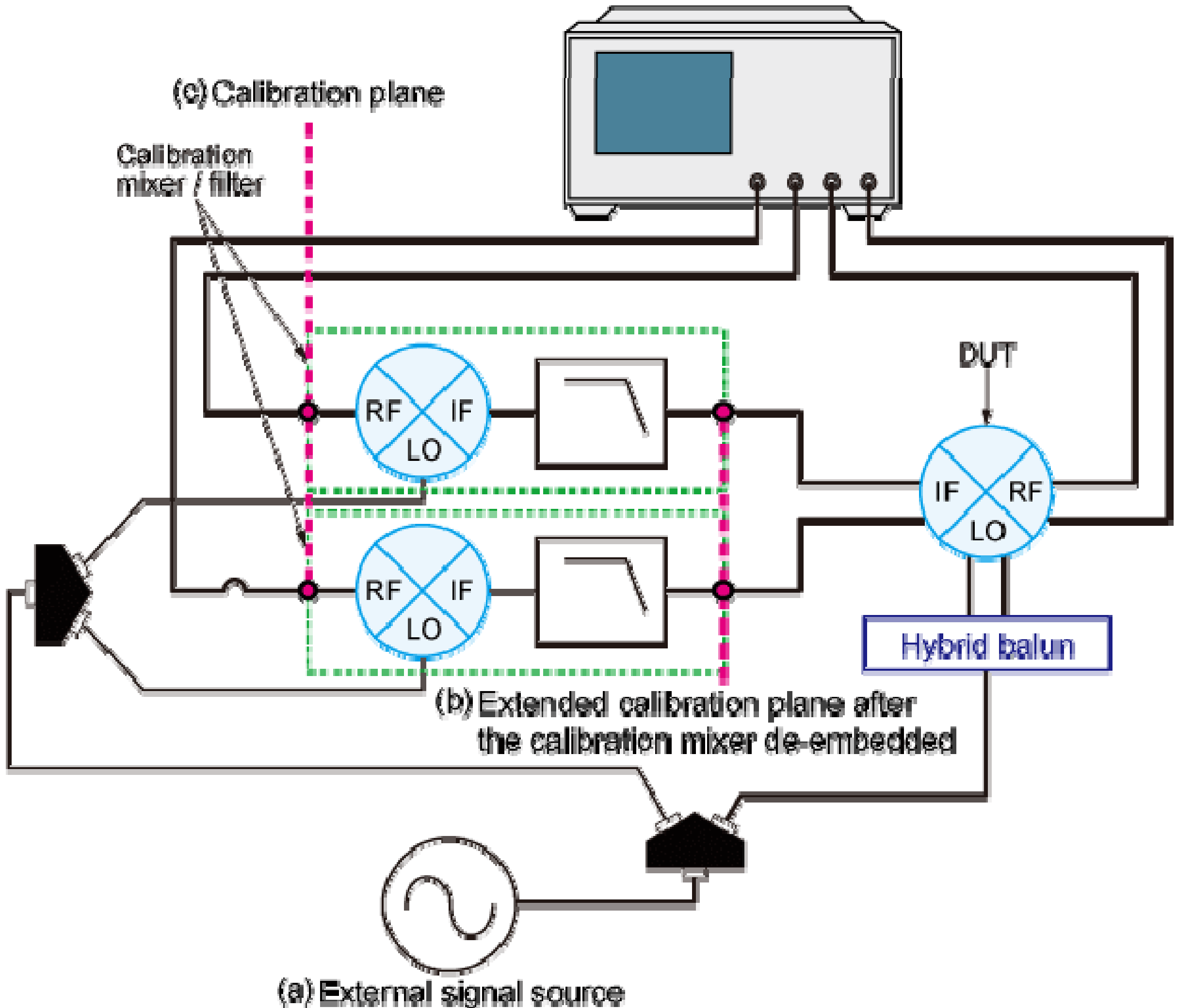
1. Press the Save button (8 in menu) to open the Save screen.
2. Press the Save button to specify a name for the characteristic data of the calibration mixer with IF filter. Then save it to a Touchstone file. If you check the Setup Option (9 in menu), the saved characteristic data will be set for the specified port of the active channel as the characteristic data file of the network de-embedding, and the fixture simulator function will be enabled. If unchecked, only the characteristic data will be saved.
3. Click the **Close** button (10 in menu) to exit the macro.

Characterizing Calibration Mixer (with IF filter) for Balance Mixer Measurement

The VBA macro (Vector Mixer Characterization) provided with the E5071C allows you to characterize the calibration mixer (with IF filter) to be used for the balanced mixer measurement. The characterizing procedures of the calibration mixer with IF filter used for balance mixer measurement are basically the same as those used for normal mixer measurement; however, two characteristic data of the calibration mixer with IF filter are required for balanced mixer measurement, as shown in the following figure.

Connect the target calibration mixer (with IF filter) to the port of the network analyzer on which calibration has been performed and then connect the OPEN, SHORT and LOAD standards to the end of the IF filter to start reflection measurement and characterization. For a balanced mixers, the phase difference of the LO signals between the calibration mixers with IF filter will remain as an error, since each calibration mixer with IF filter is characterized independently. Therefore, you must calibrate the phase difference between the two characterized calibration mixers with IF filters.

Characteristics evaluation of calibration mixer (with IF filter) for balance mixer



e5071c342

NOTE

We recommend that you use the same products of calibration mixers, IF filters and cables between the balanced ports to which each calibration mixer (with IF filter) is connected.

You should keep the following electrical lengths the same between the two ports to which the calibration mixer is connected as much as possible - the electrical length from the external signal source output port (a) to the extended calibration plane (b), and the electrical length from the calibration plane (c) to the extended calibration plane (b). Large electrical length differences between the two ports to which the calibration mixer is connected could raise 180 degrees phase value error between the two IF ports even though the Balanced Mixer Calibration Macro is executed. You can verify it by swapping the IF cable connections with each other.

Executing Characterization

1. Characterizing Calibration Mixer (with IF filter)

Measure the characteristic data of each calibration mixer with the IF filter used for balanced mixer measurement, using any two ports.

Set the stimulus conditions for the channel you want to calibrate. For the necessary steps, refer to Setting Stimulus Conditions.
You must also set the external signal source in advance. For the necessary steps, refer to Setting External Signal Source.

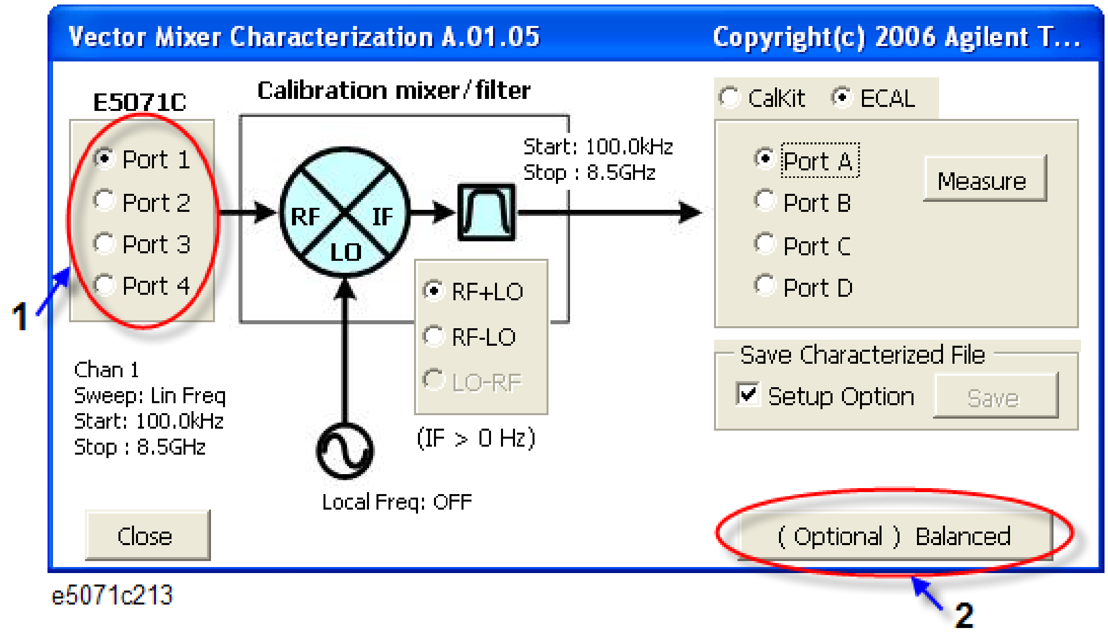
> NOTE If you characterize a calibration mixer with an IF filter, we recommend that you perform full 4-port calibration in advance, since it simplifies the evaluation procedures. For detailed information on full 4-port calibration, see Full 4-Port Calibration.

1. Press **Macro Setup** key.
2. Click **Load & Project**.
3. Specify "**D:\ Agilent\ MixerCharacterization.vba**" for the file name in the Open dialog box and then press the **Open** button.
4. Press **Macro Run** to run the macro.
5. Select **Port 1** (1 in menu) to characterize the calibration mixer 1 with IF filter. In this case, the data are saved to a temporary file (MIXER_1.s2p).
6. Select **Port 2** (1 in menu) to characterize calibration mixer 2 with IF filter. Here, the data are also saved to a temporary file (MIXER_2.s2p).

   > NOTE For detailed information on characterizing the calibration mixer, see Characterizing procedure for calibration mixer (with IF filter).

7. Click **(Optional) Balanced Mixer** (2 in menu).

Vector Mixer Characterization Macro

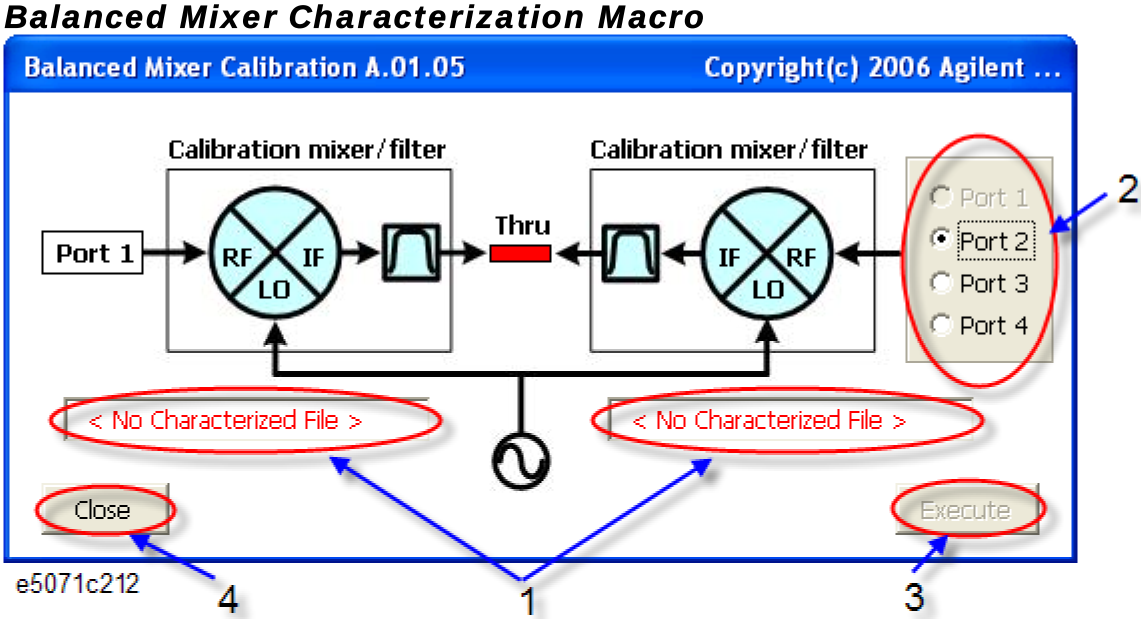


e5071c213

8. As the Vector Mixer Characterization Macro is running, the data files of the pre-measured calibration mixer with IF filter (MIXER_1.s2p, MIXER_2.s2p) are read automatically into the macro (1 in menu).

NOTE If failure occurs when reading the data file for the calibration mixer with IF filter, the characterization may have done by using only one port instead of using two ports.

Balanced Mixer Characterization Macro



e5071c212

1. Select the measurement port (2 in menu) and then connect a THRU between the IF ports of the calibration mixers to correct the phase difference of the LO signals for the calibration mixers with IF filters.

2. Pressing the **Execute** (3 in menu) button executes a phase error correction and overwrites the results on the original data file.

   NOTE You cannot run the **Execute** function when selecting the measurement port if the data file of the calibration mixer with IF filter (*.s2p) has not been set for the network removal function of the fixture simulator.

   NOTE The phase error correction data reflects the phase difference of the LO signals for the phase information of the calibration mixer's data file, which is registered in any two ports.

3. Press the **Close** button (4 in menu) to exit macro.

TRL Calibration

2-port TRL Calibration

- Overview
- Procedure

Other topics about TRL Calibration

Overview

The 2-port TRL calibration function lets you measure calibration data by connecting thru, reflection (open or short), line, or match calibration standards to (between) 2 desired test ports. This calibration provides the most accurate measurement for non-coaxial parts, using 12 error terms model in total for calibration in the same way as full 2-port calibration.

Before executing TRL calibration, you need to modify the calibration kit definition you use (or create a new one). For more information, see Changing the Calibration Kit Definition.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to perform the calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **2-Port TRL Cal**.
4. Click **Select Ports**, then select the test ports for which you want to execute TRL 2-port calibration. (In the following procedure, test ports you select here are expressed as test ports x and y.)
5. Connect the port.

Thru/Line

1. Make a THRU or Line connection between ports x and y (between the connectors to which the DUT is to be connected).
2. Click **Thru/Line**.
3. Click **Port x-y Thru** to execute a Thru/Line measurement.
4. Click **Return**.

Reflection

1. Click **Reflect**
2. Click **Port x Reflect** to execute a reflection calibration for test port x.
3. Click **Port y Reflect** to execute a reflection calibration for test port y.
4. Click **Return**.

Line/Match

1. Click **Line/Match**.
2. Click **softkey** to execute Line/Match calibration.

Softkey	Function
x-y Line/Match	Executes line/match calibration for test ports x and y. Use this to perform forward-direction measurement and reverse-direction measurement at the same time. This is equivalent to measuring both **x-y Fwd (Syx)** and **x-y Rvs (Sxy)**.
x-y Fwd (Syx)	Executes line/match calibration for test ports x and y. Use this to perform forward-direction measurement only.
x-y Rvs (Sxy)	Executes line/match calibration for test ports x and y. Use this to perform reverse-direction measurement only.

3.
4. Click **Return**.
5. Click **Done** to finish TRL 2-port calibration. At this point, the calibration coefficient is calculated and saved. The error correction function is automatically turned on.

3-port TRL Calibration

- Overview
- Procedure

Other topics about TRL Calibration

Overview

The 3-port TRL calibration function lets you measure calibration data by connecting thru, reflection (open or short), line, or match calibration standards to (between) 3 desired test ports. This calibration provides the most accurate measurement for non-coaxial parts, using 27 error terms model in total for calibration in the same way as full 3-port calibration.

Before executing TRL calibration, you need to modify the calibration kit definition you use (or create a new one). For more information, see Changing the Calibration Kit Definition.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to execute calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **3-Port TRL Cal** > **Select Ports**.
4. Select test ports for which you want to execute TRL 3-port calibration. (In the following procedure, test ports you select here are expressed as test ports x, y, and z.)
5. Click **Thru/Line** to start the measurement of the calibration standard.
6. Click **Reflect** to start the measurement of the calibration standard.
7. Click **Line/Match** to start the measurement of the calibration standard.

Softkey	Function
x-y Line/Match	Executes line/match calibration for test ports x and y. Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
x-y Fwd (Syx)	Executes line/match calibration for test ports x and y. Use this to perform forward-direction measurement only.
x-y Rvs (Sxy)	Executes line/match calibration for test ports x and y. Use this to perform reverse-direction measurement only.
x-z Line/Match	Executes line/match calibration for test ports x and z. Use this to perform forward-direction measurement and reverse-direction measurement at the same time.

x-z Fwd (Szx)	Executes line/match calibration for test ports x and z. Use this to perform forward-direction measurement only.
x-z Rvs (Sxz)	Executes line/match calibration for test ports x and z. Use this to perform reverse-direction measurement only.
y-z Line/Match	Executes line/match calibration for test ports y and z. Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
y-z Fwd (Szy)	Executes line/match calibration for test ports y and z. Use this to perform forward-direction measurement only.
y-z Rvs (Syz)	Executes line/match calibration for test ports y and z. Use this to perform reverse-direction measurement only.

8. Click **Done** to finish TRL 3-port calibration. At this point, the calibration coefficient is calculated and saved. The error correction function is automatically turned on.

4-port TRL Calibration

- Overview
- Procedure

Other topics about TRL Calibration

Overview

The 4-port TRL calibration function lets you measure calibration data by connecting thru, reflection (open or short), line, or match calibration standards to (between) 4 test ports. This calibration provides the most accurate measurement for non-coaxial parts, using 48 error terms model in total for calibration in the same way as full 4-port calibration.

Before executing TRL calibration, you need to modify the calibration kit definition you use (or create a new one). For more information, see Changing the Calibration Kit Definition.

Procedure

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which you want to execute calibration.
2. Press **Cal** key.
3. Click **Calibrate** > **4-Port TRL Cal** > **Select Ports.**
4. Click **Thru/Line** to start the measurement of the calibration standard.
5. Click **Reflect** to start the measurement of the calibration standard.

Softkey	Function
Port 1 Reflect	Executes reflection calibration for test port 1.
Port 2 Reflect	Executes reflection calibration for test port 2.
Port 3 Reflect	Executes reflection calibration for test port 3.
Port 4 Reflect	Executes reflection calibration for test port 4.

6. Click **Line/Match** to start the measurement of the calibration standard.

Softkey	Function
1-2 Line/Match	Executes line/match calibration for test ports 1 and 2.Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
1-2 Fwd (S21)	Executes line/match calibration for test ports 1 and 2.Use this to perform forward-direction measurement only.

1-2 Rvs (S12)	Executes line/match calibration for test ports 1 and 2.Use this to perform reverse-direction measurement only.
1-3 Line/Match	Executes line/match calibration for test ports 1 and 3.Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
1-3 Fwd (S31)	Executes line/match calibration for test ports 1 and 3.Use this to perform forward-direction measurement only.
1-3 Rvs (S13)	Executes line/match calibration for test ports 1 and 3.Use this to perform reverse-direction measurement only.
1-4 Line/Match	Executes line/match calibration for test ports 1 and 4.Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
1-4 Fwd (S41)	Executes line/match calibration for test ports 1 and 4.Use this to perform forward-direction measurement only.
1-4 Rvs (S14)	Executes line/match calibration for test ports 1 and 4.Use this to perform reverse-direction measurement only.
2-3 Line/Match	Executes line/match calibration for test ports 2 and 3.Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
2-3 Fwd (S32)	Executes line/match calibration for test ports 2 and 3.Use this to perform forward-direction measurement only.
2-3 Rvs (S23)	Executes line/match calibration for test ports 2 and 3.Use this to perform reverse-direction measurement only.
2-4 Line/Match	Executes line/match calibration for test ports 2 and 4.Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
2-4 Fwd (S42)	Executes line/match calibration for test ports 2 and 4.Use this to perform forward-direction measurement only.
2-4 Rvs (S24)	Executes line/match calibration for test ports 2 and 4.Use this to perform reverse-direction measurement only.
3-4 Line/Match	Executes line/match calibration for test ports 3 and 4. Use this to perform forward-direction measurement and reverse-direction measurement at the same time.
3-4 Fwd (S43)	Executes line/match calibration for test ports 3 and 4. Use this to perform forward-direction measurement only.
3-4 Rvs (S34)	Executes line/match calibration for test ports 3 and 4. Use this to perform reverse-direction measurement only.

7. Press **Done** to finish TRL 4-port calibration. At this point, the calibration coefficient is calculated and saved. The error correction function is automatically turned on.

Making Measurements

Making Measurements

- Setting Up Trigger and Making Measurements
- Setting Point Trigger
- Setting Low-Latency External Trigger Mode
- Setting Averaging Trigger Function
- Setting External Trigger Output

Setting Trigger and Making Measurements

- Overview
- Sweep Order in Each Channel
- Trigger Source
- Trigger Scope
- Trigger Mode
- Setup Procedure for Trigger

Other topics about Making Measurement

Overview

The E5071C has one trigger source. When this trigger source detects a trigger signal that has occurred, a sweep is performed for channels in the "Initiate" state in the order of channel 1 to channel 36. You set the "Initiate" or "Idle" status of each channel by changing the trigger mode.

NOTE The execution of measurement for each channel does not depend on whether the channel is displayed. Channels that have been activated can be measured even if they are not displayed.

NOTE For each channel, a sweep is performed only for the stimulus ports needed to update the parameters of the displayed trace.

Sweep Order in Each Channel

In a channel, each test port is set to a stimulus port in the order of port number and updates each trace.

Sweep Order	Stimulus Port	Updated Trace
1	Port 1	S11, S21, S31, S41
2	Port 2	S12, S22, S32, S42
3	Port 3	S13, S23, S33, S43
4	Port 4	S14, S24, S34, S44

NOTE If full 2, 3, or 4-port error correction is in effect, no trace between calibrated ports is updated until the last calibrated port is swept as the stimulus port.

NOTE Sweep is not executed for stimulus ports that are not required for updating traces.

Trigger Source

The trigger source generates a cue signal that initiates a measurement process. Four types of trigger sources are available.

Trigger Sources	Function
Internal (**Internal**)	Uses a consecutive signal generated by the firmware as a trigger source. Triggers are sent immediately following the completion of each measurement.
External (**External**)	Uses the external trigger input terminal (BNC) or Handler I/O (Pin No. 18) as a trigger source.
Manual (**Manual**)	A trigger is generated by pressing **Trigger** > **Trigger**.
Bus (**Bus**)	A trigger is generated through GPIB/LAN/USB.

Trigger Scope

The Trigger Scope specifies the scope of the triggering, whether it is for all channels (default) or for the selected channel.

When this function is enabled with a value of "ACTive", only active channel is triggered. When this function is enabled with a value of "ALL", all channels of the E5071C are triggered.

For example, if Trigger Scope is set to "ACTive" when **Trigger** > **Continuous** is selected for all channels, a measurement channel is automatically changed by switching over the active channel.

Trigger Scope	Function
All Channel	Triggers ALL channels which are in "Initiate" status
Active Channel	Triggers Active channel which is in "Initiate" status

Trigger Mode

You can set the trigger mode for each channel independently. This allows you to control the operation of each channel after a trigger signal is detected by setting the channel's status with the trigger mode.

Trigger mode name	Function
Sweep stop (**Hold**)	The status ("Idle" status) in which the sweep is stopped. When a trigger signal is detected, the sweep is not performed.
Single sweep	An "Initiate" status. When a trigger signal is detected, a sweep is

(**Single**)	performed. After completion of the sweep, the "Idle" status is activated.
Continuous sweep (**Continuous**)	An "Initiate" status. When a trigger signal is detected, a sweep is performed. After completion of the sweep, the "Initiate" status is maintained. The sweep is repeated each time a trigger signal is detected.
Hold All Channels	Sets all channel trigger modes to hold sweep mode
Continuous Disp Channels	Sets trigger modes of all displayed channels (**Display** > **Allocate Channels**) to continuous sweep mode

Setup Procedure For Trigger

1. Selecting a Trigger Source

Follow the procedure below to select a trigger source.

1. Press **Trigger**.
2. Click **Trigger Source**.
3. Click the softkey that corresponds to the desired trigger source.
4. When **External** is selected as a trigger source, click **Ext Trig Input** to select trigger polarity.

Softkey	Function
Negative Edge	Triggered at negative edge of an external trigger input signal.
Positive Edge	Triggered at positive edge of an external trigger input signal.

5. The setting for trigger polarity is not valid for the external trigger from Hander I/O.

2. Selecting a Trigger Scope

Follow the procedure below to select a trigger mode.

1. Press **Trigger** key.
2. Click **Trigger Scope**.
3. Click the softkey that corresponds to the desired trigger scope:

Trigger Scope	Function
All Channel	Triggers ALL channels
Active Channel	Triggers Active channel alone.

3. Selecting a Trigger Mode

Follow the procedure below to select a trigger mode.

1. Press **Channel Next**/**Channel Prev** keys to select the channel for which the trigger mode will be set.
2. Press **Trigger** key.
3. Press the softkey that corresponds to the desired trigger mode. Repeat the procedure until each channel is set for its trigger mode.

Softkey	Function
Hold	Sets active channel trigger mode to hold sweep mode
Single	Sets active channel trigger mode to single sweep mode
Continuous	Sets active channel trigger mode to continuous sweep mode
Hold All Channels	Sets all channel trigger modes to hold sweep mode
Continuous Disp Channels	Sets trigger modes of all displayed channels (**Display** > **Allocate Channels**) to continuous sweep mode

4. Generating the Trigger

Next, it is necessary to generate a trigger by using the trigger source selected in Selecting a Trigger Source.

NOTE Once the internal trigger source is selected, a series of triggers is continuously generated as soon as the setting becomes effective.

NOTE Pressing **Trigger** > **Restart** during a sweep forces the analyzer to abort the sweep.

Setting Point Trigger

- Overview
- Setting Point Trigger

Other topics about Making Measurement

Overview

The point trigger provides a point measurement at every trigger, and it can be used to change the trigger event to point trigger mode.

Trigger event name	Function
On Point (On)	Point measurement is performed when trigger is applied.
On Sweep (Off)	Measurement is performed for all measurement points when trigger is applied.

NOTE When the trigger source is selected as the internal trigger, the point trigger does not work.

Setting Point Trigger

1. Press **Trigger** key.
2. Click **Trigger Event.**
3. Click the softkey that corresponds to the desired trigger event.

Softkey	Function
On Point	Measures at each measurement point
On Sweep	Measures all measurement points

Setting Low-Latency External Trigger Mode

Other topics about Making Measurement

Overview

In the low-latency external trigger mode, variations in delay time between the reception of a trigger and the start of a one-point measurement are decreased for point trigger measurement by using trigger pulses supplied to the external trigger input terminal. In addition, using this mode will allow you to decrease the variations in the time required to be ready for the next measurement trigger after execution of a one-point measurement. This mode allows you to perform continuous point trigger measurement in sync with external trigger pulses and accurately set a trigger delay time at each point-trigger point.

NOTE The time required for measurement at each measurement point varies depending on the settings of the E5071C and the type of measurement. For more information, see External trigger delay time and point trigger interval.

The low-latency external trigger mode becomes available by turning on the point trigger function and setting the trigger source to "External."

Low-latency external trigger mode	Function
ON	Sets the low-latency external trigger mode to ON. The external trigger delay time becomes valid.
OFF	Sets the low-latency external trigger mode to OFF.

NOTE If the trigger source is not the external trigger, the low-latency external trigger mode does not work if it is specified to ON.

NOTE The setting of the low-latency external trigger mode is valid for all channels.

Setting Low-Latency External Trigger

1. Setting the low-latency external trigger

If the trigger source is set to "External" and the trigger event to "On" (On Point), set the low-latency external trigger mode.

1. Press **Trigger** key.

2. Click **Low Latency**.
3. Click **ON** to activate the-low latency external trigger mode.

2. Setting the external trigger delay time

Set the external trigger delay time at each point. The setting resolution is 10 μs.

1. Press **Trigger** key.
2. Click **Trig Delay**.
3. Enter an external trigger delay time.

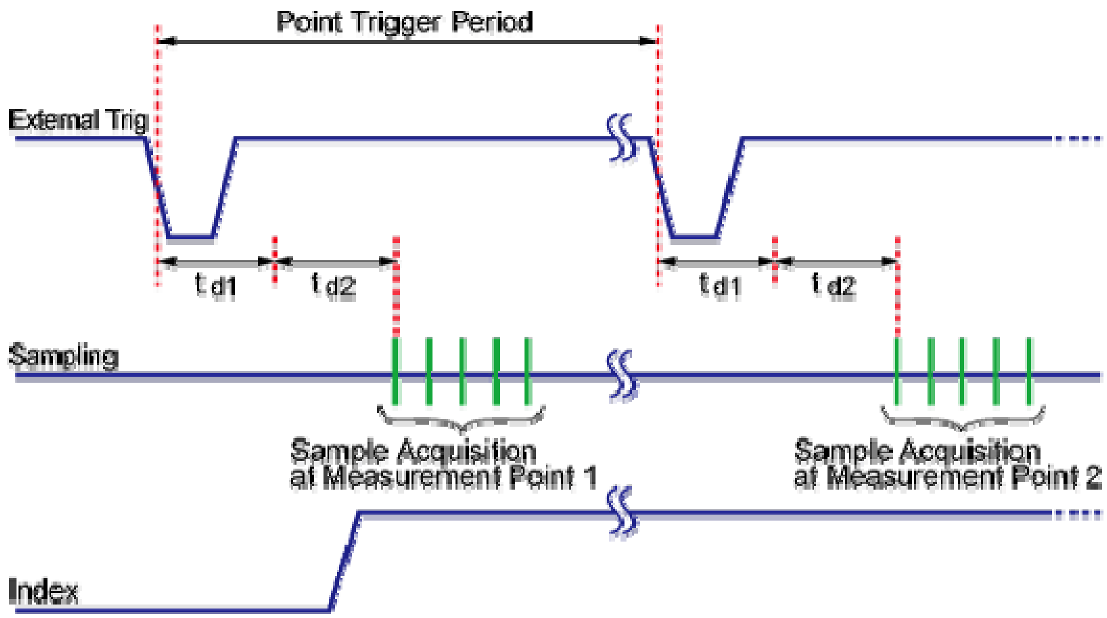
External Trigger Delay Time and Point Trigger Interval

External trigger pulses, supplied until the next measurement becomes ready after the start of a one-point measurement, are ignored, and the next trigger is generated by a pulse supplied after the completion of the one-point measurement.

The time until the next trigger can be accepted after the start of a one-point measurement depends on the IFBW and other settings of the analyzer. For example, in the case of a frequency's zero-span measurement, the time until the next measurement is ready after the start of a one-point measurement is obtained by dividing the time required for a single sweep in On Sweep mode, instead of On Point mode, by the number of measurement points. If you use the point trigger function with external trigger pulses that are wider than this time, point trigger measurement is performed at each pulse input.

The figure below shows the timing chart of an external trigger when the point trigger function is on.

Timing chart of external trigger (trigger event = on, trigger source = external, low-latency external trigger mode = on)



e5071c345

The table below describes signals and time as shown in the above figure.

Signal, time	Description
External Trig	External trigger signal to be supplied.
Sampling	Time while the E5071C is actually performing measurement.
Index	/Index signal of the handler I/O port. When the point trigger function is ON and the low-latency external trigger mode is ON, it goes to the High level only before staring measurement of the first sweep point and returns to the Low level after completing measurement of all measurement points.
Point Trigger Period	Time until the E5071C is ready to accept a trigger for the next measurement point. The value depends on the measurement conditions and the settings of the E5071C.
td1	Time set as the external trigger delay time.
td2	Time that the E5071C requires internally for setting. This depends on the settling time for setting the frequency, IF frequency band, and so on. For zero-span measurement, td2 = 13±1 μs (Typ.).

Setting Averaging Trigger Function

- Overview
- Averaging Trigger Function

Other topics about Making Measurement

Overview

The averaging trigger function is used to execute the sweep the number of times specified by the averaging factor with a single trigger when the sweep averaging function is **ON**.

Averaging Trigger	Function
ON	Performs the sweep the number of times specified by the averaging factor with a single trigger.
OFF	Performs the sweep once with a single trigger.

The averaging factor is cleared before the start of measurement.

NOTE When the point trigger function is on, its setting has priority, and you need to generate triggers based on "(number of measurement points)× (averaging factor)".

NOTE When the sweep averaging function is off, sweep is performed only once even if the averaging trigger function is set to on.

NOTE The averaging trigger function is valid for all channels. Note that you can set the sweep averaging function for each channel.

Averaging Trigger Function

1. Setting Averaging Trigger Function

When the sweep averaging function is **ON**, follow these steps to set the averaging trigger function.

1. Press **Avg** key.
2. Click **Avg Trigger**.
3. Click **ON** to activate the averaging trigger.

2. Executing Averaging Measurement

1. Press **Trigger** key.
2. Click **Single**. The averaging factor is cleared before the start of measurement, the sweep is executed the number of times specified by the averaging factor, and then the instrument waits for the next trigger.

Setting External Trigger Output

- Overview
- Setting External Trigger Output

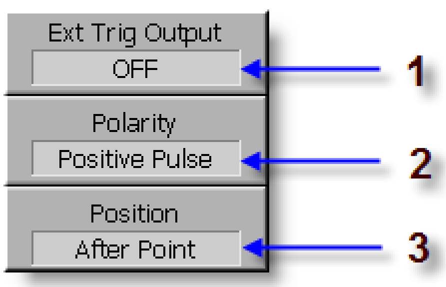
Other topics about Making Measurement

Overview

The External Trigger Output port (located at the rear panel) can be used to provide trigger to an external device. This is useful in cases where an external device needs to be triggered through the E5071C.

Setting External Trigger Output

1. Press **Trigger** key, and then set the value of **Ext Trig Output** (**1**) as **ON**.



e5071c196

2. Click **Polarity**, and then select **Positive Pulse** or **Negative Pulse** (**2**).

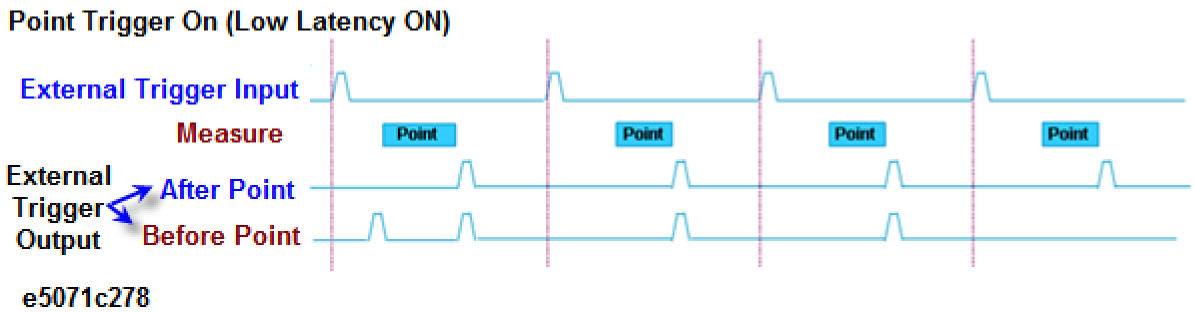
Polarity Property	Description
Positive Pulse	Generates a Positive Pulse as a trigger.
Negative Pulse	Generates a Negative Pulse as a trigger.

3. Click **Position**, and then select **After Point** or **Before Point** (**3**).

Position Property	Description
After Point	Generates a Pulse (trigger) after the measurement points.
Before Point	Generates a Pulse (trigger) before the measurement points.

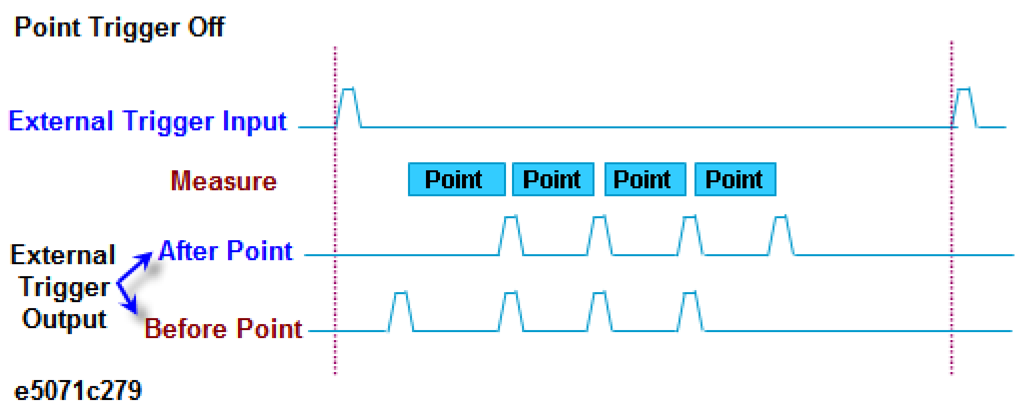
Difference between After and Before Point Settings

External trigger output function displays the difference result by setting the Point trigger as follows:



After Point:Pulse is generated after every measurement point.

Before Point:Pulse is generated before first measurement point. Rest of pulse are same timing as After Point. Number of pulse is same as number of measurement point.



After Point:Pulse is generated after every measurement point.

Before Point:Pulse is generated before every measurement point.

Data Analysis

Data Analysis

- Analyzing Data on the Trace Using the Marker
- Searching for Positions that Match Specified Criteria
- Determining the Bandwidth of the Trace (Bandwidth Search)
- Determining the Bandwidth of the Trace (Notch Search)
- Determining the Mean, Standard Deviation, and p-p of the Trace
- Comparing Traces/Performing Data Math
- Performing Parameter Conversion of Measurement Results
- Holding Max/Min Points for the Trace
- Limit Test
- Ripple Test
- Bandwidth Test
- Point Limit
- Equation Editor

Analyzing Data on the Trace Using the Marker

- About Marker Functions
- Reading Marker Values on Trace
- Reading Relative Value from Reference Point on Trace
- Reading Actual Measurement Point/Value Interpolated between Measurement Points
- Setting up Markers for Each Trace/Setting up Markers for Coupled Operation between Traces
- Listing Marker Values in All Displayed Channels
- Specifying Display Position of Marker Values
- Aligning Marker Value Display
- Displaying All Marker Values for Displayed Traces

Other topics about Data Analysis

About Marker Functions

The marker can be used in the following ways:

- Reading a measured value as numerical data (as an absolute value or a relative value from the reference point)
- Moving the marker to a specific point on the trace (marker search)
- Analyzing trace data to determine a specific parameter
- Using the value of the marker to change the stimulus (sweep range) and scale (value of the reference line)

For the procedure used to change the sweep range and scale by using the marker, refer to Setting the Sweep Range Using the Marker and Setting the value of a reference line using the marker.

The E5071C is capable of displaying up to 10 markers including the reference marker on each trace. Each marker has a stimulus value (the value on the X-axis in rectangular display format) and a response value (the value on the Y-axis in rectangular display format). The Smith chart and polar formats each have two marker response values (log amplitude and phase).

Reading Marker Values on Trace

You can read the value of a marker displayed on the trace.

In rectangular display format, the marker response value is always in the same data format as that of the Y-axis. On the contrary, one format for the marker response values (two values: main and auxiliary) can be selected from among several types. The selection is performed in the data format.

Softkey for selecting data format	Marker response value	
	Main	**Auxiliary**
Smith - Lin / Phase	Linear amplitude	Phase
Smith - Log / Phase	Log amplitude	Phase
Smith - Real / Imag	Real component	Imaginary component
Smith - R + jX	Resistance	Reactance
Smith - G + jB	Conductance	Susceptance
Polar - Lin / Phase	Linear amplitude	Phase
Polar - Log / Phase	Log amplitude	Phase
Polar - Real / Imag	Real component	Imaginary component

For setting up data formats, refer to Selecting a Data Format.

Activating the marker on the Trace

1. Press **Channel Next**/**Channel Prev** keys to activate the channel on which a marker is used.
2. Press **Trace Next**/**Trace Prev** to activate the trace on which a marker is used.
3. Press **Marker** key. At this point, marker 1 is turned on and becomes active (you can operate the marker). When using marker 1, you can omit the next step.
4. Select a marker and turn it on. The softkey used to turn on a marker is also used to activate that marker.

Moving the marker

1. Change the marker stimulus value. This operation enables you to move the marker to a point on the selected trace.
2. Read the marker stimulus value and marker response value displayed in the upper-left part of the trace screen.

Turning off the marker

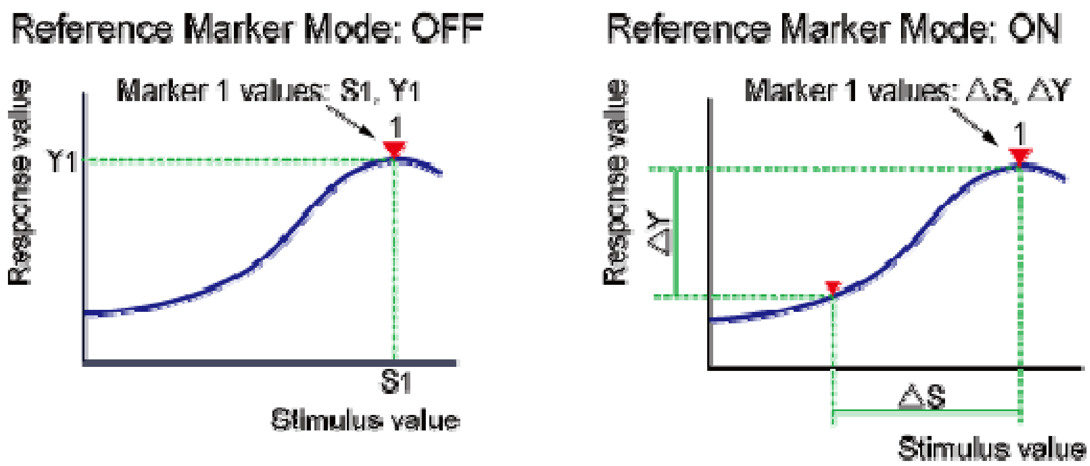
1. Press **Marker** key.

2. Click **Clear Marker Menu** and then click one of the options.

NOTE In the preset configuration, the marker settings on traces in a channel are coupled (Marker Couple is turned on). For marker coupling, refer to Setting up markers for each trace/Setting up markers for coupled operations between traces.

Reading Relative Value from Reference Point on Trace

You can convert the marker reading into a relative value from the reference point.



e5071c310

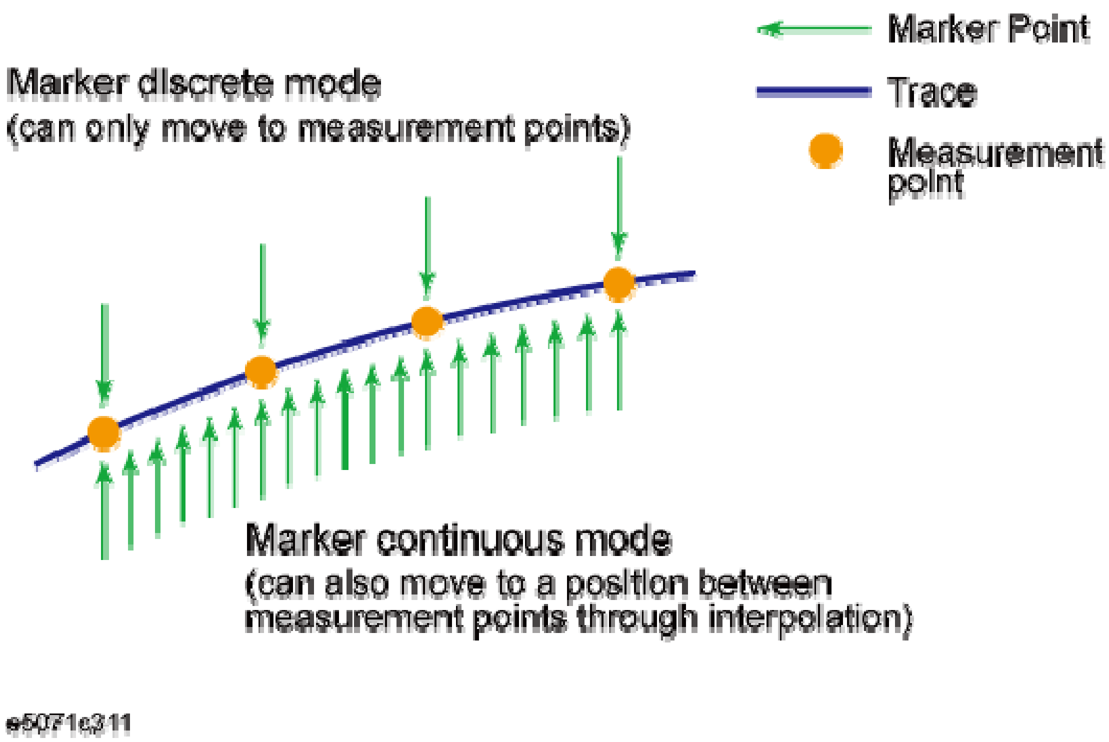
Converting From a Reference Point to a Relative Value

1. Activate the reference marker.
2. Move the reference marker at the point to be used as the reference.
3. Click **Ref Marker Mode** to turn on the reference mode.
4. With the reference mode turned on, the stimulus values and response values are indicated in relative values referred to by the position of the reference marker.
5. Activate your desired marker, then move it to your desired position.
   6. Pressing Marker and then clicking **Ref Marker** enables you to place the reference marker at the position of the currently *active marker* . The reference mode will then turn on automatically.

Reading Actual Measurement Point/Value Interpolated between Measurement Points

The point on the trace on which a marker can be placed differs depending on how the discrete marker mode is set up.

Value	Description
Turning on discrete mode (**Discrete ON**)	A marker moves only between actual measurement points. When a specific marker stimulus value is specified as a numerical value, the marker is placed at the measurement point closest to the specified value. A marker that is placed between interpolated points with the discrete mode off automatically moves to the nearest measurement point when the discrete mode is turned on.
Turning off discrete mode (**Discrete OFF**)	The marker can move from one actual measurement point to another. Because it is interpolated, it can also move in the space between measurement points.

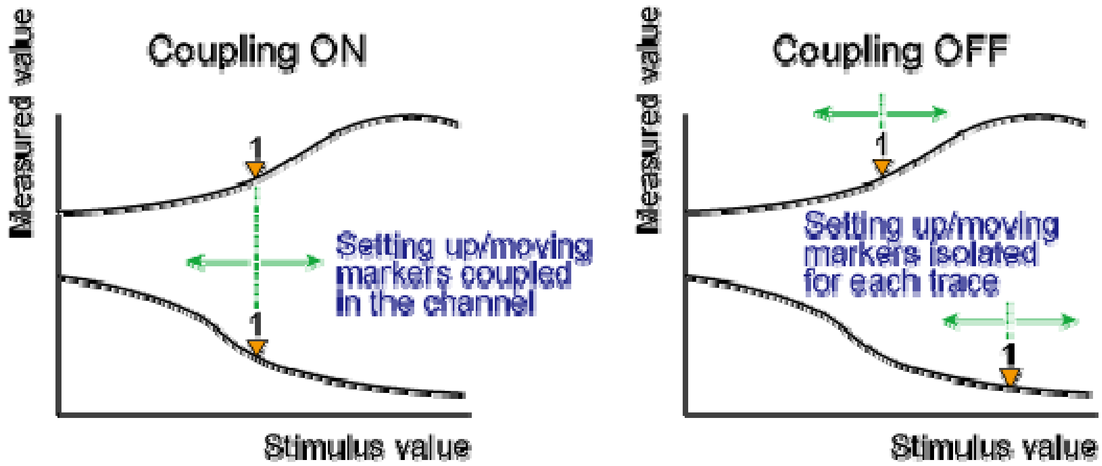


Turning Discrete Mode On or Off

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** to activate the trace on which the discrete mode is set up.
2. Press **Marker Fctn** key.
3. Click **Discrete** to turn the discrete mode on or off.

Setting up Markers for Each Trace / Setting up Markers for Coupled Operation between Traces

Makers can be set up and moved either in coupled operation for all traces in a channel or independently for each trace.



e5071c388

Value	Description
Marker Couple is on (**Coupling ON**)	Markers are set up and moved in coupled operation on all traces in a channel.
Marker Couple is off (**Coupling OFF**)	Markers are set up and moved independently for each trace.

Turning Marker Coupling On or Off

1. Press **Channel Next**/**Channel Prev** keys to activate the channel on which the marker couple will be set.
2. Press **Marker Fctn** key.
3. Click **Couple** to turn the marker coupling on or off.

Listing all Marker Values in all Displayed Channels

You can list all of the marker values in all of the displayed channels on the screen.

Turning On the Marker Table Display

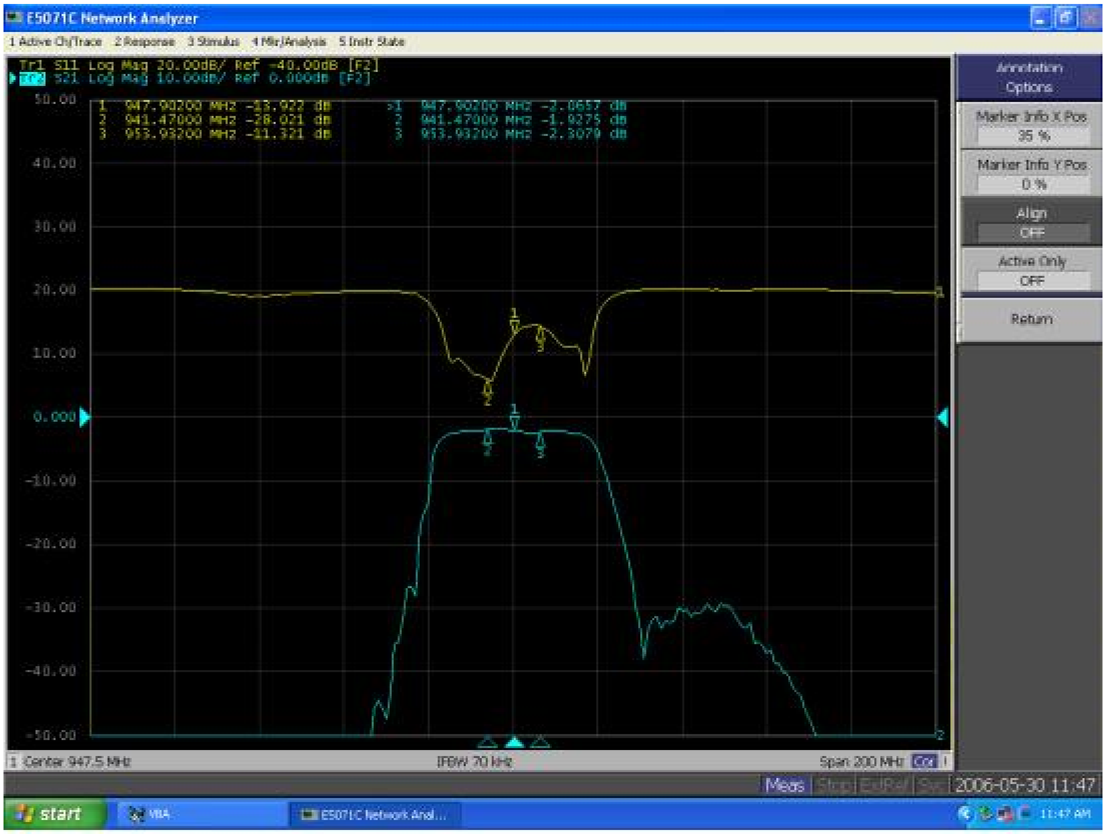
1. Press **Marker Fctn** key.
2. Click **Marker Table** to turn on the marker table display.

The marker table appears in the lower part of the screen.

e5071c063

Specifying Display Position of Marker Values

This section describes how to specify the marker value display position for each active trace.



e5071c064

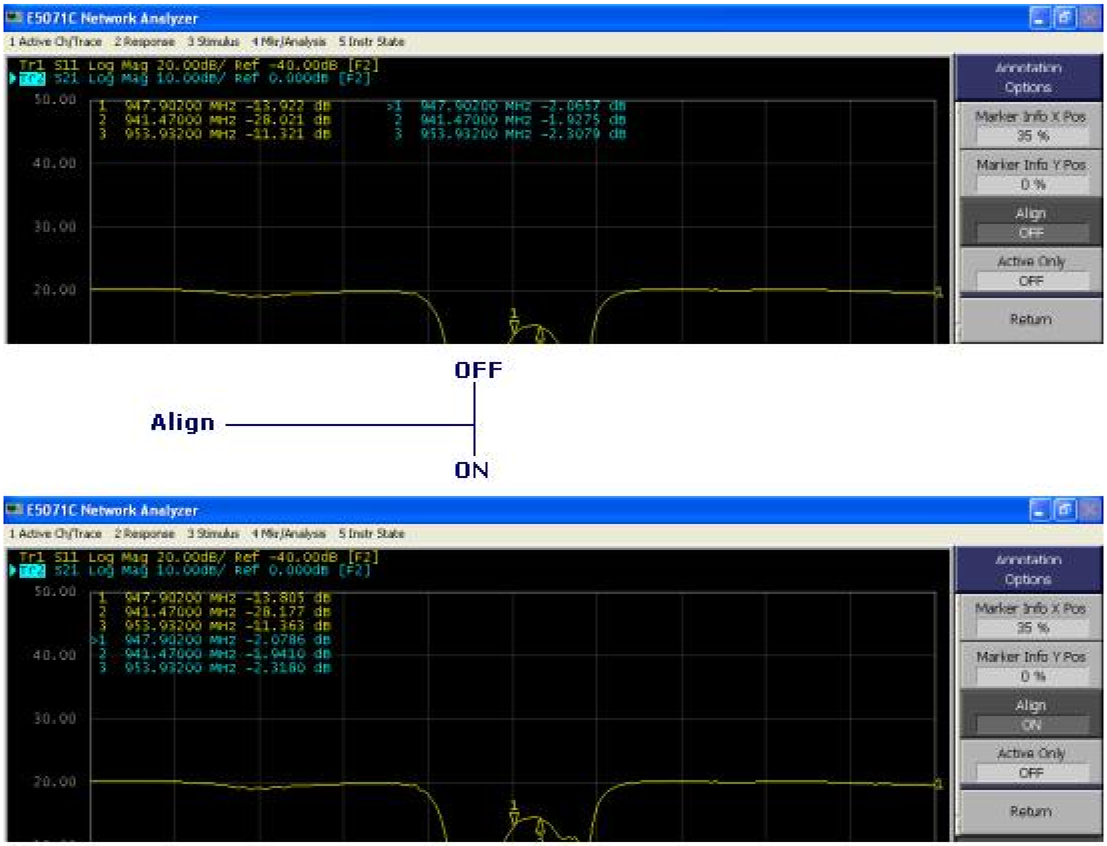
Value	Description
Marker Info X Pos	Specifies the horizontal display position by the width of the display area as a percentage.
Marker Info Y Pos	Specifies the vertical display position by the height of the display area as a percentage.

Operational procedure

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set marker coupling.
2. Press **Marker Fctn** key.
3. Click **Annotation Options.**
4. Click **Marker Info X Pos** to set the horizontal display position.
5. Click **Marker Info Y Pos** to set the vertical display position.

Aligning Marker Value Display

This section describes how to align maker value displays.



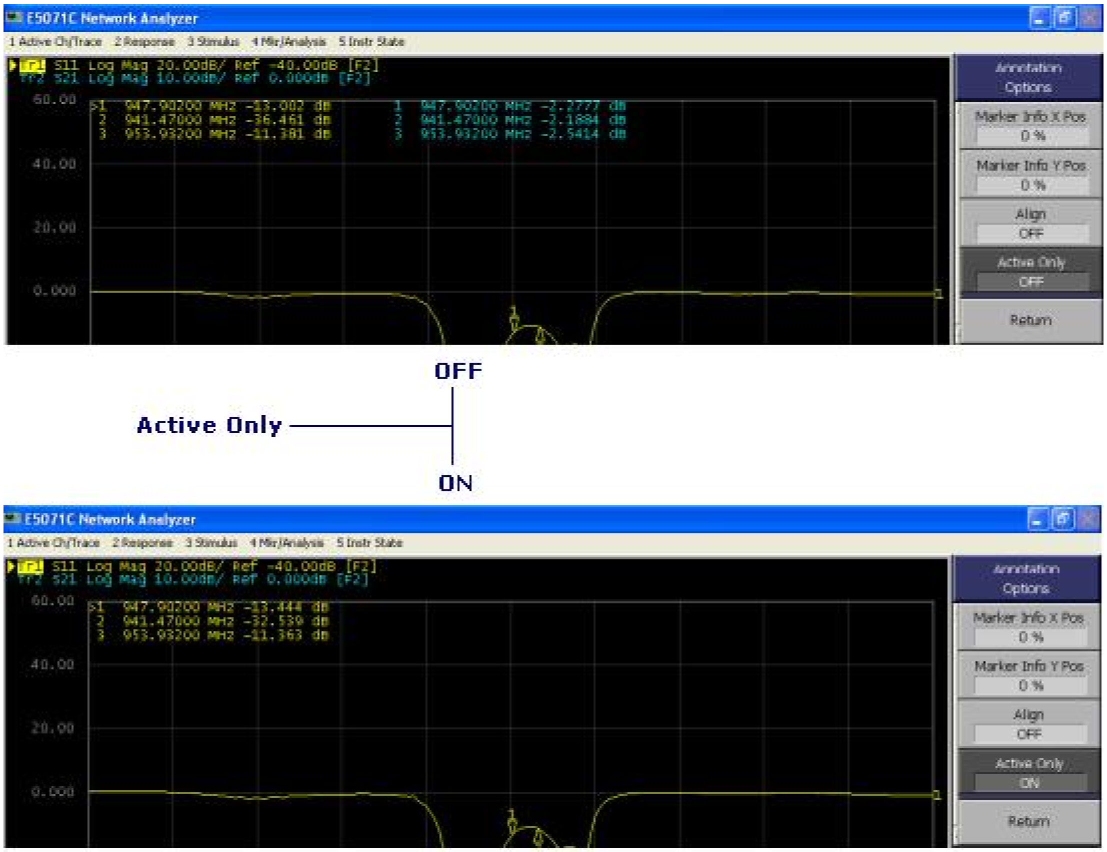
e5071c065

Value	Description
On (**Align ON**)	Displays marker values to align to the display position of trace 1.
Off (**Align OFF**)	Displays marker values in the display position defined for each trace.

1. Press **Marker Fctn** key.
2. Click **Annotation Options.**
3. Click **Align** to toggle ON/OFF.

Displaying All Marker Values for Displayed Traces

This section describes how to display all marker values for displayed traces.



e5071c066

Value	***Description***
Displays all *(***Active Only OFF***)*	Displays all marker values for displayed traces.
Displays active markers *(***Active Only ON***)*	Displays markers for the active trace only.

1. Press **Marker Fctn** key.
2. Click **Annotation Options.**
3. Click **Active Only** to toggle ON/OFF.

Searching for Positions that Match Specified Criteria

- Overview
- Setting Search Range
- Setting Multiple Range
- Automatically Executing a Search (Search Tracking)
- Searching for Maximum and Minimum Values
- Searching for the target value (Target search)
- Searching for the peak

Other topics about Data Analysis

Overview

You can search for a position that matches your specified criteria by using the Marker Search feature. Marker Search allows you to search for a position that matches any of the following criteria.

- Maximum value
- Minimum value
- Target (a point that has a target measurement value)
  - Target nearest to marker position
  - Target on left/right-hand side nearest to marker position
  - Multiple Target (Searching all target points at one time)
- Peak
  - Maximum peak (for a positive peak), minimum peak (for a negative peak)
  - Peak on left/right-hand side nearest to marker position
  - Multiple Peak (Searching all peak points at one time)

Setting Search Range

The Marker Search feature allows you to set part of the sweep range as the search target (Partial Search feature) as well as the entire search range. For the Partial Search feature, you can select whether to couple traces in the channel. The Marker Statistics value is calculated by using the search range.

Procedure to Turn ON/OFF Trace Coupling within Search Range

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace for which you want to set the search range.
2. Press **Marker Search** key.

3. Click **Search Range**.
4. Click **Couple** to toggle *ON/OFF* trace coupling within the search range.

Procedure to Set Search Range

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace for which you want to set the search range.
2. Press **Marker Search** key.
3. Click **Search Range**.
4. Click **Search Range** to turn ON the Partial Search feature.
5. Click **Start**, then enter the start value (lower limit) of the search range.
6. Click **Stop**, then enter the stop value (upper limit) of the search range.

Procedure to Show Statistics Value of the Search Range

1. Set and define a search range as described above.
2. Press **Marker Fctn** key.
3. Click **Statistics** to turn ON. The statistics values are calculated and displayed on the screen.

Setting Multiple Range

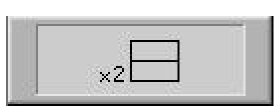
The Multiple Range feature allows you to set up to 16 search range in a channel. You can select more than one marker in a search range. You can select to couple the search range among the traces in the channel.

The Marker Statistics value is calculated by using the search range of the active marker. If no active marker is selected, by default the search range of marker 1 is used.

Procedure to Place Multiple Markers within a Search Range (in Multiple Range)

The below example shows how to place multiple markers within several search range on two traces. In this example, stimulus is set for two. Then, two multiple ranges are set and two markers are assigned to each search range as min and max. The effect of search range coupling and trace coupling on multiple range is also shown in this example.

Setting up traces and stimulus

1. **Display** > **Num of Traces** > **2.**
2. **Display** > **Allocate Traces** > 

.
3. **Start** > **2.0000 GHz.**

4. **Stop** > **8.0000 GHz**.
5. **Scale** > **Autoscale All**.

Turning ON and setting up multiple range

1. Select **Trace 1.**
2. Press **Marker Search** key.
3. Click **Search Range** > **Search Range** > **ON** to turn ON the partial search feature.
4. Click **Multiple Range** > **Multiple Range** > **ON** to turn ON the multiple search range feature.
5. Click **Target Range** and select within 1 to 16 which represents the search range number. In this example, select **1** which represents Search Range 1.
6. Click **Start**, then enter **3.0000 GHz**, the start value (lower limit) of the search range.
7. Click **Stop**, then enter **4.5000 GHz**, the stop value (upper limit) of the search range.
8. Similarly, click **Target Range** > **2**.
9. **Start** > **5.0000 GHz**.
10. **Stop** > **6.5000 GHz**.
11. The search range is set for both, Trace 1 and Trace 2. Note that the search range is not displayed as no Marker is assigned to these search range.

Assigning markers to search range

1. **Marker** > **Marker 1** to turn it ON.
2. **Marker Search** > **Search Range** > **Multiple Range** > **Target Range** > **1** to activate search range 1.
3. Similarly, turn ON **Marker 2**.
4. Activate search range 1, by following Step 2.
5. Note that now, search range 1 is displayed, and marker 1 and marker 2 are placed within search range 1.

Moving markers to min or max

1. **Marker** > **Marker 1** to activate it.
2. **Marker Search** > **Max**. Marker 1 is placed at the maximum point within search range 1.

3. **Marker** > **Marker 2** to activate it.
4. **Marker Search** > **Min**. Marker 2 is placed at the minimum point within search range 1.

Using search range coupling and marker coupling in multiple search range

1. Select **Trace 2.**
2. By default, search range coupling is turned ON. **Marker Search** > **Search Range** > **Couple** > **OFF** to turn OFF search range coupling.
3. **Marker** > **Marker 3** to activate it.
4. Activate search range 2.
5. Note that Trace 1 displays search range 1 while Trace 2 displays search range 2.
6. **Marker Search** > **Max.** Marker 3 is now placed at the maximum point within search range 2. Marker 3 at Trace 1 is in parallel with Marker 3 at Trace 2. However, Trace 1 still displays search range 1 while Trace 2 displays search range 2.
7. By default, marker coupling is turned ON. **Marker Fctn** > **Couple** > **OFF** to turn OFF marker coupling.
8. **Marker** > **Marker 4** to activate it. As marker coupling is turned OFF, marker 4 is activated for Trace 2 only. To learn more about marker coupling, refer to Setting up markers for each trace/Setting up markers for coupled operations between traces.
9. Activate search range 2.
10. **Marker Search** > **Min**. Marker 4 is now placed at the minimum point within search range 2. However, note that Marker 4 is not shown at Trace 1.

Procedure to Show Statistics Value of the Active Search Range

1. Click **Search Range** > **Multiple Range** > **Target Range**.
2. Select the search range number which you want its statistics value to be calculated and shown on the screen.
3. Press **Marker Fctn** key.
4. Click **Statistics** to turn ON the display of statistics data.

Automatically Executing a Search (Search Tracking)

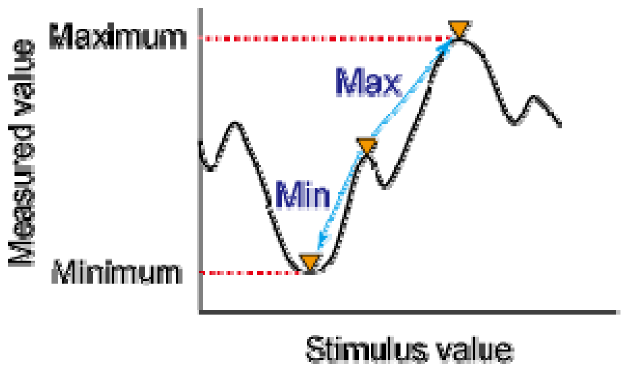
Search tracking is a function that sets a search to be repeated every time a sweep is done even if the execution key for the search (maximum, minimum, peak, and target) is not pressed. This function facilitates observation of measurement results such as the maximum value of traces (e.g., the insertion loss of a band pass filter).

Performing Search Tracking

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace on which you want to set up search tracking.
2. Press **Marker Search** key.
3. Click **Tracking** and turn the search tracking function *ON/OFF*.

Searching for Maximum and Minimum Values

You can search for the maximum or minimum measured value on the trace and move a marker to that point.



e5071c386

Search for maximum (**Max**)	Move *active marker* to point on trace where measured value is greatest
Search for minimum (**Min**)	Move active marker to point on trace where measured value is lowest

Procedure

1. Activate the marker you are using to search for the maximum and minimum values.
2. Press **Marker Search** key.
3. Click the corresponding softkey to move the marker to the maximum or minimum measured value.

NOTE When the data format is in Smith chart or polar format, execute the search only for the main response value.

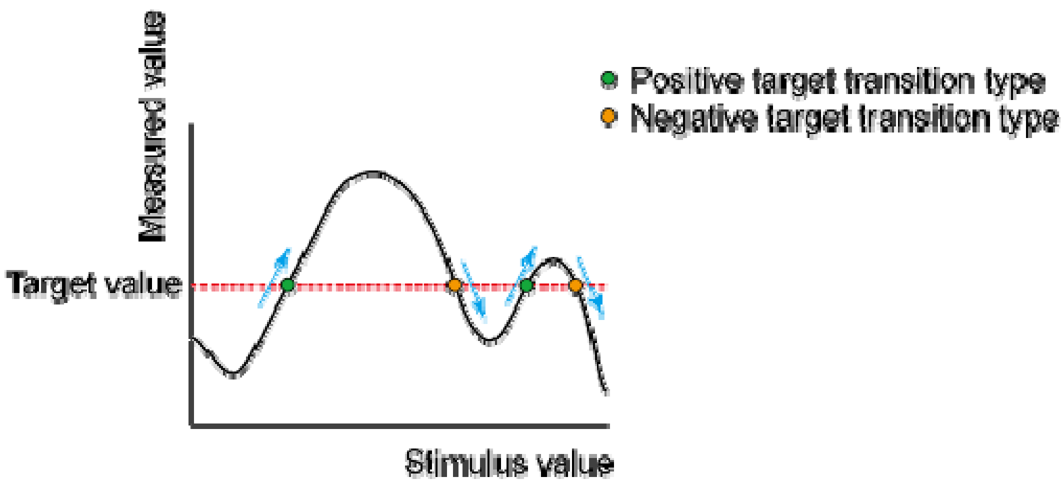
Searching for the target value (target search)

The target search is a function that searches for a target that matches the pre-defined target value and transition type(s) (positive, negative, or both positive and negative) and then moves the marker to that target.

Target and Transition Types

A target is a point that has a specific measured value on the trace. Targets can be divided into the two groups shown below depending on their transition type.

Transition type: Positive (**Positive**)	When the value of the target is larger than the measured value that immediately precedes it (on the left side)
Transition type: Negative (**Negative**)	When the value of the target is smaller than the measured value that precedes immediately it (on the left side)

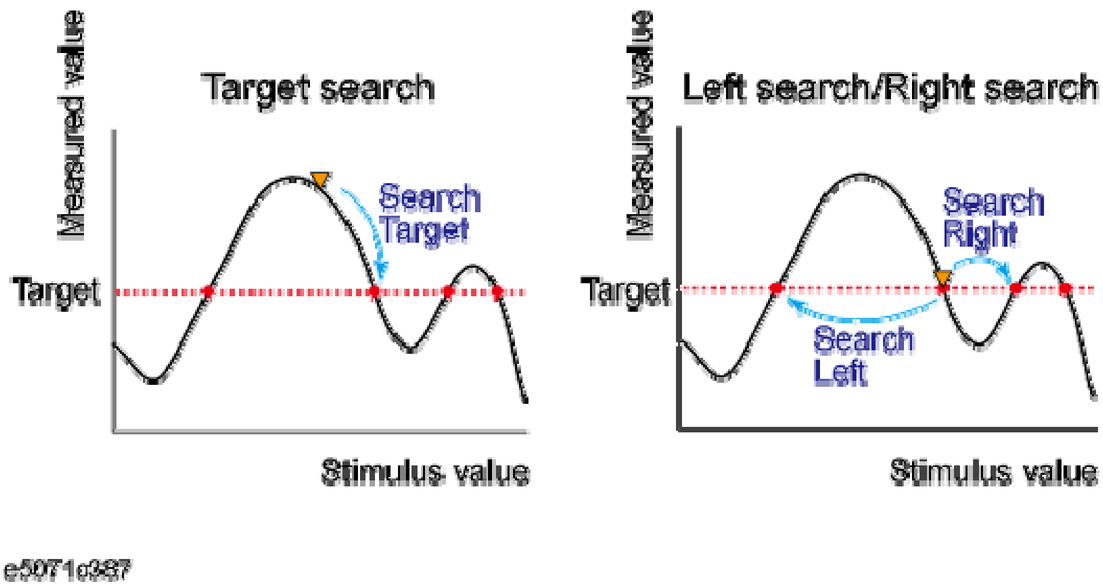


e5071c389

Executing a Target Search

The following three methods are available for executing the target search:

Target search (**Search Peak**)	The marker moves to the peak with maximum response value if the peak polarity is **Positive** or **Both** or to the peak with minimum response value if the peak polarity is **Negative**.
Search left (**Search Left**)	Executes the search from the current marker position to the smaller stimulus values and moves the marker to first target encountered.
Search right (**Search Right**)	Executes the search from the current marker position to the larger stimulus values and moves the marker to first target encountered.



Procedure

1. Activate the marker you are using for the target search.
2. Press **Marker Search** key.
3. Click **Target** > **Target Value**, then enter the target value.
4. Click **Target Transition**, then select a transition type.
5. Press the corresponding softkey to move the marker to the target.

NOTE When the data format is in Smith chart or polar format, execute the search for the main response value of the two marker response values.

NOTE Changing the settings of target value or transition type executes new search for multiple target.

Searching all target points at one time (Multiple Target)

The multi-target search is a function that searches for targets that match to pre-defined target value and transition type(s) (positive, negative, or both of positive and negative) and displays markers on the targets being searched.

Depending on the number of detected targets, markers 1 through 9 are displayed from the start frequency. The reference marker is not affected.

When the multi-target search is executed, search and tracking settings for markers 1 through 9 are ignored and the settings for the multi-target search are used.

Procedure

1. Press **Marker Search** > **Multi Target** > **Target Value**, then enter the target value.
2. Click **Target Transition**, then select a transition type.
3. Click **Search Multi Target** to show the markers on the multiple target.
4. Changing the settings of target value or transition type executes new search for multiple target.
5. Press **Marker** > **Clear Marker Menu** > **All OFF** to clear all markers.

NOTE Search Range is applied for multiple target search, however Multiple Range is not applied.

NOTE Search tracking is valid for multiple peak search.

Searching for the peak

The peak search function enables you to move the marker to the peak on the trace.

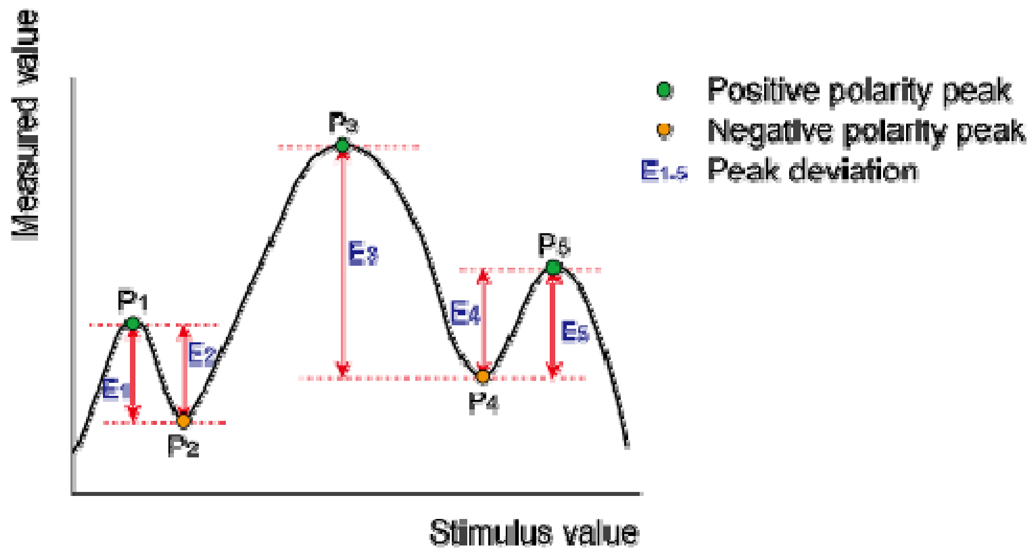
Definition of the peak

A peak is a measurement point whose value is greater or smaller than the adjoining measurement points on its right and left sides. Peaks are classified into the following two types depending on the difference in magnitude from the measurement points on either side of it.

Positive peak (**Positive**)	A peak whose measured value is greater than those of the measurement points on either side of it (peak polarity: positive)
Negative peak (**Negative**)	A peak whose measured value is smaller than those of the measuring points on either side of it (peak polarity: negative)

About Peak Excursion Value

The peak excursion value is the smaller of the differences in measured values from the adjoining peaks of the opposite polarity.

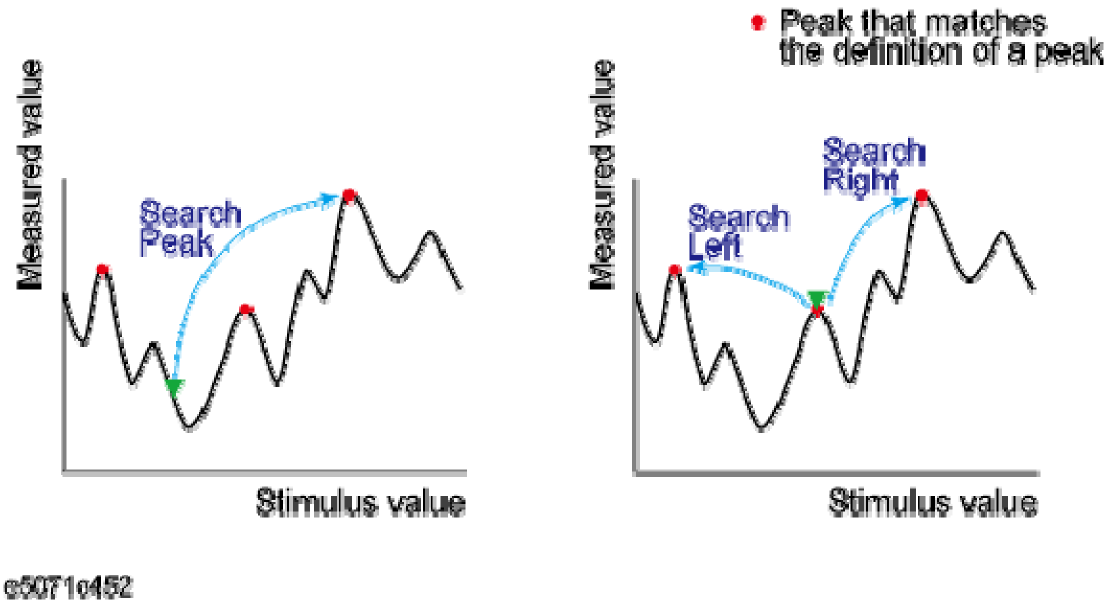


e5071c390

Executing a Peak Search

The following three methods are available for executing the peak search:

Peak search (**Search Peak**)	Moves the marker to the maximum peak when peak polarity is **Positive** or **Both.** Moves the marker to the minimum peak when peak polarity is **Negative.**
Left search (**Search Left**)	Executes the search from current marker position to the smaller stimulus values and moves the marker to first peak encountered.
Right search (**Search Right**)	Execute the search from current marker position to the larger stimulus values and moves the marker to first peak encountered.



Procedure

1. Activate the marker you are using for the peak search.
2. Press **Marker Search** key.
3. Click **Peak** > **Peak Excursion**, then enter the lower limit for the peak excursion value.
4. Click **Peak Polarity**, then select a peak polarity.
5. Click the corresponding softkey to move the marker to the peak.

NOTE When the data format is in Smith chart or polar format, execute the search for the main response value of the two marker response values.

NOTE Changing the settings of peak excursion value or peak polarity executes new search for multiple peak.

Searching all peaks at one time (Multiple Peak Search)

The multi-peak search function enables you to display markers on multiple peaks on traces. Depending on the number of detected peaks, markers 1 through 9 are displayed from the start frequency. The reference marker is not affected.

When the multiple peak search is executed, search and tracking settings for markers 1 through 9 are ignored and the settings for the multiple peak search are used.

Procedure

1. Press **Marker Search** > **Multi Peak** > **Peak Excursion**, then enter the lower limit for the peak excursion value.

2. Press **Peak Polarity**, then select a peak polarity.
3. Press **Search Multi Peak** to show the markers on the multiple peaks.
4. Changing the settings of peak excursion value or peak polarity executes new search for multiple peak.
5. Press **Marker** > **Clear Marker Menu** > **All OFF** to clear all markers.

NOTE Search Range is applied for multiple peak search, however Multiple Range is not applied.

NOTE Search tracking is valid for multiple peak search.

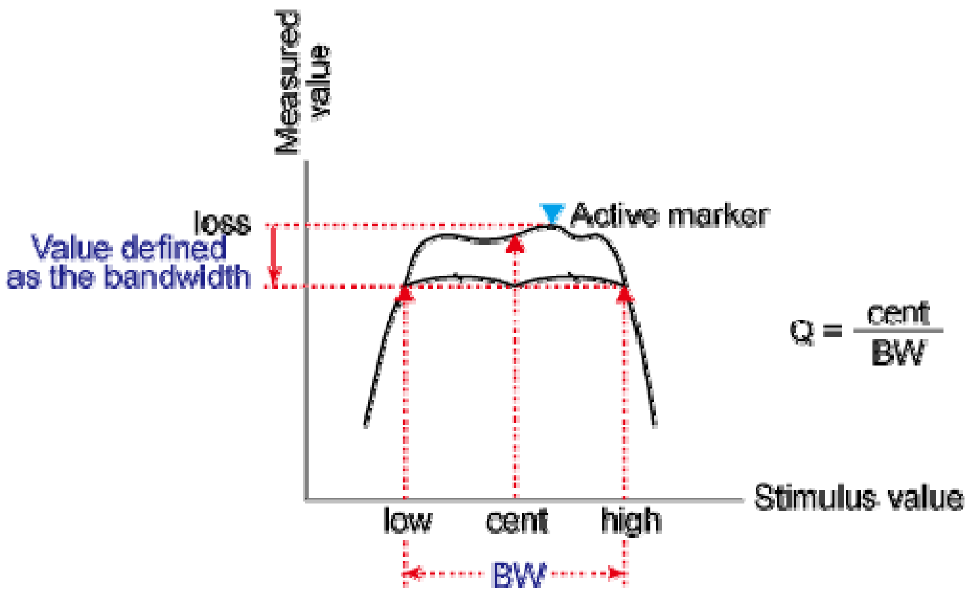
Determining the Bandwidth of the Trace (Bandwidth Search)

- Overview
- Executing a Bandwidth Search

Other topics about Data Analysis

Overview

The bandwidth search is a function for determining the bandwidth of the trace, center frequency, cut-off points (on the higher frequency and the lower frequency sides), Q, and insertion loss, based on the position of the *active marker* . The definitions of the parameters determined through the bandwidth search are shown in below.



e5071c392

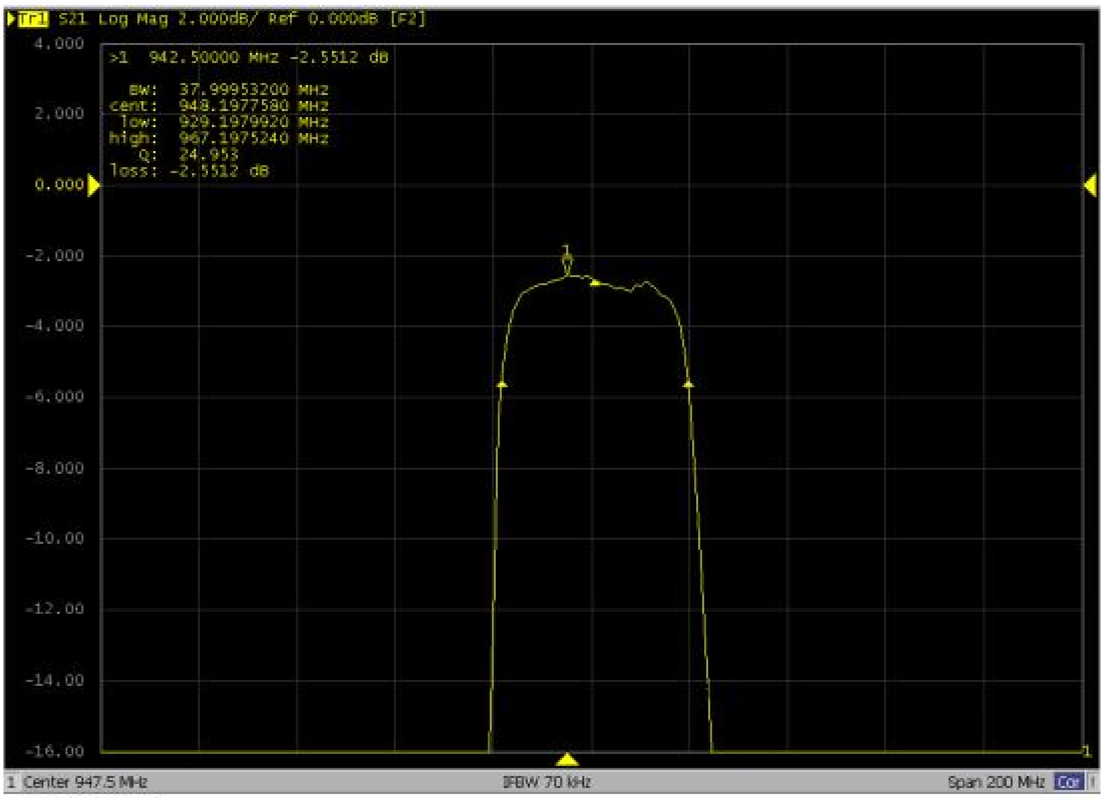
Bandwidth Parameter	Definition
Insertion loss (*loss*)	The measured value of the position of the active marker at the time the bandwidth search is executed.
Lower frequency cut-off point (*low*)	Lowest frequency of two measurement points, both separated by the defined bandwidth value from the active marker position.
Higher frequency cut-off point (*high*)	Highest frequency of two measurement points, both separated by the defined bandwidth value from the active marker position.

Center frequency (*cent*)	Frequency at the midpoint between the lower frequency cut-off and higher frequency cut-off points. (*high+low)/2*
Bandwidth (*BW*)	The difference in frequency between the higher frequency cut-off and lower frequency cut-off points. (*high-low)*
Q	Value obtained by dividing the center frequency by the bandwidth.(*cent*/*BW*)

Executing a Bandwidth Search

1. Place the active marker at the desired point on the trace on which the bandwidth search is executed. The response value of this active marker itself is the insertion loss in the bandwidth search (**loss**).
2. Press **Marker Search** key.
3. Click **Bandwidth Value** and enter the defined bandwidth value in the entry area that appears.
4. Click **Bandwidth** to turn on the bandwidth search. In the upper left of the trace display, six bandwidth parameters are displayed.

Bandwidth search results



e5071c077

Determining the Bandwidth of the Trace (Notch Search)

- Overview
- Executing a Notch Search

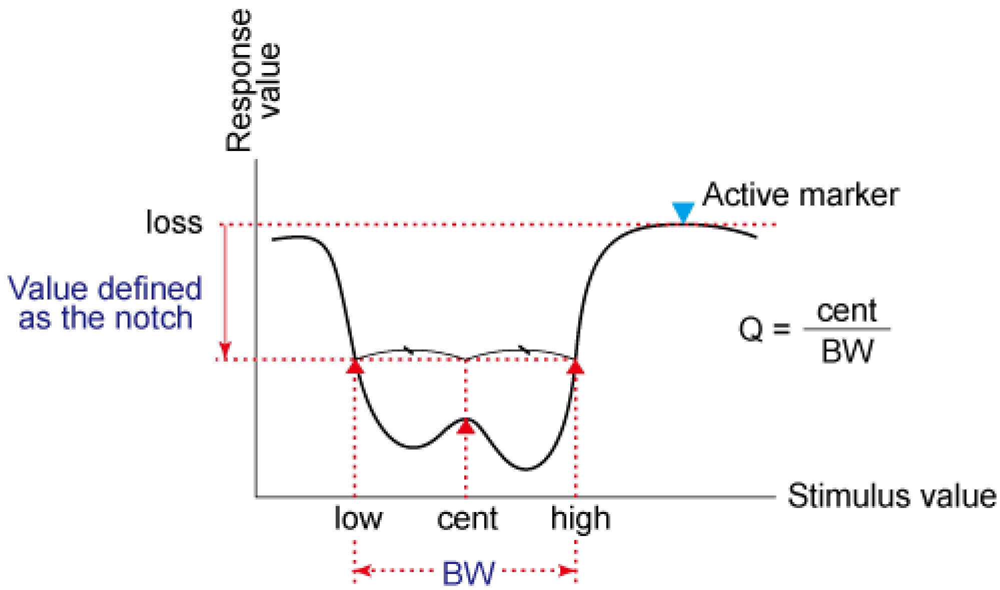
Other topics about Data Analysis

Overview

The notch search function is used to obtain the bandwidth, center frequency, cutoff points (high-frequency side and low-frequency side), Q, and insertion loss of a trace based on the active marker position. The notch search function starts from the left side of the active marker position, and ends when points that meet the conditions are found.

The figure and the table below shows the definition of parameters obtained by notch search function. The notch value in figure below must be specified by the user.

Bandwidth Parameters



e5071c484

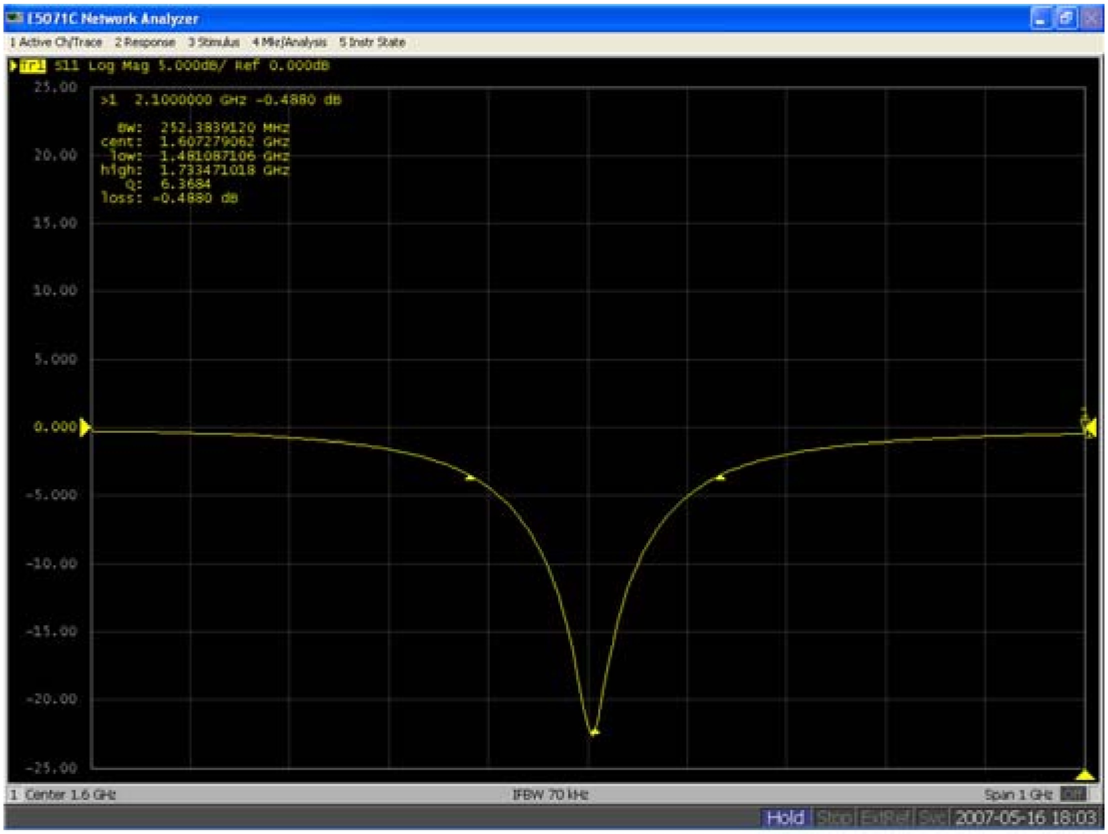
Definition of bandwidth parameters

Bandwidth parameter name	Definition
Insertion loss (loss)	Measurement value at the active marker position when the notch search is executed.

Lower cutoff point (low)	Lower frequency of the 2 points on both sides that have the measurement value apart from the active marker position by the notch value.
Higher cutoff point (high)	Higher frequency of the 2 points on both sides that have the measurement value apart from the active marker position by the notch value.
Center frequency (cent)	Frequency of the middle point between the lower cutoff point and the higher cutoff point (high + low)/2.
Bandwidth (BW)	Frequency difference between the higher cutoff point and the lower cutoff point (high – low).
Q	Value obtained by dividing the center frequency by the bandwidth (cent/BW).

Executing a Notch Search

1. Place the active marker on the desired point on the trace on which the notch search is executed. The response value of this active marker itself is the insertion loss in the notch search (loss).
2. Press **Marker Search.**
3. Press **Notch Value** and enter the notch value in the entry area that appears.
4. Press **Notch** to turn on the notch search. In the upper left of the trace display, six bandwidth parameters are displayed (see the figure below).



e5071c485

NOTE For more information on displaying the notch search result, see Notch Search

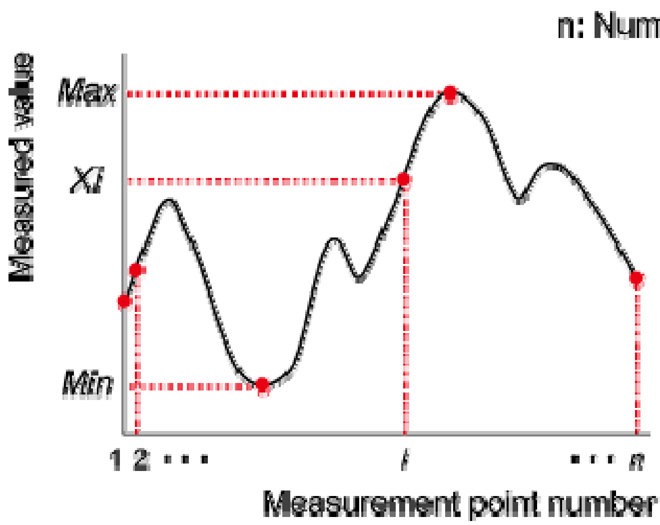
Determining the Mean, Standard Deviation, and p-p of the Trace

- Overview
  - Displaying Statistical Data

Other topics about Data Analysis

Overview

You can easily determine the statistics data for a trace (mean, standard deviation, and peak-to-peak). The definitions for the statistics data elements are shown below.



e5071c391

Statistics data element	Definition
Mean (*mean*)	$$\frac{\sum_{i=1}^{n} x_i}{n}$$ (n: number of points; x_i: measured value at the i-th measurement point)

Standard deviation (*s. dev*)	$$\sqrt{\frac{\sum_{i=1}^{n}(x_i - mean)^2}{n-1}}$$ (n: number of points; x_i: measured value at the i-th measurement point; mean: Mean)
Peak-to-peak (*p - p*)	Max - Min (Max: greatest measured value; Min: smallest measured value

Displaying Statistical Data

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace for which statistical data is required.
2. Press **Marker Fctn** key.
3. Click **Statistics** to turn on the display of statistics data.

Comparing Traces/Performing Data Math

- Overview
- Performing Data Math Operations

Other topics about Data Analysis

Overview

Each of the traces for which measured data is displayed is provided with an additional trace, called a memory trace, that temporarily stores measured data. You can use the memory trace to compare traces on the screen or to perform complex data math between the memory trace and measured data.

The following data math operations are available:

Value	Description
Data / Memory	Divides the measured data by the data in the memory trace. This function can be used to evaluate the ratio of two traces (e.g., evaluating gain or attenuation).
Data * Memory	Multiplies the measured data by a memory trace.
Data - Memory	Subtracts a memory trace from the measured data. This function can be used, for example, to subtract a vector error that has been measured and stored (e.g., directivity) from data subsequently measured on a device.
Data + Memory	Adds the measured data and the data in the memory trace.

Performing Data Math Operations

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace to be stored in memory.
2. Press **Display** key.
3. Press **Data -> Mem** to store the measured data in memory.
4. Press **Data Math.**
5. Select the data math operation to perform.
6. Press **Display**.
7. Select the type of data to display on the screen.
8. Send the trigger to make measurements.

Performing Parameter Conversion of Measurement Results

- Overview
- Procedure

Other topics about Data Analysis

Overview

You can use the parameter conversion function to convert the measurement results of the S-parameter (S_{ab}) to the following parameters.

- Equivalent impedance (Z_r) and equivalent admittance (Y_r) in reflection measurement

$$Z_r = Z_{0a} \times \frac{1 + S_{aa}}{1 - S_{aa}}, Y_r = \frac{1}{Z_r}$$

- Equivalent impedance (Z_t) and equivalent admittance (Y_t) in transmission measurement

$$Z_t = \frac{2 \times \sqrt{Z_{0a} \times Z_{0b}}}{S_{ab}} - (Z_{0a} + Z_{0b}), Y_t = \frac{1}{Z_t}$$

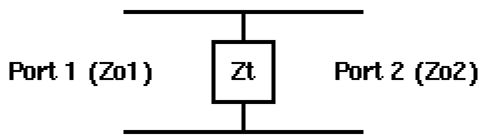
- Inverse S-parameter ($1/S_{ab}$)

where:

Z_{0a}: Characteristic impedance of port a

Z_{0b}: Characteristic impedance of port b

- Z/Y Transmission Shunt

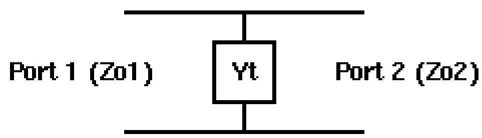


Z Transmission-Shunt

$$Zt = \frac{1}{Yt}$$

$$Yt = \frac{2\sqrt{Yo1 \cdot Yo2}}{S} - (Yo1 + Yo2)$$

$$Yo1 = \frac{1}{Zo1} \quad Yo2 = \frac{1}{Zo2}$$



Y Transmission-Shunt

- Conjugation

  Conjugation converts the measurement value to complex conjugate number.

When the fixture simulator function is ON and the port impedance function is ON, the value set in the port impedance conversion is used. In other cases, the system Z0 (preset value: 50 ohm) is used.

Procedure

ON/OFF

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate a trace on which you want to use the marker.
2. Press **Analysis** key.
3. Click **Conversion**.
4. Click **Conversion** to turn ON the conversion function.

Selecting Conversion Target Parameter

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate a trace on which you want to use the marker.
2. Press **Analysis** key.
3. Click **Conversion** > **Function**.
4. Click the softkey corresponding to the parameter to which you want to convert the result.

When the conversion function is ON, the selected parameter is displayed in Trace Status Area.

Limit Test

Other topics about Data Analysis

Overview

The limit test feature allows you to set the limit line for each trace and then perform the pass/fail judgment for the measurement result.

Concept of Limit Test

The limit test is a function to perform pass/fail judgment based on the limit line you set with the limit table.

In the limit test, if the upper limit or lower limit indicated by the limit line is not exceeded, the judgment result is pass; if it is exceeded, the judgment result is fail for all measurement points on the trace. Measurement points in a stimulus range with no limit line are judged as pass.

NOTE The targets of the pass/fail judgment are measurement points only. Parts interpolated between measurement points are not judged.

You define the limit line by specifying the stimulus value (Begin Stimulus) and response value (Begin Response) of the begin point, the stimulus value (End Stimulus) and response value (End Response) of the end point, and the type (lower limit/upper limit). For more information, refer to Defining the limit line.

When the limit test is on, measurement points that fail are displayed in red on the screen and the trace's pass/fail judgment result based on the results of individual measurement points (fail if one or more measurement points on the trace fail) is also displayed. You can check the pass/fail judgment result for the channel (fail if one or more traces fail in any of the limit test, the ripple test or the bandwidth test within the channel) on the screen as well. For more information, refer to Displaying judgment result of limit test.

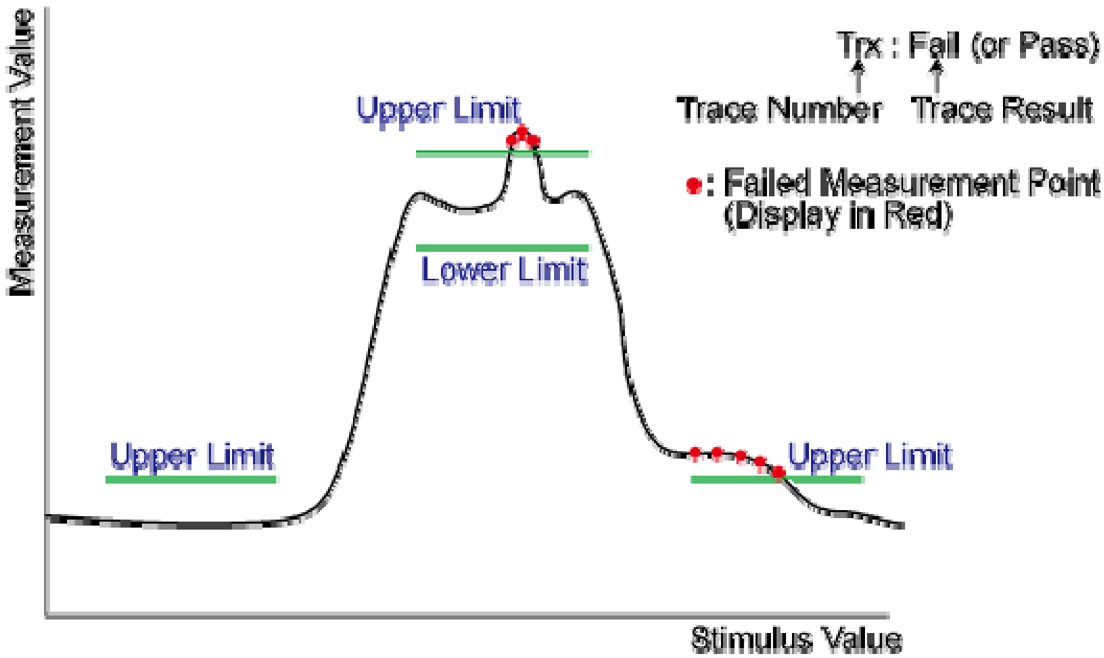
In addition to viewing the screen, you can check the judgment result of the limit test by the following methods.

- Beep that occurs when the judgment result is fail.
- Using the status register.

Displaying Judgement Result of Limit Test

Judgment result of measurement points and trace

Measurement points that fail are displayed in red on the screen. The judgment result of the trace is indicated by Pass or Fail displayed in the upper right part of the graph.



e50F1c446

Judgment result of channels

If a channel has a judgment result of fail, the message below appears on the screen (it will be judged as failed if one or more unsatisfactory trace exists in any of the limit test, the ripple test or the bandwidth test within the channel.)

e5070bue085

Follow these steps to turn on/off the display of the channel fail message.

1. Press **Analysis** key.
2. Click **Limit Test**.
3. Click **Fail Sign**. Each press toggles between on/off.

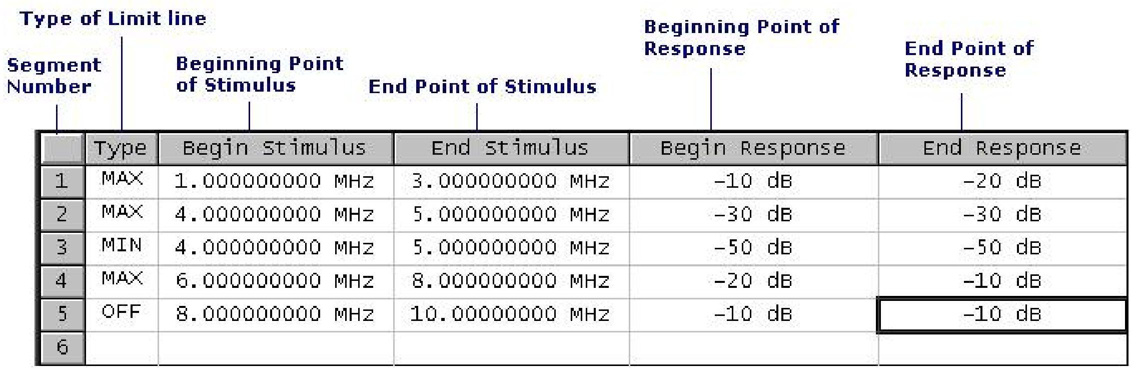
Defining Limit Line

To use the limit test, you must first define the limit line. You can define a limit table for each trace, and you can define up to 100 limit lines (segments) in a limit table.

Defining a segment

The following steps describe how to define a segment.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace on which the limit test function will be used.
2. Press **Analysis** key to display the Analysis menu.
3. Click **Limit Test** to display the softkeys associated with the limit test.
4. Click **Edit Limit Line** to display the limit table.



	Type	Begin Stimulus	End Stimulus	Begin Response	End Response
1	MAX	1.000000000 MHz	3.000000000 MHz	-10 dB	-20 dB
2	MAX	4.000000000 MHz	5.000000000 MHz	-30 dB	-30 dB
3	MIN	4.000000000 MHz	5.000000000 MHz	-50 dB	-50 dB
4	MAX	6.000000000 MHz	8.000000000 MHz	-20 dB	-10 dB
5	OFF	8.000000000 MHz	10.00000000 MHz	-10 dB	-10 dB
6					

e5071c068

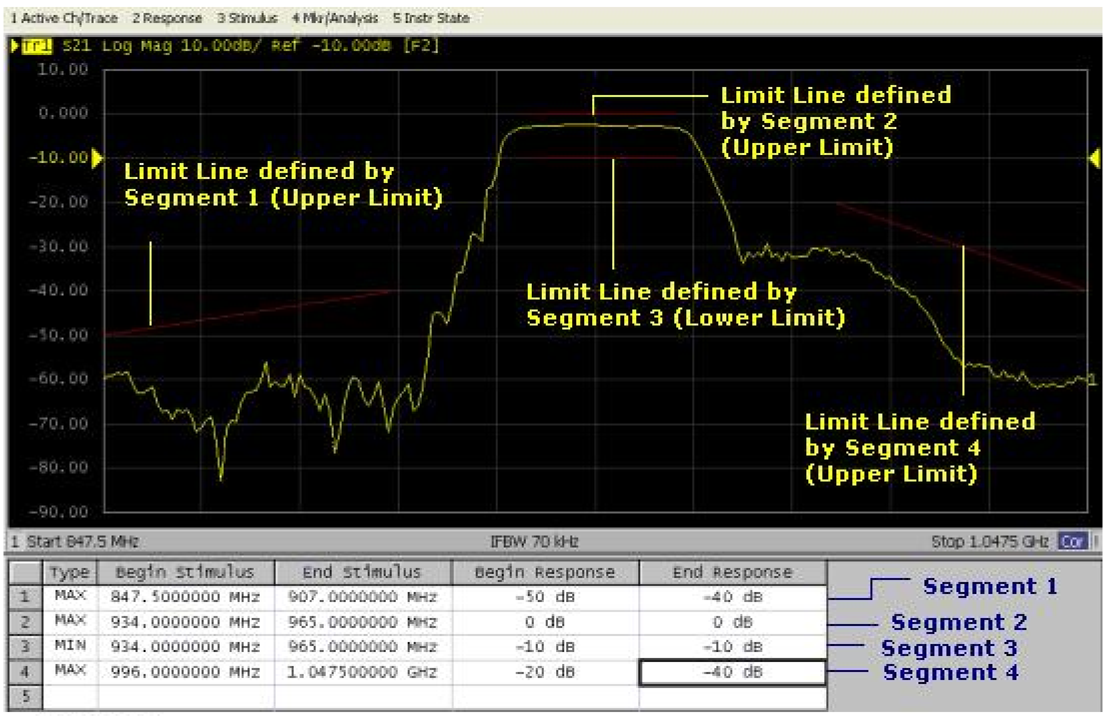
5. Using the limit table, create/edit a segment. Initially, no segments are entered in the limit table. At the same time, the Edit Limit Line menu used to create/edit the limit table is displayed.
6. Click **Add** to add a segment to the limit table and then specify the segment parameter values shown below.

Segment Parameter	Description	
Type	Select the type of segment from the following:	
	OFF	Segment not used for the limit test
	MIN	Segment at which the minimum is specified
	MAX	Segment at which the maximum is specified

Begin Stimulus	Specify starting point for stimulus value on the limit line
End Stimulus	Specify ending point for stimulus value on the limit line
Begin Response	Specify starting point for response value on the limit line
End Response	Specify ending point for response value on the limit line

NOTE The range in which stimulus values can be specified is from -500 G to +500 G. When a value outside the range is entered, a suitable value within the range is specified. Once the stimulus value is specified, changing the sweep range of the E5071C does not affect the stimulus value.

NOTE The range in which response values can be specified is from -500 M to +500 M. When a value outside this range is entered, a suitable value within the range is specified. After the response value is specified, changing formats results in changing the units but not the value.



e5071c078

NOTE You can define a limit line that is able to freely overlap the stimulus range of another limit line.

Defining one limit line having the same type as a second limit line whose stimulus range overlaps with the first one results in two or more limit values at the same measurement point. In this case, the limit value to be used in the limit test is defined as follows:

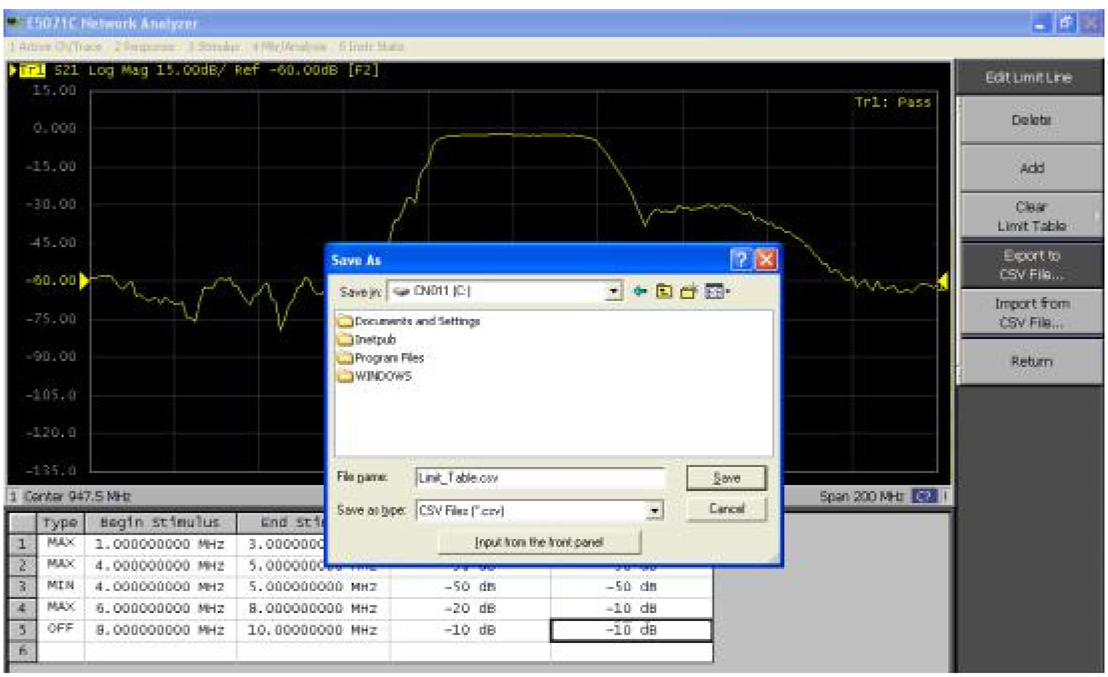
- When two or more limit values whose type is set to maximum (MAX) exist, the smallest one is used as the maximum.
- When two or more limit values whose type is set to minimum (MIN) exist, the largest one is used as the minimum.

NOTE Even if the span of the sweep range on the E5071C is set to 0, enter the two parameters of Begin Stimulus and End Stimulus.

NOTE When two or more response values are returned as a result of using the Smith or polar chart format, the first response value of the marker provides the object of the limit test.

Saving/calling the limit table

You can save the limit table to a file that you can then freely bring up on the screen later and use. You can import a file saved in CSV format (extension: *.csv) into spreadsheet software on a PC for later use (a numerical value will be saved as strings that include its unit).



e5071c069

1. Display the limit table.
2. In the Edit Limit Line menu, press **Export to CSV File** to open the Save As dialog box. In this step, CSV (extension: *.csv) is selected as the file type

3. Specify the folder in which to save the file and enter the file name. Press **Save** to save the limit table displayed on the screen to the file.
4. Conversely, to recall a saved limit table, press **Import from CSV File** in the Edit Limit Line menu to display the Open dialog box. In this step, CSV (extension: *.csv) is selected as the file type.
5. After specifying the folder containing the file, select the file. Press **Open** to display the limit table on the screen.
   6. The limit table can be called from any trace of any channel, regardless the channel or trace as of saving in a file.

Limit Table Saved in CSV Format

The limit table is saved in the following format.

- On the first line, the channel number of the active channel that was valid when the file was saved is output.
- On the second line, the trace number of the active trace that was valid when the file was saved is output.
- The third line provides the header showing the items for the segments to be output on the fourth and later lines.
- Data on segments are output on the fourth and later lines.

"# Channel 1"

"# Trace 1"

Type, Begin Stimulus, End Stimulus, Begin Response, End Response

MAX, 200.0000000 MHz, 400.0000000 MHz, -100 dB, -100 dB

MAX, 490.0000000 MHz, 510.0000000 MHz, -10 dB, -10 dB

MIN, 490.0000000 MHz, 510.0000000 MHz, -20 dB, -20 dB

MIN, 600.0000000 MHz, 800.0000000 MHz, -100 dB, -100 dB

Turning the limit test ON/OFF

You can set the limit test ON/OFF for each trace individually.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace on which the limit test function will be used.
2. Press **Analysis** key to display the Analysis menu.
3. Press **Limit Test** to display the Limit Test menu.

Softkey	Function
Limit Test	Sets the limit test ON/OFF.
Limit Line	Sets the limit line display ON/OFF.

Edit Limit Line	Opens the limit table for editing the limit line. See Defining_the_limit_line.

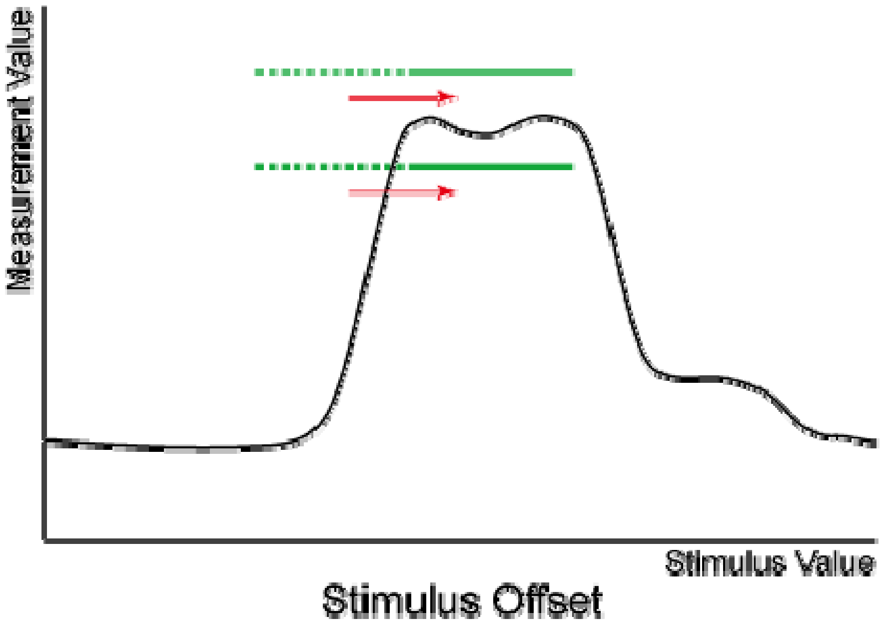
Limit line offset

By adding a certain offset to the limit value, you can adjust the limit line so that it conforms to the device output.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace on which the limit test function will be used.
2. Press **Analysis** key to display the Analysis menu.
3. Click **Limit Test** to display the softkeys for the limit test.
4. Click **Limit Line Offsets** to display the limit line offset function menu. The following functions correspond to each softkey.

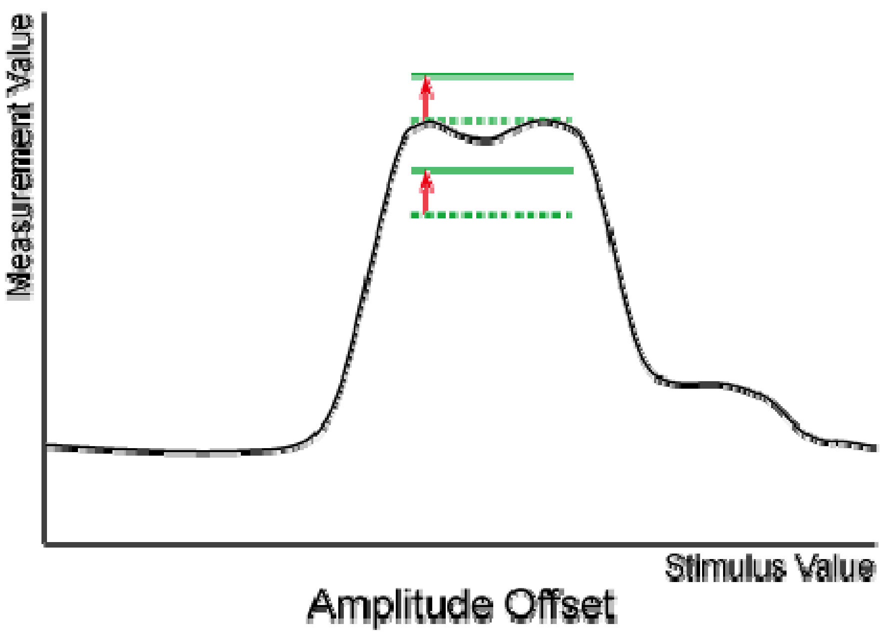
Softkey	Function
Stimulus Offset	Adds a certain offset to the stimulus value of the entire segment in the limit table. (Stimulus offset)
Amplitude Offset	Adds a certain offset to the response value of the entire segment in the limit table. (Amplitude offset)
Marker -> Amp. Ofs.	Adds the amplitude offset by the same amount as the retrieved value of the *active marker* . You can confirm the current value set for the amplitude offset by pressing **Amplitude Offset**. (Marker amplitude offset)

Stimulus offset



e5071c447

Amplitude offset



e5071c448

Initializing the limit table

The following operations initialize the limit table.

- At power-on
- When presetting
- When calling a limit table with zero segments
- When **Clear Limit Table** - **OK** is pressed in the Edit Limit Line menu

Bandwidth Test

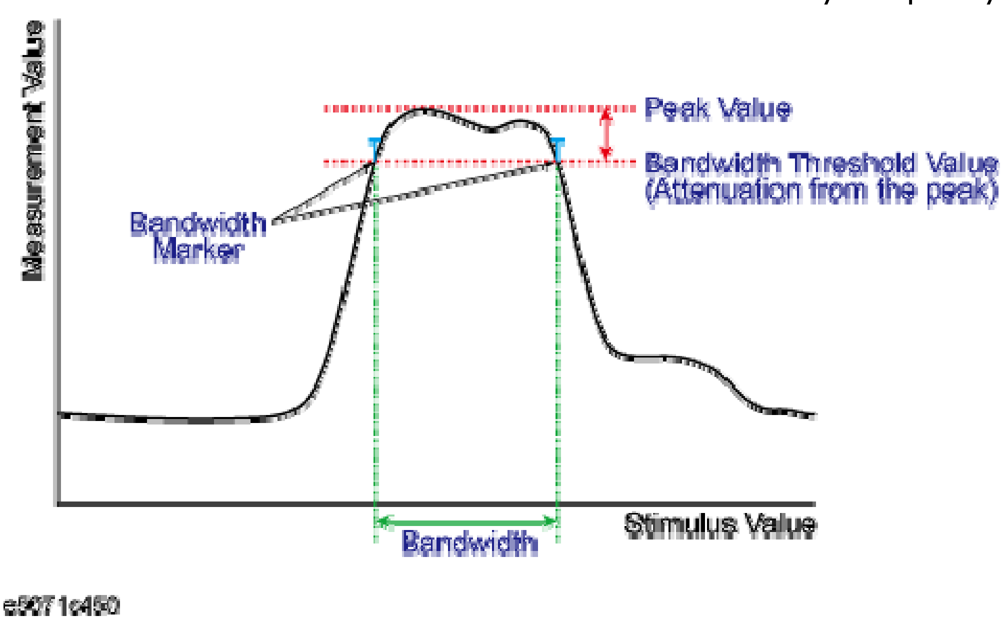
- Overview
- Displaying Bandwidth Test Results
- Bandwidth Test Setup
- Turning Bandwidth Test and Displaying Results ON/OFF

Other topics about Data Analysis

Overview

The bandwidth test function can be used for testing bandwidth for the band-pass filters.

The bandwidth test find the peak of a signal in the passband and locates a point on each side of the passband at an amplitude below the peak specified in test setup. The frequency between these two points is the bandwidth of the filter. Then the obtained bandwidth is compared to minimum and maximum allowable bandwidth that you specify beforehand.

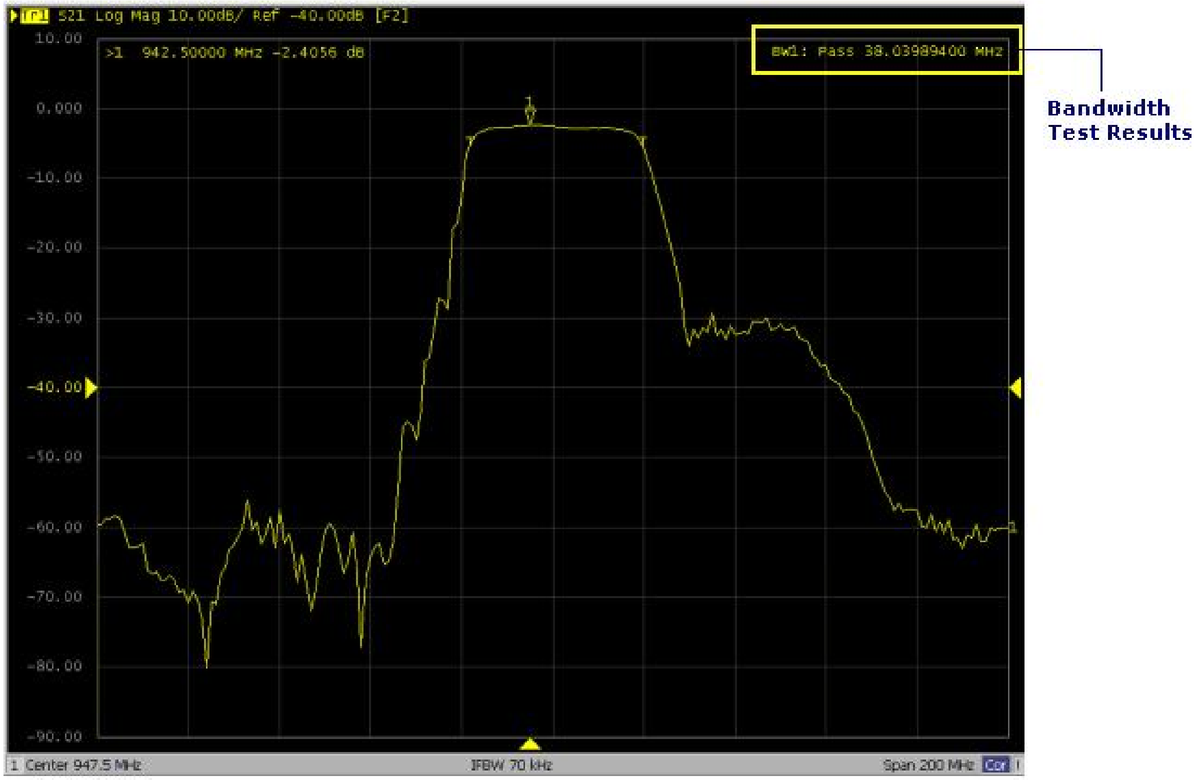


Displaying Bandwidth Test Results

Test Result for Trace

The test result of the trace will be indicated in the upper-right area of the graph for each trace, following BWn:. "n" denotes the trace number. The results are shown as Pass, Wide, Narrow, or >Span (Fail). You can also display the bandwidth value.

For information on how to display the results, see Turning On/Off Bandwidth Test and Displaying Results.



e5071c076

Test Results for Channel

If any channel is unsatisfactory, the message will be displayed as shown in Judgment result of channels. (It will be judged as failed if one or more failed traces are found for the limit test, ripple test, or bandwidth test within the channel.)

You can also specify this On/Off setting from the Fail Sign, which is provided in the limit test menu and ripple test menu. From the bandwidth test menu, follow the steps below to turn it on/off.

1. Press **Analysis** key.
2. Click **Bandwidth Limit**.
3. Click **Fail Sign**. This menu toggles between on and off.

In addition to the screen, the following features also let you confirm the test results:

- Beep notifying the result was unsatisfactory
- Status register

Bandwidth Test Setup

You must set up the bandwidth threshold and the upper and lower limits before you can use the bandwidth test function. You can specify the threshold, upper limit, and lower limit for each trace.

Operational procedure

Follow the steps below to set up the bandwidth test.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace to which you want to apply the bandwidth test function.
2. Press **Analysis** key to display the Analysis menu.
3. Click **Bandwidth Limit** to display the softkeys for the bandwidth test.
4. Click **N dB Points** to specify the bandwidth threshold. The unit is dB.
5. Press **Min Bandwidth** to enter the lower limit for the bandwidth. Similarly, press **Max Bandwidth** to enter the upper limit for the bandwidth. The unit is Hz for both Min and Max bandwidths.

   NOTE If the data format is Smith chart or polar, the limit test is performed for the main response value among the two marker response values.

Turning Bandwidth Test and Displaying Results ON / OFF

You can configure the on/off setting of the bandwidth test function for each trace independently.

Operational procedure

Follow the steps below to configure the on/off setting of the bandwidth test.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** keys to activate the trace to which you want to apply the bandwidth test function.
2. Press **Analysis** key to display the Analysis menu.
3. Click **Bandwidth Limit** to display the softkeys for the bandwidth test.
4. Click **BW Test** and turn *ON* the bandwidth test. You can display the bandwidth marker on the screen by turning **BW Marker** *ON*. To display the bandwidth value, turn **BW Display** *ON*.

Ripple Test

Other topics about Data Analysis

Overview

Independently of the limit test, you can evaluate the measurement results on a pass/fail basis by setting a limit for the ripple. This function is called the ripple test.

Concept of Ripple Test

The ripple test is a function for evaluating the results on a pass/fail basis based on the ripple limit, which is set using the ripple limit table. You can specify up to 12 frequency bands, which permits a test for each frequency band.

The ripple test judges the measurement as "Pass" when the ripple value specified with the ripple limit is not exceeded by any of the measurement points on the trace; otherwise, it judges the measurement as "Fail." For the measurement points in a stimulus range without a specified ripple limit, the test judges the measurement as "Pass."

> NOTE The measurement point alone is the target of evaluation for pass/fail. The interpolated part between measurement points is not evaluated.

The ripple limit is defined with the start point stimulus value, end point stimulus value, ripple limit value, and type (on/off.) For detailed information, see Configuring ripple limit.

While the ripple test function is turned on, the measurement points corresponding to a "fail" judgment will be indicated in red on the screen, and the trace's test results based on the results of each measurement point will be displayed (judged as "fail" if one or more red measurement point exist on the trace). For information on how to display the results, see Turning on/off ripple test and displaying results. You can also confirm the channel test results on the screen (judged as "fail" if one or more failed traces appear in the limit test, ripple test, or bandwidth test within the channel).

Displaying Ripple Test Results

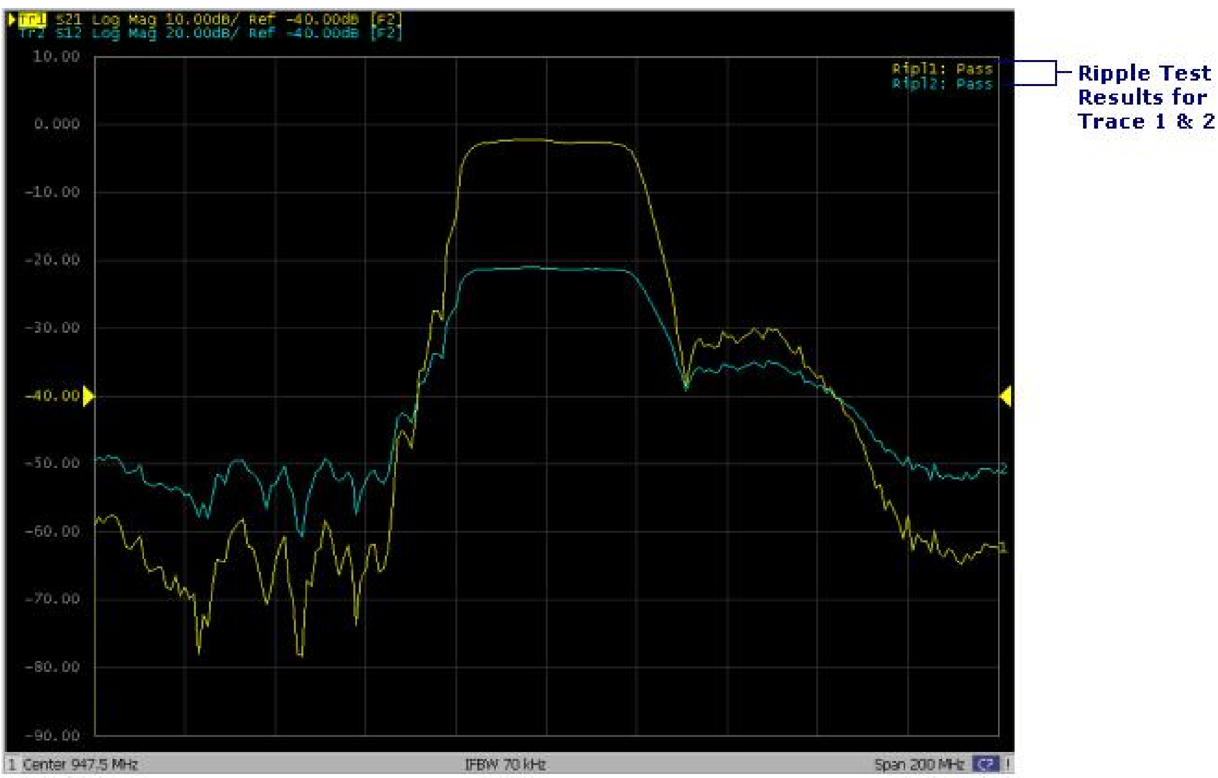
Measurement point and test results

Failed measurement points will be displayed in red on the screen. The test result for the trace will be indicated as Pass or Fail in the upper-right area of the graph. You can also display the ripple value at the selected frequency band.

The result will be displayed as Ripln:Pass (or Fail) for each trace. n denotes the trace number. Bn will be followed by the ripple value (if the ripple display is turned off, only Bn will be displayed without the ripple value).

For example, in the following figure, Ripl1:Pass in the first line indicates the result for trace 1. The value following B3 is the ripple value at the third frequency band specified in the ripple test. Similarly, the second line indicates the test result for trace 2, showing the ripple value at the first frequency band.

For information on how to display the results, see Turning on/off ripple test and displaying results.



e5071c070

Test result for channel

If a channel has a judgment result of "fail," the message will be displayed as shown in Test Result for Trace. (It will be judged as failed if one or more failed traces are found in the limit test, ripple test, or bandwidth test within the channel.)

e5071c071

You can also specify the On/Off setting for the Fail Sign in the limit test menu and as well as in the bandwidth test menu. From the ripple test menu, follow the steps below to turn it on/off.

1. Press **Analysis** key.
2. Click **Ripple Limit**.
3. Click **Fail Sign**. This menu toggles between on and off.

In addition to the screen, the following features also let you confirm the test results:

- Beep notifying that the result was "fail"
- Status register

Configuring Ripple Limit

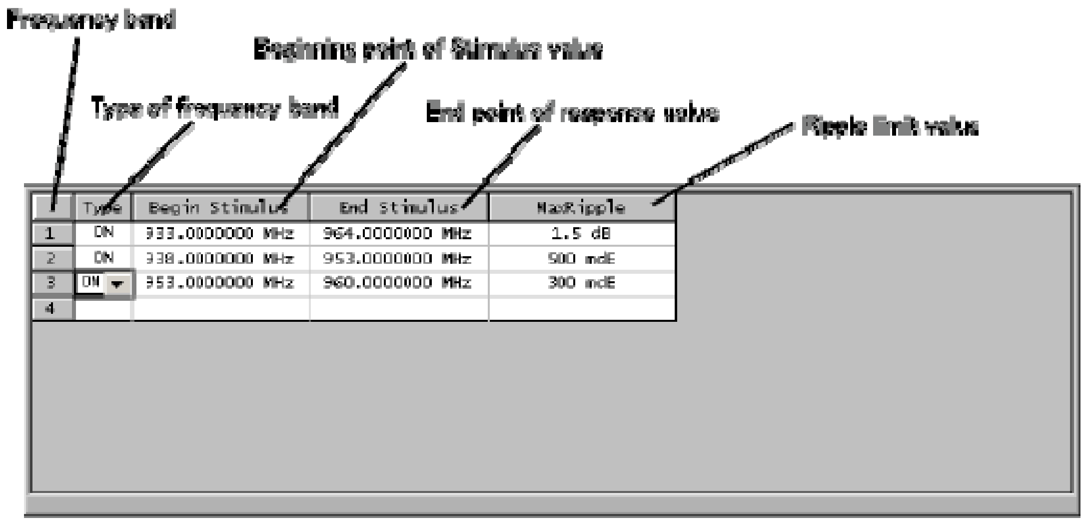
You must configure the ripple limit before you can use the ripple test function. You can specify a ripple limit table for each trace, where up to 12 ripple limit bands (frequency bands) can be configured.

Operational procedure

Follow the steps below to configure the ripple limits.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace on which you want to apply the ripple test function.
2. Press **Analysis** key to display the Analysis menu.

3. Click **Ripple Limit** to display the softkeys for to the ripple test.
4. Click **Edit Ripple Limit** to display the ripple limit table shown below.



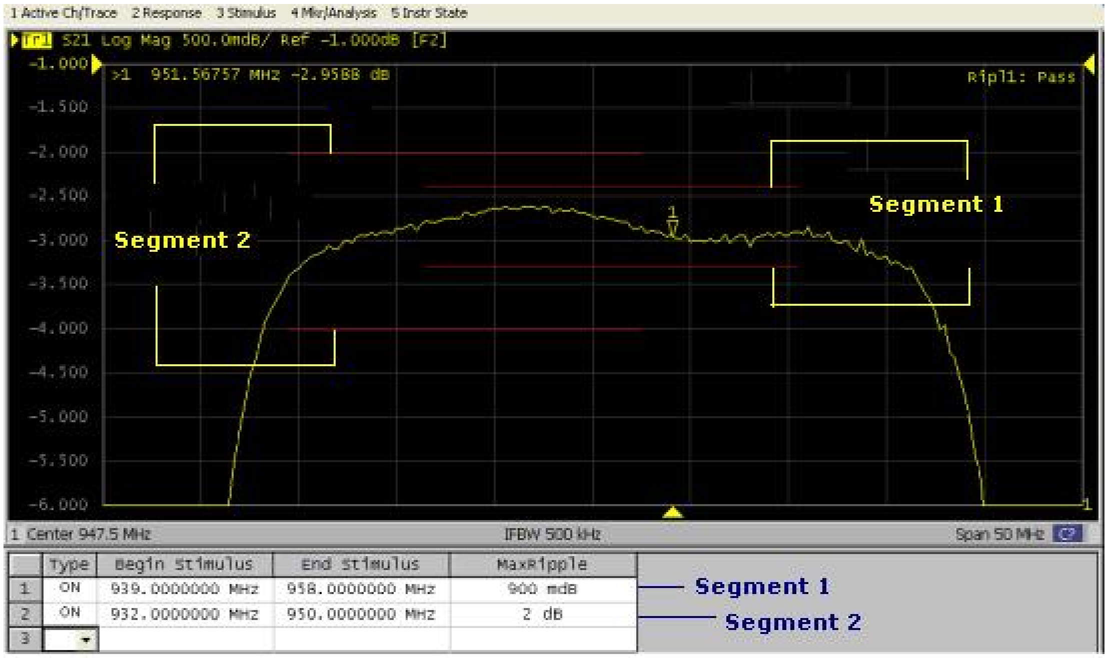
5. Create or edit the bands by using softkeys. Note that no frequency band is provided in the ripple limit table by default. The Edit Ripple Limit menu is also displayed, which can be used for creating or editing the frequency band.
6. Click **Add** to add a frequency band to the ripple limit table and then specify the following parameters for the frequency band:

Parameter	**Description**
Type	Selects a frequency type, either ON or OFF. **ON** Band used for the ripple test. **OFF** Band not used for the ripple test.
Begin Stimulus	Specifies the start point for the stimulus value in the ripple test.
End Stimulus	Specifies the end point for the stimulus value in the ripple test.
Max Ripple	Specifies the ripple limit value.

NOTE Acceptable range for the stimulus value: -500G to +500G. If any outranging value is specified, it will be reset to fall within the range.

NOTE Even if the E5071C's sweep range is changed after the stimulus value has been set, the stimulus value is not susceptible.

Example of ripple limit configuration



e5071c079

NOTE The individual frequency bands for the ripple test can overlap each other; in this case, the ripple limit test is performed for each frequency band.

NOTE Even if the E5071C's span value is set to zero, you must enter a parameter for both Begin Stimulus and End Stimulus.

NOTE If the data format is Smith chart or polar, the limit test is performed for the main response value among the two marker response values.

Saving / Recalling Ripple Limit Table

The ripple limit table can be saved in a file and recalled later for use on the screen. The file is saved in the csv format (with the extension *.csv), and values are saved as a character string with the unit. The csv formatted file can also be reused in spreadsheet software made for PCs.

Follow the steps below to save/recall the ripple limit table. This operation should be done by using the external keyboard and/or mouse.

1. Display the ripple limit table.

2. Click **Export to CSV File** from the Edit Ripple Limit menu to open the dialog box. At this time, CSV Files (with the extension *.csv) is selected as the file type.
3. Specify any folder in which you want to save the file, and enter the file name. Click **Save** to save the ripple limit table displayed on the screen to a file.
4. To recall the saved ripple limit table, click **Import from CSV File** from the Edit RippleLimit menu to display the Open dialog box. At this time, CSV Files (with the extension *.csv) is selected as the file type.
5. Specify the folder that contains the file, and then select the file. Click **Open** to recall the saved limit table on the screen.

NOTE You can recall a limit table from a trace on any channel independently of the channel and trace that were active when the limit table was saved to the file.

The ripple limit table is saved in the following format:

- In the first line, the channel number for the active channel at the time of file saving will be output.
- In the second line, the trace number for the active trace at the time of file saving will be output.
- The third line is a header indicating the segment items that are output from the fourth line onward.
- From the fourth line onward, the segment data are output.

Operational procedure

"# Channel 1"

"# Trace 1"

Type, Begin Stimulus, End Stimulus, MaxRipple

ON, 933.0000000 MHz, 964.0000000 MHz, 1.5 dB

ON, 938.0000000 MHz, 953.0000000 MHz, 500 mdB

ON, 953.0000000 MHz, 960.0000000 MHz, 300 mdB

Turning ON/OFF Ripple Test and Result Display

You can set the limit test ON/OFF for each trace individually.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace on which you want to apply the ripple test function.
2. Press **Analysis** key to display the Analysis menu.
3. Click **Ripple Limit** to display the softkeys for to the ripple test.
4. Click the following corresponding softkeys.

Softkey	Function
Ripple Test	Sets the ripple test ON/OFF.
Ripple Limit	Sets the ripple limit line display ON/OFF.
Ripple Value	Sets how the ripple values are displayed. Available settings are off, absolute value (difference between maximum and minimum values within the band) display, and margin (difference between absolute value of ripple and ripple limit) display.
Ripple Value Band	Selects the band for which you want to display the ripple value.
Edit Ripple Limit	Opens the ripple limit table for editing the ripple limit. To use the ripple test function, you must first define the ripple limit.

Initializing Limit Table

The following operations initialize the limit table.

- At power-on
- When presetting
- When calling a Ripple table with zero segments
- When **Clear Limit Table** > **OK** is clicked in the Edit Ripple Line menu

Equation Editor

Other topics about Data Analysis

Overview

Equation Editor allows you to enter an algebraic equation of standard mathematical operators and functions, referencing data that is available in the E5071C. 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, on entering the equation “Example=S21/(1-S11)” in the E5071C Equation Editor (**4** in the Figure below), 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, if you want the ENA to make a directivity measurement of your 3-port DUT, then the desired result is the sum and difference of LogMag formatted traces, which can be expressed as: S12 + S23 - S13.

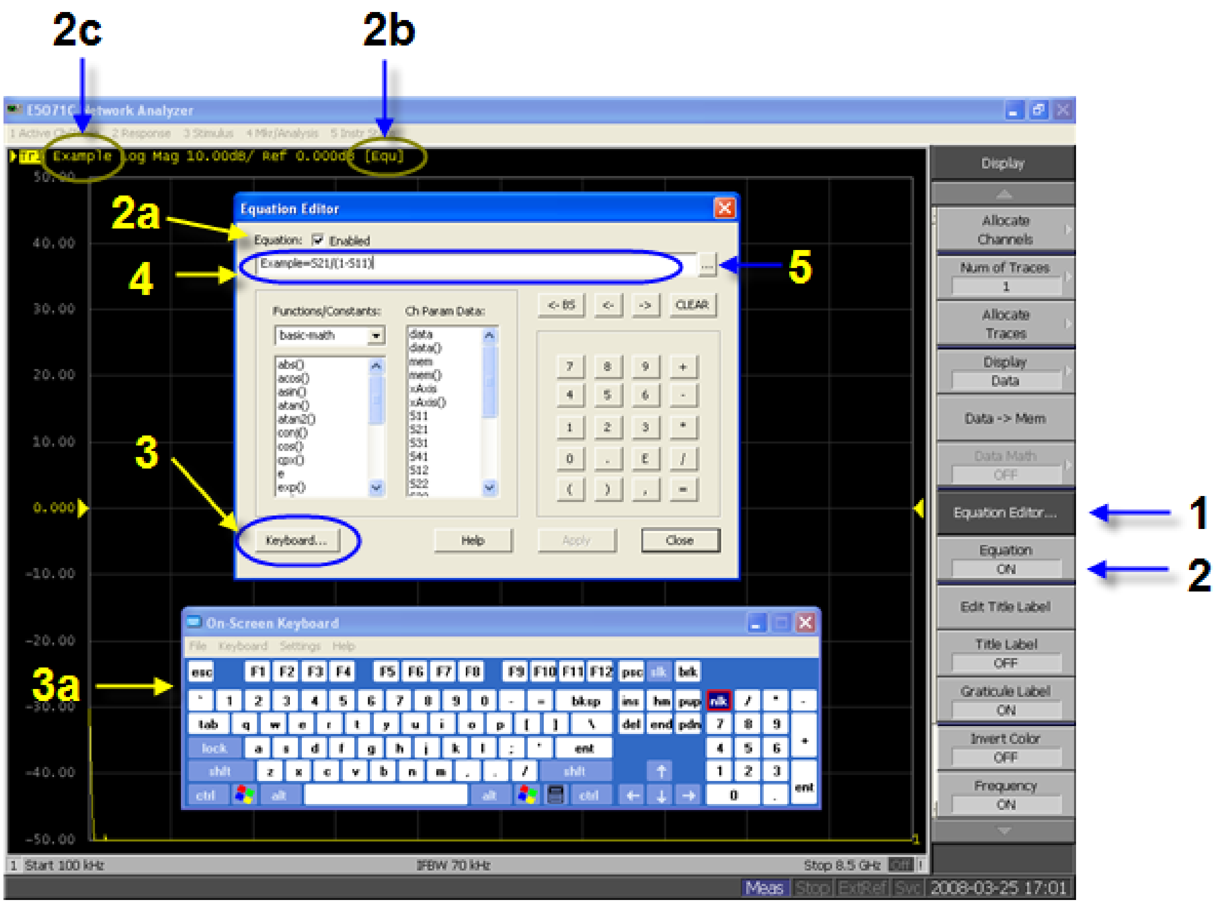
Because Equation Editor operates on unformatted complex data, the required equation is:

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.

Using Equation Editor

The step-by-step procedure of using Equation Editor is described below:



e5071c282

1. Select a trace in which you want to enter the equation and activate the trace.
    2. Activating a trace is required as Equation Editor works on traces.
2. Follow the steps below to enter an equation:
    1. Press **Display**.
    2. Click **Equation Editor** (**1** in the figure above). The Equation Editor dialog box appears.
    3. Enter an equation in the equation field (**4** in the figure above).

    NOTE Referring to traces in a different channel is NOT available with Equation Editor on the ENA.

    NOTE The equation can be entered with the software keyboard enabled by selecting **Keyboard...** (**3** and **3a** in the figure above).

3. Follow the steps below to apply the defined equation. When a valid equation is entered, the Equation Enabled check box becomes available for checking.
    1. Check **Equation Enabled** check box (**2a** in the figure above).
    2. Click **Apply**. The equation becomes visible and annotation of [Equ] (**2b** in the figure above) is displayed in the trace title area.
    3. Click **Close** to hide the dialog box.
4. The equation can also be applied by selecting **Display** > **Equation ON**.

NOTE If error correction is not turned on, then the raw, uncorrected data is used in the equation trace.

NOTE If an equation is NOT valid (i.e. referring to a trace that is not measured in the measured channel), annotation of [Equ!] is displayed in the trace title area instead.

Equation Editor Examples

The following examples may help you in getting started with Equation Editor:

Offset each data point in trace 2 from trace 1 by XdB

The equation is entered into trace 2 as "**Offset= data(1)*pow(10,X/20)**" (**4** in the figure e5071c282).

To multiply a trace by X-times in log format

The equation is entered as "**Example_Trace= pow(data,X)**" (**4** in the figure e5071c282).

Differential Return Loss using a 2-port ENA

The equation is entered as "**Sdd11= (S11-S21-S12+S22)/2)**" (**4** in the figure e5071c282).

NOTE For detailed on mixed-mode S-parameters on the 2-port ENA, refer to the related FAQ on the Agilent website.

CMRR of a Balanced Component

The equation is entered as "**CMRR = data(2) / data (1)**" (**4** in the figure e5071c282) where:

data (1) - a trace that measures Scc21 with Fixture Simulator

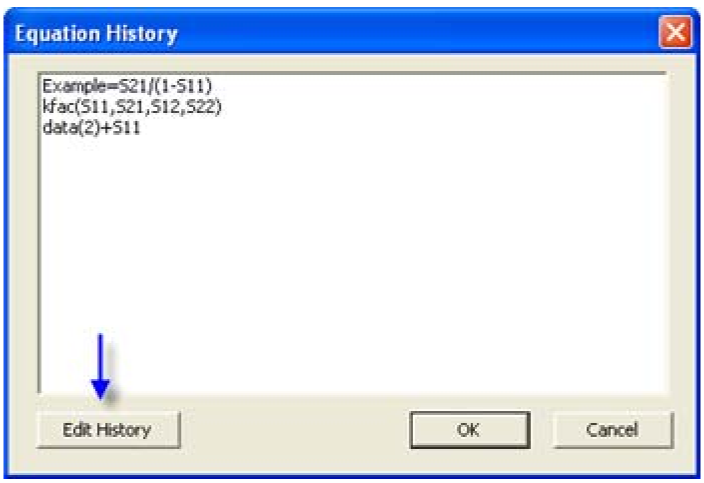
data (2) - a trace that measures Sdd21 with Fixture Simulator

Equation History

Equation Editor has the capability to save and recall all previously defined equations, which can be accessed by clicking the **...** button (shown as **5** in the figure e5071c282). All equations can be viewed in the **Equation History** dialog box.

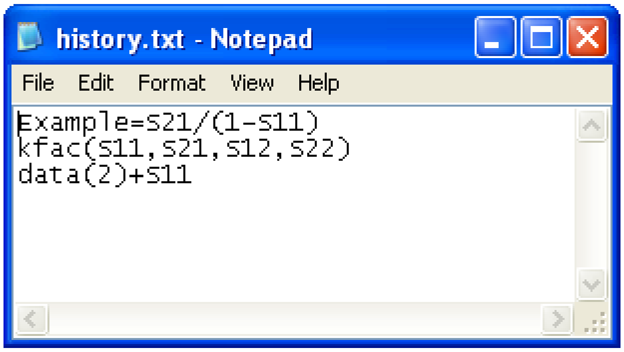
To view the equations in the list, follow this procedure:

1. Open Equation Editor by **Display** > **Equation Editor**
2. Enter an equation and click **Apply** in the Equation Editor dialog box (Figure e5071c282) to save the defined equation in the directory of the ENA. To view a list of saved equations, click the ... button (**5** in Figure e5071c282) to open the Equation History dialog box.



3. To store an equation in the History List, the equation must be applied first. This can be done by clicking on the **Apply** button.

3. To edit the equations in the list, click **Edit History**. The text file of **history list** is opened with Notepad.



4. The **History** List is stored as a text file **D:\Agilent\Equation\history.txt** and can save a maximum of 50 lines (equations) with a maximum of 254 character per line (equation).

Functions and Constants

The following table describes the different functions and constant available in the E5071C Equation Editor. In the following table:

- Function(scalar x) means that the function requires a scalar value. If a complex value is entered, it is automatically converted to a scalar value; complex(x,y) -> scalar(x)
- Function(complex x) means that the function requires a complex value. If a scalar value is entered, it is automatically converted to a complex value; scalar(x) -> complex(x, 0)
- **a,b** are arguments that are used in the function.

Basic Math Functions

Function	Description
abs(complex **a**)	returns the sqrt(a.re*a.re+a.im*a.im)
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(complex **a**)	returns the phase of a = (re, im) in radians
atan2(scalar **a**, scalar **b**)	returns the phase of (a, b) in radians
conj(complex **a**)	returns the conjugate of a
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
exp(complex **a**)	returns the exponential of a
im(complex **a**)	returns the imaginary part of a as the scalar part of the result (zeroes the imaginary part)
ln(complex **a**)	returns the natural logarithm of a
log10(complex **a**)	returns the base 10 logarithm of a
mag(complex **a**)	returns sqrt(a.re*a.re+a.im*a.im)
phase(complex **a**)	returns atan2(a) in degrees
pow(complex **a**,complex **b**)	returns a to the power b
re(complex **a**)	returns the scalar part of a (zeroes the imaginary 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
Constants	
e	2.71828182845904523536
PI	3.14159265358979323846

- Mutual transformation is automatically made for scalar and complex.

  scalar(x) -> complex(x, 0)
  complex(x, y) -> scalar(x)

Advanced Math Functions

In the following table: **a**,**b**,**c**,**d** are arguments of complex value that are used in the function. For 2-port network measurement: **a**,**b**,**c**,**d** corresponds to Sii,Sji, Sij, Sjj respectively.

NOTE The functions can also be defined by scalar arguments with port numbers of the ENA. For example, the function, kfac(1,2) returns the K-factor of 2-port measurement between port 1 and port 2 of the ENA.

NOTE When a scalar argument of the port number in the function is out of range (i.e. mu1(1,3) for the 2-port ENA), the ENA indicates an error message, "Equation runtime error" and equation is not reflected on the trace.

Z0 means the system characteristic impedance (Z0) which can be accessed under Cal.

NOTE Although the port impedance for each port can be changed with the port impedance conversion function by Fixture Simulator, the change is not reflected on the computed trace by Equation Editor.

Function	Description
A(complex **a**, complex **b**, complex **c**, complex **d**)	F_{11} conversion = ((1+a)*(1-d) + b*c)/(2*b)
A(scalar i, scalar j)	returns A(Sii, Sji, Sij, Sjj)
B(complex **a**, complex **b**, complex **c**, complex **d**)	F_{12} conversion = Z0*((1+a)*(1+d) - b*c)/(2*b)
B(scalar i, scalar j)	returns B(Sii, Sji, Sij, Sjj)
C(complex **a**, complex **b**, complex **c**, complex **d**)	F_{21} conversion = (1/Z0)*((1-a)*(1-d) - b*c)/(2*b)
C(scalar i, scalar j)	returns C(Sii, Sji, Sij, Sjj)
D(complex **a**, complex **b**, complex **c**, complex **d**)	F_{22} conversion = ((1-a)*(1+d) + b*c)/(2*b)
D(scalar i, scalar j)	returns D(Sii, Sji, Sij, Sjj)
H11(complex **a**, complex **b**, complex **c**, complex **d**)	H_{11} conversion = Z0*((1+a)*(1+d) - b*c)/((1-a)*(1+d) + b*c)
H11(scalar i, scalar j)	returns H11(Sii, Sji, Sij, Sjj)
H12(complex **a**, complex **b**, complex **c**, complex **d**)	H_{12} conversion = 2*c/((1-a)*(1+d) + b*c)
H12(scalar i, scalar j)	returns H12(Sii, Sji, Sij, Sjj)
H21(complex **a**, complex **b**, complex **c**, complex **d**)	H_{21} conversion = -2*b/((1-a)*(1+d) + b*c)
H21(scalar i, scalar j)	returns H21(Sii, Sji, Sij, Sjj)

H22(complex **a**, complex **b**, complex **c**, complex **d**)	H_{22} conversion = (1/Z0)*((1-a)*(1-d) - b*c)/((1-a)*(1+d) + b*c)
H22(scalar i, scalar j)	returns H22(Sii, Sji, Sij, Sjj)
kfac(complex **a**, complex **b**, complex **c**, complex **d**)	k-factor = (1 - abs(a)^2 - abs(d)^2 + (abs(a*d - b*c)^2)/(2*abs(b*c))
kfac(scalar i, scalar j)	returns kfac(Sii, Sji, Sij, Sjj)
MAPG(complex **a**, complex **b**, complex **c**, complex **d**)	maximum available power gain = abs(b/c)*(kfac(a,b,c,d) - sqrt(kfac(a,b,c,d)^2 - 1))
MAPG(scalar i, scalar j)	returns MAPG(Sii, Sji, Sij, Sjj)
MSG(complex **a**, complex **b**, complex **c**, complex **d**)	maximum stable power gain = abs(b)/abs(c)
MSG(scalar i, scalar j)	returns MSG(Sii, Sji, Sij, Sjj)
mu1(complex **a**, complex **b**, complex **c**, complex **d**)	μ-factor = (1 - abs(a)^2) / (abs(d - conj(a) * (a*d-b*c)) + abs(b*c))
mu1(scalar i, scalar j)	returns mu1(Sii, Sji, Sij, Sjj)
mu2(complex **a**, complex **b**, complex **c**, complex **d**)	μ-factor = (1 - abs(d)^2) / (abs(a - conj(d) * (a*d-b*c)) + abs(b*c))
mu2(scalar i, scalar j)	returns mu2(Sii, Sji, Sij, Sjj)
T11(complex **a**, complex **b**, complex **c**, complex **d**)	T_{11} conversion = -(a*d - b*c)/b
T11(scalar i, scalar j)	returns T11(Sii, Sji, Sij, Sjj)
T12(complex **a**, complex **b**, complex **c**, complex **d**)	T_{12} conversion = a/b
T12(scalar i, scalar j)	returns T12(Sii, Sji, Sij, Sjj)
T21(complex **a**, complex **b**, complex **c**, complex **d**)	T_{21} conversion = -d/b
T21(scalar i, scalar j)	returns T21(Sii, Sji, Sij, Sjj)
T22(complex **a**, complex **b**, complex **c**, complex **d**)	T_{22} conversion = 1/b

T22(scalar i, scalar j)	returns T22(Sii, Sji, Sij, Sjj)
Y11(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{11} conversion = (1/Z0)*((1-a)*(1+d) + b*c)/((1+a)*(1+d) - b*c)
Y11(scalar i, scalar j)	returns Y11(Sii, Sji, Sij, Sjj)
Y12(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{12} conversion = (1/Z0)*(-2*c)/((1+a)*(1+d) - b*c)
Y12(scalar i, scalar j)	returns Y12(Sii, Sji, Sij, Sjj)
Y21(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{21} conversion = (1/Z0)*(-2*b)/((1+a)*(1+d) - b*c)
Y21(scalar i, scalar j)	returns Y21(Sii, Sji, Sij, Sjj)
Y22(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{22} conversion = (1/Z0)*((1+a)*(1-d) + b*c)/((1+a)*(1+d) - b*c)
Y22(scalar i, scalar j)	returns Y22(Sii, Sji, Sij, Sjj)
Z11(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{11} conversion = Z0*((1+a)*(1-d) + b*c)/((1-a)*(1-d) - b*c)
Z11(scalar i, scalar j)	returns Z11(Sii, Sji, Sij, Sjj)
Z12(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{12} conversion = Z0*(2*c)/((1-a)*(1-d) - b*c)
Z12(scalar i, scalar j)	returns Z12(Sii, Sji, Sij, Sjj)
Z21(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{21} conversion = Z0*(2*b)/((1-a)*(1-d) - b*c)
Z21(scalar i, scalar j)	returns Z21(Sii, Sji, Sij, Sjj)
Z22(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{22} conversion = Z0*((1-a)*(1+d) + b*c)/((1-a)*(1-d) - b*c)
Z22(scalar i, scalar j)	returns Z22(Sii, Sji, Sij, Sjj)

NOTE For both mu1 and mu2, conj is the complex conjugate. For scalars a and b, conj(a+ib)=(a-ib).

Channel Parameter Data	
data	corrected data

data(scalar i)	corrected data of trace i (trace number)
mem	memory data
mem(scalar i)	memory data of trace i (trace number)
xAxis	x-axis data
xAxis(scalar i)	x-axis data of trace i (trace number)
S11 - S44	S-parameter data

NOTE When a trace number is out of range, the ENA indicates an error message, "Equation runtime error" and equation is not reflected on the trace. The maximum number of traces can be set by **System** > **Misc Setup** > **Channel/Trace Setup.**

Converting S-Parameters to H, Y, Z, F & T-Parameters

The following section provides definition of the two port parameters in Equation Editor of the ENA.

1. **Hybrid parameters (H)**

$$\begin{pmatrix} V_1 \\ I_2 \end{pmatrix} = (H)\begin{pmatrix} I_1 \\ V_2 \end{pmatrix} \qquad (H) = \begin{pmatrix} H_{11} & H_{12} \\ H_{21} & H_{22} \end{pmatrix}$$

2. **Admittance parameters (Y)**

$$\begin{pmatrix} I_1 \\ I_2 \end{pmatrix} = (Y)\begin{pmatrix} V_1 \\ V_2 \end{pmatrix} \qquad (Y) = \begin{pmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{pmatrix}$$

3. **Impedance parameters (Z)**

$$\begin{pmatrix} V_1 \\ V_2 \end{pmatrix} = (Z)\begin{pmatrix} I_1 \\ I_2 \end{pmatrix} \qquad (Z) = \begin{pmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{pmatrix}$$

Fundamental parameters (F)

$$\begin{pmatrix} V_1 \\ I_1 \end{pmatrix} = (F) \begin{pmatrix} V_2 \\ -I_2 \end{pmatrix} \qquad (F) = \begin{pmatrix} A & B \\ C & D \end{pmatrix}$$

4.

$$\begin{pmatrix} b_1 \\ b_2 \end{pmatrix} = (S) \begin{pmatrix} a_1 \\ a_2 \end{pmatrix} \qquad (S) = \begin{pmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{pmatrix}$$

$$a_1 = \frac{V_1 + I_1 Z_0}{2 \cdot \sqrt{Z_0}} \qquad a_2 = \frac{V_2 + I_2 Z_0}{2 \cdot \sqrt{Z_0}}$$

$$b_1 = \frac{V_1 - I_1 Z_0}{2 \cdot \sqrt{Z_0}} \qquad b_2 = \frac{V_2 - I_2 Z_0}{2 \cdot \sqrt{Z_0}}$$

5. Scattering transfer parameters (T)

$$\begin{pmatrix} b_1 \\ a_1 \end{pmatrix} = (T) \begin{pmatrix} a_2 \\ b_2 \end{pmatrix} \qquad (T) = \begin{pmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{pmatrix}$$

S-parameters of 2-port network can be converted to Hybrid parameters (H), Admittance parameters (Y), Impedance parameters (Z), Fundamental parameters (F) and Scattering transfer parameters (T) using the following functions:

1. **Converting S-parameters to H-parameters**

$$H_{11} = Z_0 \cdot \frac{(1+S_{11})(1+S_{22}) - S_{12}S_{21}}{(1-S_{11})(1+S_{22}) + S_{12}S_{21}}$$

$$H_{12} = \frac{2 \cdot S_{12}}{(1-S_{11})(1+S_{22}) + S_{12}S_{21}}$$

$$H_{21} = \frac{-2 \cdot S_{21}}{(1-S_{11})(1+S_{22}) + S_{12}S_{21}}$$

$$H_{22} = \frac{1}{Z_0} \cdot \frac{(1-S_{11})(1-S_{22}) - S_{12}S_{21}}{(1-S_{11})(1+S_{22}) + S_{12}S_{21}}$$

2. Converting S-parameters to Y-parameters

$$Y_{11} = \frac{1}{Z_0} \cdot \frac{(1-S_{11})(1+S_{22}) + S_{12}S_{21}}{(1+S_{11})(1+S_{22}) - S_{12}S_{21}}$$

$$Y_{12} = \frac{1}{Z_0} \cdot \frac{-2 \cdot S_{12}}{(1+S_{11})(1+S_{22}) - S_{12}S_{21}}$$

$$Y_{21} = \frac{1}{Z_0} \cdot \frac{-2 \cdot S_{21}}{(1+S_{11})(1+S_{22}) - S_{12}S_{21}}$$

$$Y_{22} = \frac{1}{Z_0} \cdot \frac{(1+S_{11})(1-S_{22}) + S_{12}S_{21}}{(1+S_{11})(1+S_{22}) - S_{12}S_{21}}$$

3. Converting S-parameters to Z-parameters

$$Z_{11} = Z_0 \cdot \frac{(1+S_{11})(1-S_{22})+S_{12}S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$$

$$Z_{12} = Z_0 \cdot \frac{2 \cdot S_{12}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$$

$$Z_{21} = Z_0 \cdot \frac{2 \cdot S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$$

$$Z_{22} = Z_0 \cdot \frac{(1-S_{11})(1+S_{22})+S_{12}S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$$

4. Converting S-parameters to F-parameters

$$A = \frac{(1+S_{11})(1-S_{22})+S_{12}S_{21}}{2 \cdot S_{21}}$$

$$B = Z_0 \cdot \frac{(1+S_{11})(1+S_{22})-S_{12}S_{21}}{2 \cdot S_{21}}$$

$$C = \frac{1}{Z_0} \cdot \frac{(1-S_{11})(1-S_{22})-S_{12}S_{21}}{2 \cdot S_{21}}$$

$$D = \frac{(1-S_{11})(1+S_{22})+S_{12}S_{21}}{2 \cdot S_{21}}$$

5. Converting S-parameters to T-parameters

$$T_{11} = \frac{-D_S}{S_{21}}$$

$$T_{12} = \frac{S_{11}}{S_{21}}$$

$$T_{21} = \frac{-S_{22}}{S_{21}}$$

$$T_{22} = \frac{1}{S_{21}}$$

Where:

$\mathbf{D_s = S_{11}S_{22} - S_{12}S_{21}}$

Operators used in Equation Editor

Operator	Description
+	Addition
-	Subtraction
*	Multiplication
/	Division
^	Power
(	Open parenthesis
)	Close parenthesis
,	Comma - separator for arguments
=	Equal (optional)
E	Exponent (as in 23.45E6)

NOTE

Priority of operators is:

1. ^
2. *,/
3. +,-

Data Output

Data Output

- Saving and Recalling Instrument State
- Saving/Recalling Instrument State for Each Channel into/from Memory
- Saving Trace Data to a File
- Saving the Screen Image to a File
- Printing Displayed Screen

Saving and Recalling Instrument State

- Overview
- Compatibility of Files (Saving and Recalling)
- Saving Data
- Recall Procedure
- Recall Procedure Using "Recall by File Name"
- Priority of Recalling Configuration File at Startup

Other topics about Data Output

Overview

You can save the instrument state of the E5071C into a file on mass storage and then recall it later to reproduce that state. You can select the stored data from the following four types.

Type	Stored data and usage
State only (State Only)	Saves the setting of the E5071C and reproduces the state when it was saved by recalling it later into the E5071C.
State and calibration data (State & Cal)	Saves the setting of the E5071C and calibration data (calibration coefficient array) to reproduce the state when it was saved by recalling it later into the E5071C. At this time, you can perform error correction of measured values by using the recalled calibration data.
State and trace (State & Trace)	Saves the setting of the E5071C and traces (error-corrected data array and error-corrected memory array) to reproduce the state when it was saved by recalling it later into the E5071C. At this time, the traces are also recalled and displayed on the screen.
State, calibration data, and traces (All)	Saves the setting of the E5071C, calibration data, and traces to reproduce the state when it was saved by recalling it later into the E5071C. At this time, the calibration data and traces are also recalled.

In addition, the "user-preset" function is provided to allow the user to freely set up an instrument state recalled when the preset function is executed.

Compatibility of Files (Saving and Recalling)

Files saved with the E5071C cannot be recalled with the E507xA/B. The other compatibility of saving/recalling the instrument state file is as follows:

Compatibility between different frequency models

			Recalling by E5071C									
	Model	Opt. No.	230430	235435	240440	245445	260460	265465	280480	285485	2D54D5	2K54K5
Saved File by	E5071C	230430	Y	Y *	Y	Y *	Y	Y *	Y	Y *	Y #	Y #
		235435	Y	Y	Y	Y	Y	Y	Y	Y	Y #	Y #
		240440	Y #	Y #	Y	Y *	Y	Y *	Y	Y *	Y #	Y #
		245445	Y #	Y #	Y	Y	Y	Y	Y	Y	Y #	Y #
		260460	Y #	Y #	Y #	Y #	Y	Y *	Y	Y *	Y #	Y #
		265465	Y #	Y #	Y #	Y #	Y	Y	Y	Y	Y #	Y #

		280 480	Y#	Y#	Y#	Y#	Y#	Y*	Y	Y*	Y#	Y#
		285 485	Y#	Y#	Y#	Y#	Y#	Y#	Y	Y	Y#	Y#
		2D5 4D5	Y#	Y#	Y#	Y#	Y#	Y#	Y#	Y#	Y	Y
		2K5 4K5	Y#	Y#	Y#	Y#	Y#	Y#	Y#	Y#	Y#	Y
	E5070A/B		Y	Y	Y	Y	Y	Y	Y	Y	N	N
	E5071A/B		N	N	N	N	N	N	Y	Y	N	N

- Y: Recall is possible
- N: Recall is impossible
- Y*: Recall is only possible for the revision A.11.0x and above
- Y#:Recall is only possible for the revision A.11.20 and above. However, warning message prompts "*Incompatible recall file*". Some state data may be lost.

NOTE When state files with calibration data that are saved with E5070A/B and E5071A/B, only states and trace data are recalled.

NOTE Recalling state file saved in models without Bias Tee onto models with Bias Tee:

1. When the Start/Stop frequency is set to below 100 kHz at the time of save, it should be changed to 100 kHz at recall. In such cases, Center/Span frequency should also be changed.
2. Similarly, calibration factor can also be recalled. However, if some calibration points are below 100 kHz at the time of save, C? or C! is displayed after recall.

NOTE As a general rule, its not advisable to use the calibration factor of one unit onto the other.

Compatibility between different number of ports models

- If the stored data is "state only (**State Only**)," files saved with a model having a smaller number of ports can be recalled with a model having a larger number of ports, but the opposite is not possible.

		Recalling	
	Option: number of port	**2 Ports**	**4 Ports**
Saved File by	**2 Ports**	Y	Y/ c
	4 Ports	N	Y

Y: Recall is possible

N: Recall is impossible

Y/ c: Only when the stored data is "state only (**State Only**)," recall is possible.

Compatibility when the maximum number of channels/traces is different

- You cannot recall files saved by specifying all channels/traces (**All**) as the save target.
- If the number of channels/traces at save does not exceed that at recall, you can recall files saved by specifying the displayed channel/trace (**Disp Only**) as the save target.

Compatibility when the Firmware revision is different

- Files that are saved with later firmware revisions cannot be recalled by prior firmware revisions.
- Recalling the state file which is saved by prior firmware revision may lose some settings.
- Recalling the state file which is saved by prior firmware revision may cause a sweep time change and C? is displayed. In this case, re-calibration is recommended.

Compatibility when the system spec. version (available with ":SERV:SREV?") is different

- When Files that are saved with different system spec. versions include calibration data, only states and trace data are recalled.
  - If you recall an incompatible file, an error occurs and the device recovers to the presetting.

Saving Data

Selecting Content to be Saved

NOTE This setting takes effect both when saving the entire instrument state into a file and when saving the instrument state for each channel into memory.

1. Press **Save/Recall** key.
2. Click **Save Type**.
3. Click the softkey corresponding to the content of the instrument state you want to save.

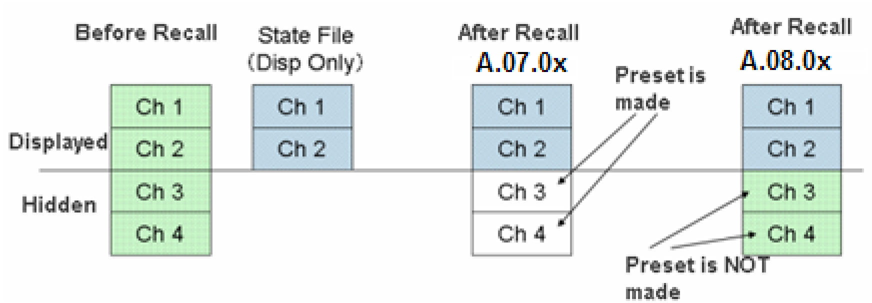
Selecting Save Target Channel/Trace

1. Press **Save/Recall** key.
2. Click **Channel/Trace** and select the save target from all channels/traces (**All**) or displayed channel/traces only (**Disp Only**).

If you specify the displayed channel/traces only as the save target, you can reduce the file size. However, for channels/traces that are not displayed, you cannot recall and reproduce the instrument state separately held for each channel/trace at a later time.

Faster Recall

From Firmware revision A.08.0x or later, the recall speed has been increased in the **DISP only** mode. In Firmware revision below A.08.0x, preset was done for hidden channels. In Firmware revision A.08.0x or later, hidden channels are not made preset and results in a faster recall.



e5071c281

Specifying the Trigger Source Setting After Recall (Auto Trigger Source)

Once auto trigger source is set to OFF and the state file is saved with State & Trace or All, trigger source remains to the setting determined by user instead of changing to manual. When auto trigger source is set to ON, trigger source will change to manual automatically at the save. This allows you to hold the trace after recall because the trigger is not generated internally. By default, auto trigger source is set to ON. This function is available in firmware revision A.11.20 and later.

To turn ON/OFF the auto trigger source:

1. Press **Save/Recall** key.
2. Click **Auto Trig Source** > **ON|OFF**.

Saving Instrument State

Follow the procedure below to save internal data from the E5071C.

1. Press **Save/Recall** key.
2. Click **Save State**.
3. Click the softkey corresponding to the destination you want to save.

Softkey	Description
State01 to **State08**	Save the instrument state into the state number.
Autorec	Save the instrument state as the auto recall setting. The E5071C is automatically configured with this state at the startup. This key saves the state into the "**D:Autorec.sta**". When **Autorec.sta** file is found on the D: drive at startup, the E5071C is automatically configured using the saved settings. To disable the auto recall function, delete the **Autorec.sta** files.
User Pres	Save the instrument state as user preset. The user can preset the analyzer at user saved status.
File Dialog...	Save the instrument state as your desired file name. You can enter a file name using the **Input from the front panel** buttons on the dialog box when storing a file.

NOTE If **D:\Autorec.sta** is found on the system at startup, the E5071C is automatically configured using the saved settings. When the external floppy disk drive is connected as A: drive, then if **A:\Autorec.sta** is found at startup, the E5071C is also automatically configured using the saved settings. If both files are found, **A:\Autorec.sta** is recalled. To disable the auto recall function, delete the **Autorec.sta** files.

NOTE An asterisk (*) in the upper right of the softkey indicates that the corresponding file of the softkey already exists. If you save into the existing file, the existing file is copied as **backup.sta** and then overwritten.

Recall Procedure

Follow the procedure below to recall internal data from the E5071C.

1. If you recall a file that includes traces (its content was set to **State &Trace** or **All** when it was saved), the trigger source is automatically set to Manual.

1. Press **Save/Recall** key.
2. Click **Recall State**.

When you want to recall State01.sta - State08.sta, Autorec.sta

1. Press **State01** - **State08** or **Autorec**.

When you want to recall other files

1. Press **File Dialog...** to open the Open dialog box.
2. Select the folder and the file using the external keyboard and mouse.
3. Click **Open**.

The following warning messages may appear when recall fails:

Warning Message	Note
Model Mismatch	This message is displayed when the state file of E5070B/E5071B is recalled by E5071C options 2D5, 4D5, 2K5 and 4K5 .
Unsupported Version	This message is displayed when the state file generated from a newer version of Firmware is recalled.
Invalid No. of Ports	This message is displayed when the state file generated from a Firmware having different Port Options is recalled.
Source Attenuator Option Mismatch	This message is displayed when the state file generated from a Firmware having different Source Attenuator Option is recalled.
Power Sweep Option Mismatch	This message is displayed when the state file generated from a Firmware having different Power Sweep Option is recalled.
Invalid No. of Points	This message is displayed when the Maximum No. of Points are not enough.
External Test Set Option Mismatch	This message is displayed when the state file generated from a Firmware having different External Test Set Option is recalled.
Frequency Option Mismatch	This message is displayed when the state file generated from a Firmware having different Frequency Option is recalled.
ch x, tr y, z pts required	This message is displayed when the Recall Function cannot recall data with maximum channel or traces. The number of channels/traces should be higher than the recall state file to recover from the error. More information is available in Setting upper limit of channels and traces.

NOTE The warning message listed above:
1. May not appear in all failures
2. Appears on the screen only when recall is done from soft key by Recall State or Recall by File Name.
3. Displays "Recall Failed" once when the recall is done through GPIB or VBA.
4. Is not kept in Error queue. This means that when SCPI.SYSTem.ERRor is executed, "Recall Failed" is returned.
5. These messages are supported for state files created in Firmware revision A.09.10 or later.

NOTE When a user file is used in Extending the Calibration Plane Using Network De-embedding, Determining Characteristics After Adding a Matching Circuit, or Determining the Characteristics that Result from Adding a Matching Circuit to a Differential Port and the setup status is saved, a recall error will occur if the user file is not located in the same folder as when the state was saved.

NOTE Pressing **Save/Recall** > **Explorer** executes Windows Explorer. This helps you to browse the files in the ENA hard disk drive.

Recall Procedure Using "Recall by File Name" Feature

You can use the recall feature with the **Recall by File Name** softkey for files you have named freely and saved in the D:\State folder. This function lets you recall a file you have named freely and saved by simple softkey operation, eliminating annoying operation using the Open dialog box.

NOTE Although there is no limit to the number of files saved in a folder, only up to 50 files are displayed on the softkeys. If more than 50 files are saved in a folder, they are sorted in the order of numbers 0 to 9 and alphabetic characters A to Z and the first 50 files are displayed as softkeys.

Although there is no limit to the number of characters of a file name, only up to 12 characters are displayed on the softkey. If a file name exceeds 12 characters, the first 12 characters are displayed on the softkey and the remaining characters are omitted and replaced with "...".

NOTE Different files may be displayed on softkeys with the same name or a saved file is not displayed on any softkey because of the above limitations.

1. Press **Save/Recall** key

2. Click **Recall by File Name**.
3. Files that have been named and saved in the D:\State folder are displayed on softkeys. Press the key for the file you want to recall.

Priority of Recalling Configuration File at Startup

If several instrument configuration files exist at the startup of the E5071C, only one file is recalled and set at a time in the following order of priority.

If these files do not exist, the normal preset (factory preset) is executed.

Priority	Recalled file
1	Configuration file for the auto-recall function in the A drive (If external floppy disk drive is connected.)
2	Configuration file for the auto-recall function in the D drive
3	Configuration file for the user-preset function in the D drive. Executed when the preset operation mode is User and the file (**D:\UserPreset.sta**) exists.

Saving/Recalling Instrument State for Each Channel into/from Memory

- Overview
- Saving Instrument State for Each Channel
- Recalling Instrument State for Each Channel
- Deleting Saved Instrument State (Clearing all Registers)

Other topics about Data Output

Overview

The E5071C allows you to save/recall the instrument state for each channel independently. This function allows you to save the instrument state of the active channel independently into one of four registers (A to D, volatile memory) and to recall the instrument state from the register to restore it as the state of the currently active channel. As in the case of saving the entire state of the instrument into a file, you can select items to be saved from four kinds.

Since you can recall the instrument state for each channel that was saved with this function from a different channel that the one used to save it, this function is very useful for copying an instrument state between channels.

NOTE Unlike when saving the entire instrument state, the instrument state for each channel is saved into volatile memory instead of a file, so if you turn off the power, this state is lost.

Saving Instrument State for Each Channel

1. Press **Channel Next**/**Channel Prev** keys to activate a channel whose state you want to save.
2. Press **Save/Recall** key.
3. Click **Save Channel**.
4. Click one of **State A** to **State D** to save the instrument state of the active channel to the specified register.

NOTE For registers having saved data, the * symbol is displayed to the right of their softkey label. If you specify one of these, its content is overwritten.

Recalling Instrument State for Each Channel

1. Press **Channel Next**/**Channel Prev** keys to activate a channel whose state you want to recall and restore.
2. Press **Save/Recall** key.

3. Click **Recall Channel**.

4. Click the softkey of the register in which the state you want to restore is saved. This instrument state is recalled to the active channel.

Deleting Saved Instrument State (Clearing all Registers)

1. Press **Save/Recall** key.

2. Click **Save Channel**.

3. Click **Clear States**. The contents of all the registers are deleted.

Saving Trace Data to a File

- Saving Data in CSV Format
- Saving Data in Touchstone Format

Other topics about Data Output

Saving Data in CSV Format

The E5071C allows the user to save data for the active trace on the active channel to a CSV file (file extension *.csv) and to load the data into PC application software for further processing.

Trace data are saved in the format shown below.

Example of saved trace data

"# Channel 1"

"# Trace 1"

Frequency, Formatted Data, Formatted Data

+3.00000000000E+005, +1.41837599227E-002, +1.43446459328E-006

+4.27985000000E+007, +1.41275293412E-002, +2.02407834551E-004

+8.52970000000E+007, +1.41334093048E-002, +4.00643331604E-004

+1.27795500000E+008, +1.41240661092E-002, +6.09250514670E-004

+1.70294000000E+008, +1.41402155348E-002, +8.05620003993E-004

The first line shows the number of the active channel at the time the data was saved.

The second line shows the number of the active trace at the time the data was saved.

The third line is a header line indicating the contents of each item of trace data written on the fourth line onward.

The fourth line onward shows the trace data. The amount of data is determined by the number of points (frequency) assigned to the trace.

Saving Trace Data

Follow the procedure below to save trace data from the E5071C.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace to be saved.
2. Press **Save/Recall** key.
3. Click **Save Trace Data** to open the Save As dialog box.
4. Select the destination folder and input a file name.
5. Click **Save** to save the file.

Saving Data in Touchstone Format

You can also save trace data of a E5071C active channel to a Touchstone format file, based on 1- to 4-port models.

Touchstone file data format

You can save data in "log magnitude - angle", "linear magnitude - angle", or "real number - imaginary number."

When AUTO is selected, the data format is automatically set according to the display format of the active trace. However, when the display format of the active trace is set to one other than log magnitude format (LogMag), linear magnitude format (LinMag), or real-imaginary number format (Real/Imag), the data format is automatically set to real-imaginary number.

You can use data saved in Touchstone format for a circuit simulator such as Agilent Advanced Design System (ADS) on your PC (personal computer) or workstation. For more information on the ADS, refer to the operation manual that comes with the system.

NOTE The fixture simulator function enables you to recall to the E5071C the s2p (2-port Touchstone file) file type used to save user-defined matching circuit information. However, no other type of file saved in Touchstone format can be recalled to the E5071C.

NOTE The port reference impedance value when outputs the touchstone format file has to be the same impedance value (real) to every measurement port.

NOTE You can output the measurement result when using complex port impedance conversion, or when setting different port impedance at each port. However, the touchstone file is valid only when the same impedance value (real) to every measurement port is used.
When you recall the file saved with above condition, the file is recalled as single normalized reference Z0 defined at option line, and it provides in-consistency result.

File types of Touchstone files

File types of the E5071C Touchstone files are s1p, s2p, s3p, and s4p. The file type indicates the number of ports of the data structure that is output to the Touchstone file.

Data structure in Touchstone file

Data structure of the Touchstone file consists of a header part and a data part. The contents of the file is text data, which is ready to be read with a general text editor.

The header part consists of the comment line and option line.

- Comment line is preceded by an exclamation mark (!), and includes returned value of *IDN?, file created date, calibration state, list of all S parameters of a specified port, and port impedance setting when saving the data.
- Option line includes format information.

The header parts of s1p and s4p are shown below.

Header of s1p

```
!Agilent Technologies,E5071B,<ID>,<FW Revision>
!Date <Date>
!Data & Calibration Information
!Freq Sww:Method(Stat)
!PortZ PORTw:R+jX
!Above PortZ is port Z conversion or system Z0 setting when saving the data.
!When reading, reference impedance value at option line is always used"
# Hz S FMT R Z0
```

Header of s4p

```
!Agilent Technologies,E5071B,<ID>,<FW Revision>
!Date <Date>
!Data & Calibration Information
!Freq Sww:Method(Stat) Sxw:Method(Stat) Syw:Method(Stat) Szw:Method(Stat)
.
.
!PortZ PORTw:R+jX     Portx:R+jX     Porty:R+jX     Portz:R+jX
!Above PortZ is port Z conversion or system Z0 setting when saving the data.
!When reading, reference impedance value at option line is always used"
Swz:Method(Stat) Sxz:Method(Stat) Syz:Method(Stat) Szz:Method(Stat)
# Hz S FMT R Z0
```

Comment line

Touchstone data files include comments. Comments are preceded by an exclamation mark (!).

Each item has the following meaning:

Parameter	Description
Sww to Szz	S parameters of the selected test port; corresponds in ascending order, beginning with w to z.
Method	Calibration type applied to S parameter.
Stat	State of S parameter calibration and error correction setting (ON, OFF, or --) ON = Error correction is set to ON OFF =Error correction is set to OFF

	-- = Calibration is not performed
PORTw to PORTz:	Single-end port impedance setting when saving the data. This is not valid when reading the file. When reading, the reference impedance value Z0 at option line is always used.

Option Line

The option line is formatted as follows.

Hz S FMT R Z0

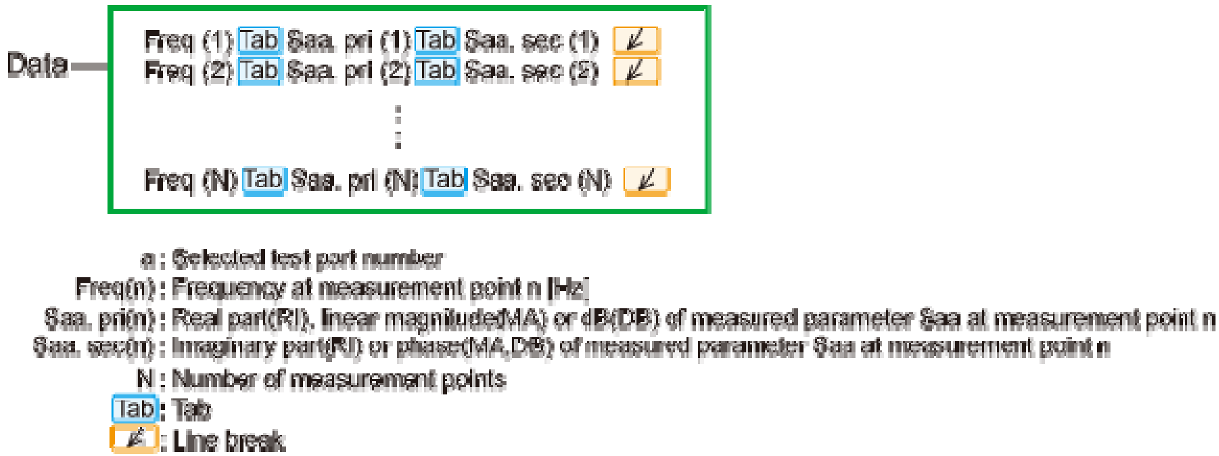
Parameter	Description
FMT	Data format RI = Real number - imaginary number MA = Linear magnitude - angle DB = Log magnitude - angle
Z0	Reference impedance value

The structure of the data part depends on the combination of the selected file type and specified port.

NOTE When the S parameter of a port on which calibration is not performed is specified, if data measured with the S parameter exists, that data are output to a Touchstone file. If no measurement data exists, 0 (for log magnitude - angle, log magnitude = -200 dB) is output in the corresponding field.

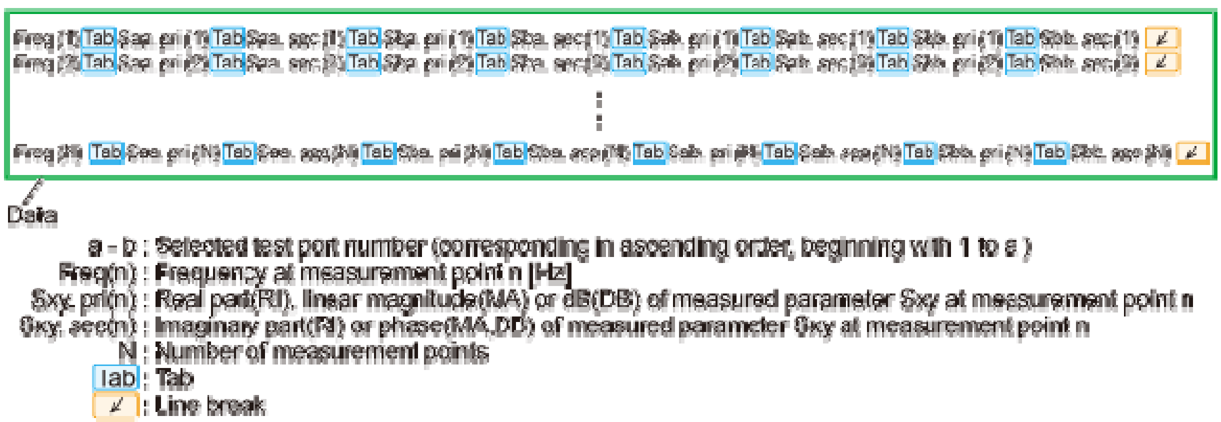
The following figures show the data structures of files saved in Touchstone format.

1-port Touchstone file



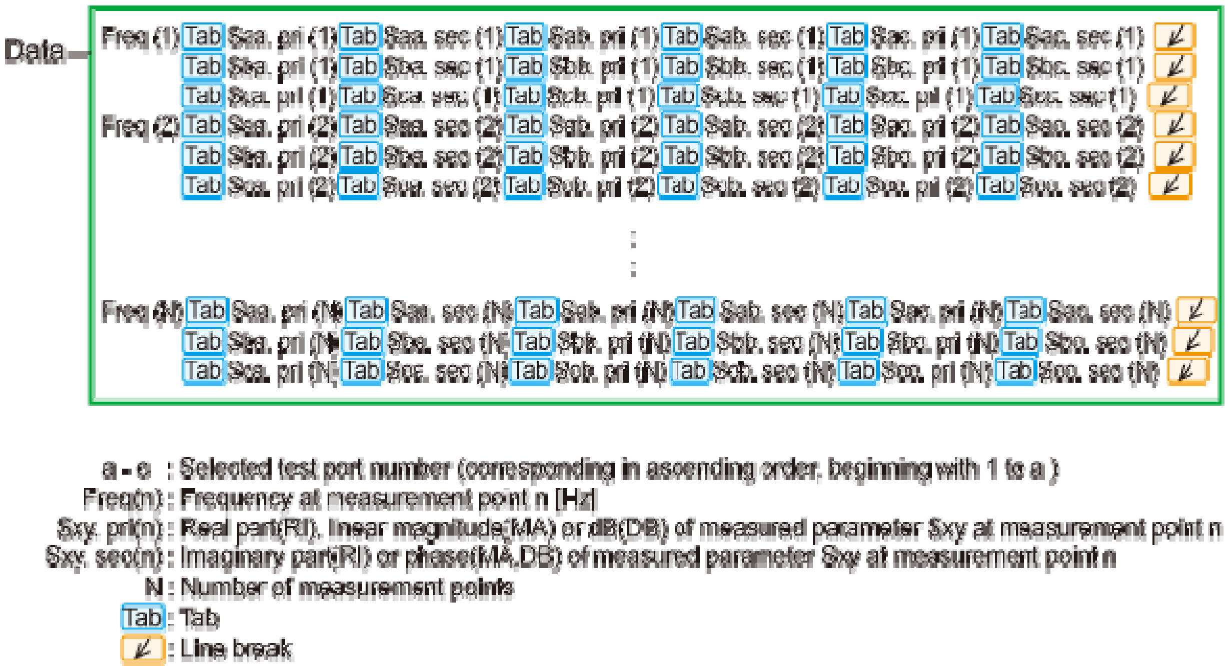
e5071c346

2-port Touchstone file



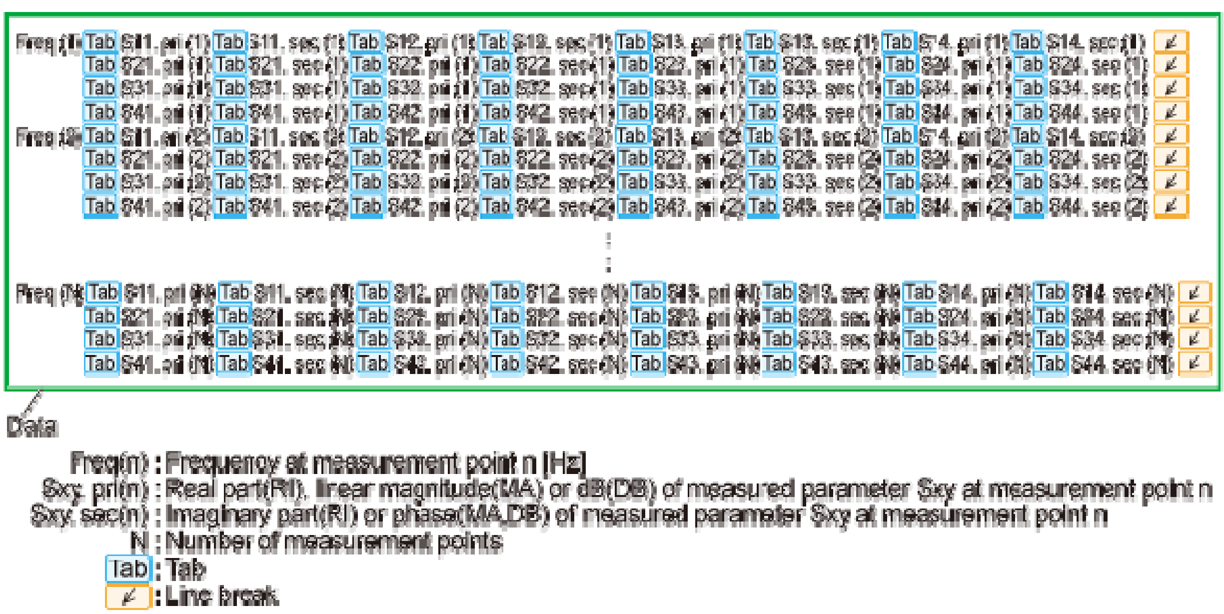
e5071c347

3-port Touchstone file



e5071c348

4-port Touchstone file



e5071c349

Restrictions when saving data in Touchstone format

The following restrictions apply when saving measurement data into Touchstone format.

- When both fixture simulation and port impedance conversion are on, all Z0 of the ports to be saved must be set to the same value. If Z0 is different among the ports, no error occurs, but only the Z0 of the smallest port number is output to the header.
- When the time domain function is on, the saved data are not the displayed data but the data of the S parameter before conversion.
- For data saved in touchstone format, data operation, time domain, parameter conversion, data format, electrical delay, equation editor, and smoothing are not reflected in the output data.
- An error occurs when attempting to save data that use the frequency offset function.

Saving procedure

Follow the steps below to save trace data in Touchstone format.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to select the trace to be saved.
2. Press **Save/Recall** key.
3. Click **SnP** > **SnP Format**.
4. Click the softkey that corresponds to the data format you want to save.

Softkey	Function
Auto	Data format is automatically set according to the display format of the active trace. When the display format of the active trace is set to one other than log magnitude format (LogMag), linear magnitude format (LinMag), or real-imaginary number format (Real/Imag), the data format is automatically set to real-imaginary number.
LogMag/Angle	Select "log magnitude - angle" data format
LinMag/Angle	Select "linear magnitude - angle" data format
Real/Imaginary	Select "real - imaginary number" data format

5. Click the softkey for **s1p**, **s2p**, **s3p**, or **s4p** according to the file type you want to save.
6. Click the softkey that corresponds to the combination of ports that you want to save. For example, when pressing **s3p** in Step5, softkeys displayed are **1-2-3**, **1-2-4**, **1-3-4**, and **2-3-4**. If you want to save the data of the channel for which port 1, port 2, and port 4 are measured, press **1-2-4**.

7. SaveAs dialog box opens. For its operations, use an external keyboard and mouse.
8. Specify the folder to which the file should be saved, enter a file name, and then press **Save** to save the file.

   NOTE When saving data in a sweep process, the data during sweep is saved into a Touchstone file. That is, the previous sweep data is saved as data that has not been swept; or, if sweep was not performed previously, zero data might be saved. Therefore, you should set the active channel to the HOLD state when saving data into a Touchstone file.

Saving the Screen Image to a File

- Overview
- Saving Screen Image as File

Other topics about Data Output

Overview

Along with printing, the E5071C allows the user to save screen images as bitmap (.bmp) or portable network graphics (.png) files. Saved files can be loaded into PC application software for further processing.

Saving Screen Image as File

Follow the procedure below to save a screen image to a file.

1. Display the screen to be saved as a file. If you want to save the screen with a white background, set the display mode to inverted display before saving the screen. For details about display mode, see Setting Display Colors.
2. Press **System** key. The screen image at the time **System** key is pressed is the image that will be saved.
3. Press **Dump Screen Image** to open the Save As dialog box. "Bitmap Files (*.bmp)" or "Portable Network Graphics (*.png)" can be selected as the file type.
4. Select the file type.
5. Select the destination folder and type a file name.
6. Press **Save** to save the screen image of E5071C to a file.

Printing Displayed Screen

- Overview
- Printed/Saved Images
- Print Procedure

Other topics about Data Output

Overview

By connecting a printer to the USB port of the E5071C, you can print the displayed screen of the E5071C.

Printed/Saved Images

The display image saved in the volatile memory (clipboard) is printed/saved. If no image is saved in the clipboard, the image displayed at the time of print execution is printed/saved.

Saving image to clipboard

The **System** key also has a screen capture feature. When you press **System** key, the image displayed on the screen immediately before pressing is saved in the clipboard.

NOTE The image in the clipboard is cleared when you execute print/save.

Print Procedure

Preparation before printing

Follow these steps to prepare for printing:

1. Turns off the E5071C.
2. Turn on the printer and connect it to E5071C.
3. Turn on the E5071C.
4. Press **System** key.
5. Press **Printer Setup**. The Printers window opens. The icons of the printers that have been connected are displayed in the window. When you connect a print for the first time, it is automatically registered and its icon is added in the window.
6. The printer with the check mark (✔) on its icon is selected as the default printer for printing. If you want to change it, select (highlight) the icon of your preferred printer in the Printers window and then click **Set as Default Printer** in the File menu.
7. Click **Printing Preferences...** in the File menu. The Printing Preferences dialog box for the selected printer appears. Set items

necessary before printing such as Page Size and then click the **OK** button .

8. Click **Close** in the File menu.

Executing print

Follow these steps to print the screen information:

1. Display the screen you want to print.
2. Press **System** key to save the currently displayed screen onto the clipboard.
3. As necessary, press **Invert Image** to toggle between [OFF] for printing in colors close to the actually displayed screen and [ON] for printing in inverse colors.
4. Click **Print** to start printing.
5. To cancel the printing in progress, press **Abort Printing**.

NOTE If you start printing when the printer is not ready (for example, it is not turned on) by mistake, the Printers Folder dialog box may appear. In this case, click **Cancel** to close the Printers Folder dialog box, prepare your printer, and then start printing again.

Optimizing Measurements

Optimizing Measurements

- Expanding Dynamic Range
- Reducing Trace Noise
- Improving Phase Measurement Accuracy
- Improving Measurement Throughput
- Performing Segment-by-Segment Sweep (segment sweep)
- Setting Shift LO

Expanding Dynamic Range

- Overview
- Lowering Receiver Noise Floor

Other topics about Optimizing Measurement

Overview

The dynamic range is the finite difference between the maximum input power level and the minimum measurement power level (noise floor) of the analyzer. In evaluating a characteristic accompanied by a large change in the amplitude (the pass band and stop band of a filter, for example), it is important to increase the dynamic range.

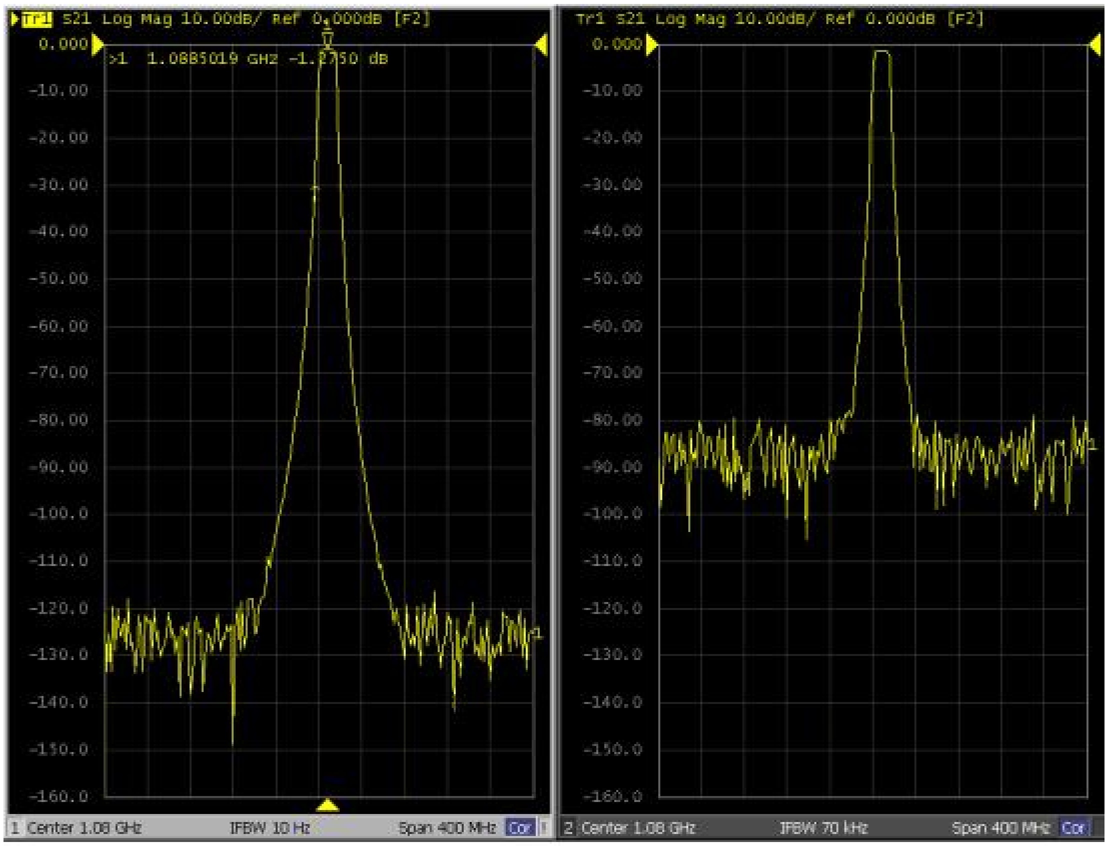
Lowering Receiver Noise Floor

Lowering the noise floor of the receiver enables you to expand the dynamic range. The following methods can be used to lower the receiver noise floor.

- Narrowing the IF bandwidth
- Turning on Sweep Averaging

Narrowing the IF bandwidth

Narrowing the receiver IF bandwidth enables you to reduce the effect of random noise on measurements. Narrowing the IF bandwidth to 1/10 the original bandwidth causes the receiver noise floor to decrease by 10 dB.



e5071c080

To specify the IF bandwidth, follow the steps described below.

1. Press **Channel Next**/**Channel Prev** keys to select a channel on which to specify the IF bandwidth.
2. Press **Avg** key.
3. Click **IF Bandwidth**, then Change the IF bandwidth in the data entry area.

Turning on Sweep Averaging

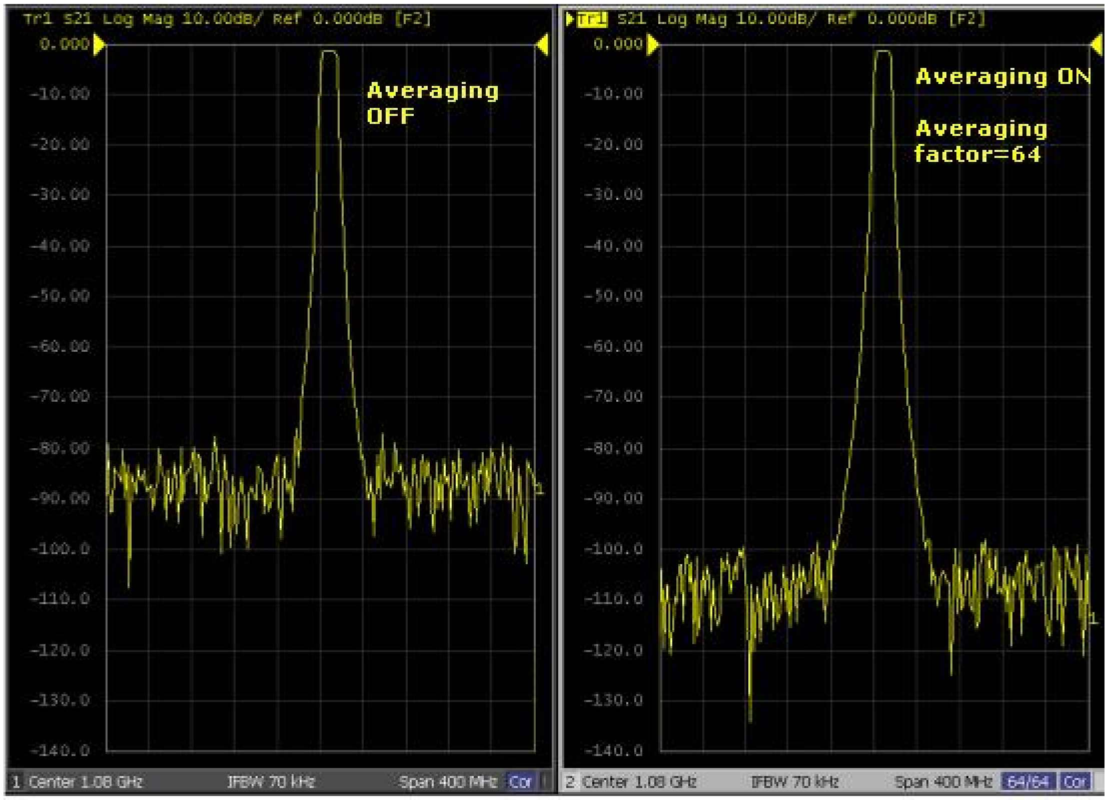
Using sweep averaging also enables you to reduce the effects of random noise on measurements.

Sweep averaging averages data from each point (vector quantity) based on the exponential average of a continuous sweep weighted by the averaging factor specified by the user. Sweep averaging is expressed in following equation.

$$A_n = \frac{S_n}{F} + \left(1 - \frac{1}{F}\right) \times A_{n-1}$$

where:

- A_n = Result of the calculation of sweep averaging for the nth sweep operation at the point in question (a vector quantity)
- S_n = Measurement value obtained at the nth sweep operation at the point in question (a vector quantity)
- **F** = Sweep averaging factor (an integer between 1 and 999)



e5071c081

Define the sweep averaging by following the steps below.

1. Press **Channel Next**/**Channel Prev** keys to select the channel on which you want to define the sweep averaging.
2. Press **Avg** key.
3. Click **Avg Factor**, then change the averaging factor in the data entry area.
4. Click **Averaging** to turn ON the averaging.
5. Clicking **Averaging Restart** resets n to 1 in Sweep Averaging equation in Turning on Sweep Averaging

Reducing Trace Noise

Any of the following methods can be used to lower the trace noise. This section provides the description of Turing on Smoothing.

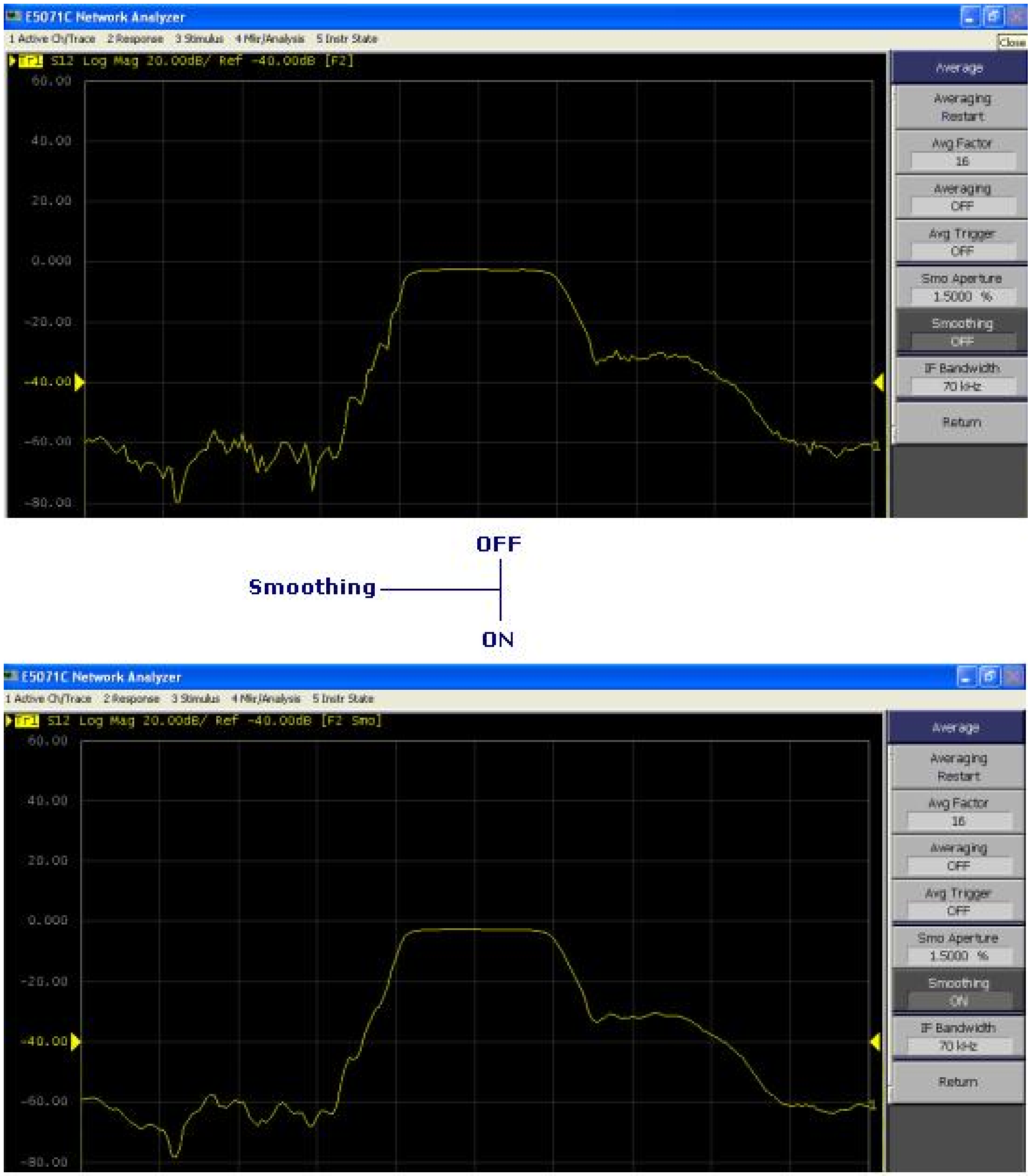
- Turning on Smoothing
- Turning on Sweep Averaging
- Narrowing IF Bandwidth

Other topics about Optimizing Measurements

Turning on Smoothing

Smoothing can be used to reduce noise that has relatively small peaks. By turning on smoothing, the value of each point on a trace is represented by the moving average over the values of several nearby points. The smoothing aperture (percentage of sweep span) defines the range of points to be included in the calculation of the moving average.

NOTE You can define the smoothing trace by trace.



e5071c075

Setting up smoothing

Set up the smoothing operation by following the steps below:

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace on which smoothing will be defined.
2. Press **Avg** key.
3. Click **Smo Aperture**, then change the smoothing aperture (%) in the data entry area.
4. Click **Smoothing** to turn ON smoothing.

Improving Phase Measurement Accuracy

This section describes the following functions that can be used to improve phase measurement accuracy.

- Electrical Delay
- Velocity Factor
- Phase Offset
- Port Extensions and Loss Values

Other topics about Optimizing Measurements

Electrical Delay

Electrical Delay is a function that adds or removes a pseudo-lossless transmission line with a variable length corresponding to the receiver input. Using this function enables you to improve the resolution in phase measurement and thereby measure deviation from the linear phase. You can specify the electrical delay trace by trace. Depending on the media type, the calculation method of the electrical delay, which is required to correct the phase delay, differs.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the phase trace for which you want to specify the electrical delay.
2. Press **Scale** key.
3. Click **Electrical Delay**.
4. Change the electrical delay (in seconds) in the data entry area.
5. Click **Media**, and select a media type for calculating the electrical delay. If the electrical delay is 0 second, the calculation result is always the same regardless of media type.

Softkey	Function
Coaxial	Selects Coaxial as the media type.
Waveguide	Selects Waveguide as the media type.

6. If you have selected **Waveguide** as the media type, click **Cutoff Frequecy**, and specify a cutoff frequency.
   7. The cutoff frequency is available only when the media type is **Waveguide**.

For how to determine the deviation from a linear phase, see Measuring the Deviation from a Linear Phase.

Procedure using marker

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to set the electrical delay.
2. Place the *active marker* in an appropriate position.
3. Press **Maker Fctn**.
4. Click **Marker -> Delay** to set the electrical delay to the group delay value at the position of the active marker (a value smoothed with the aperture of 20% regardless of the smoothing setting).

Phase offset

Phase offset is a function used to add or subtract a predetermined value relative to the frequency to and from the trace. Using this function enables you to simulate the phase offset occurring as a result of, say, adding a cable.

The phase offset can be specified from - 360° to +360° .

Using the Phase Offset Function

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to specify the phase offset.
2. Press **Scale** key.
3. Click **Phase Offset**, then enter the phase offset (°) in the data entry area.

Velocity factor

The velocity factor is the ratio of the propagation velocity of a signal in a coaxial cable to the propagation velocity of that signal in free space. The velocity factor for a common cable is about 0.66. The propagation velocity depends on the dielectric constant (ε_r) of the dielectric substance the cable.

$$\text{Velocity factor} = \frac{1}{\sqrt{\varepsilon_r}}$$

By specifying the velocity factor, you can match the equivalent length (in meters) appearing in the data entry area to the actual physical length when using the Electrical Delay or Setting port extensions to specify the electrical delay (in seconds).

You can define the velocity factor channel by channel.

Using the velocity factor

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to specify the velocity factor.

2. Press **Cal** key.
3. Click **Velocity Factor**, then the velocity factor in the data entry area.

Port Extensions and Loss Values

Setting port extensions

Port Extension is a function for moving the calibration reference plane by specifying the electrical delay. This function is useful, for example, when you cannot directly perform calibration at the DUT terminal because the DUT is inside the test fixture. In such a case, this function enables you to first perform calibration at the test fixture terminal and then move the calibration plane to the DUT terminal by extending the port.

Port extension corrects the electrical delay of each test port (phase shift) only. It cannot remove errors caused by the loss in and incorrect matching of cables, adapters, or test fixtures.

NOTE You can define port extension channel by channel. Setting port extension for one particular channel does not affect other channels.

NOTE Auto Port Extension does not supports waveguide port extension.

Operational procedure

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set port extension.
2. Press **Cal** key.
3. Click **Port Extensions**, and then Select **Extension Port 1/2/3/4**.
4. Specify Loss values by clicking **Loss**.

Setting coaxial delay, waveguide delay and cut off frequency

Follow the steps below to set coaxial or waveguide delay:

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set port extension.
2. Press **Cal** key.
3. Click **Port Extensions.**

Softkey	Function
Auto Port Extension	Sets the Auto Port Extension
Extension Port 1	Sets port extension (in seconds) for test port 1.
Extension Port 2	Sets port extension (in seconds) for test port 2.

Extension Port 3	Sets port extension (in seconds) for test port 3.
Extension Port 4	Sets port extension (in seconds) for test port 4.

4. Click **Extension Port n** where n can be from 1 to 4.
5. Click **Waveguide** to set delay in sec.
6. Click **Cutoff Frequency** to set cut off frequency of selected port.
7. Click **Coax. Extension** to set the coaxial extension in sec.
8. If you want to use waveguide port extension, then setting of coaxial delay is not necessary. If only waveguide measurement is needed, coaxial delay should be set to zero. Coaxial extension value should be set while measuring adapters between coaxial and waveguide.

Waveguide delay and Cut off frequency can be set using the following commands:

SCPI.SENSe(Ch).CORRection.EXTension.PORT(Pt).WAVeguide.TIME

SCPI.SENSe(Ch).CORRection.EXTension.PORT(Pt).WAVeguide.CUToff

NOTE System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.

NOTE Some calibration kits such as the waveguide calibration kit have operational frequency range defined by Minimum and Maximum frequency. When the E5071C stimulus setting is out of the operational frequency range, you cannot click Done key or finish the calibration by remote control. In this case, use a calibration kit that has proper frequency range, or change the E5071C stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5071C with **Cal** > **Modify Cal Kit** > **Define STDs** for the Maximum & Minimum frequency of the calibration kit.

Setting loss values

In addition to port extension, you can set loss values for each port. By correcting loss due to port extension, more accurate measurement results are obtained.

There are two types of loss value settings: loss values at two frequency points for a specified port, and a DC loss value. You can make these settings at the same time for each port.

NOTE You can set loss values channel by channel. Setting loss values for one particular channel does not affect other channels.

1. Setting loss values

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set loss values.
2. Press **Cal** key.
3. Click **Port Extensions.**
4. Click **Extension Port n** where n can be from 1 to 4.
5. Click **Loss** to set a loss value.
6. Click **Loss1 [OFF]** to toggle to **Loss1 [ON]** (enabled), and enter a loss value (**Loss1**) and a frequency (**Freq1**).
7. If you want to set loss at two frequency points, press **Loss2 [OFF]** to toggle to **Loss [ON]** (enabled), and enter a loss value (**Loss2**) and a frequency (**Freq2**).
8. If you want to set loss values for other ports, repeat steps 4 to 6.

- Expression to calculate loss using Loss 1:

$$\mathrm{Loss(f)} = \mathrm{Loss1} \times \sqrt{\left(\frac{\mathrm{f}}{\mathrm{Freq1}}\right)}$$

- Expression to calculate loss using Loss 1 and Loss 2:

$$\mathrm{Loss(f)} = \mathrm{Loss1} \times \left(\frac{\mathrm{f}}{\mathrm{Freq1}}\right)^{n}$$

$$n = \frac{\log_{10}\left|\frac{\mathrm{Loss1}}{\mathrm{Loss2}}\right|}{\log_{10}\frac{\mathrm{Freq1}}{\mathrm{Freq2}}}$$

NOTE When you specify two frequency points, set the lower frequency to Loss1, and the higher one to Loss2.

2. Setting a DC loss value

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set DC loss values.
2. Press **Cal** key.
3. Click **Port Extensions** > **Loss**.

4. Click **Select Port** to select the port for which you want to set a DC loss value.
5. Click **Loss at DC**, then enter a DC loss value.
6. If you want to set a DC loss value for other ports, repeat steps 4 to 5.

Enabling port extensions and loss values

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to enable port extension.
2. Press **Cal** key.
3. Click **Port Extensions**.
4. Turn on **Extensions**.

Using the auto port extension function

The auto port extension function measures port extension and loss values for each port using the OPEN/SHORT standard connected to the port, automatically calculates them, and set them.

Softkey		Function
Auto Port Extension	**Select Ports**	Sets the Ports (1 to 4)
	Measure OPEN	Performs a OPEN measurement
	Measure SHORT	Performs a Short measurement
	Method	Sets the Span
	Include Loss	Sets Loss ON/OFF
	Adjust Mismatch	Sets Adjust Mismatch ON/OFF

When the auto port extension function is completed, the port extensions and loss values are updated to the calculated values.

- You can use both open and short measurement values in the auto port extension function. Note that in this case, the average value of the calculation results is used for updating.
- You can set the auto port extension function channel by channel. Setting the auto port extension function for one particular channel does not affect other channels.
- When the sweep type is power sweep or the frequency offset function is ON, the auto port extension is not available.
- Auto Port Extension does not supports waveguide port extension.

1. Selecting a port(s)

Select the port(s) for which you want to use the auto port extension function.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set auto port extension.
2. Press **Cal** key.
3. Click **Port Extensions** > **Auto Port Extension** > **Select Ports** to select the port(s) for which you want to use the auto port extension function.

2. Setting frequencies used for calculation

Set the frequency points with which you want to calculate a loss value.

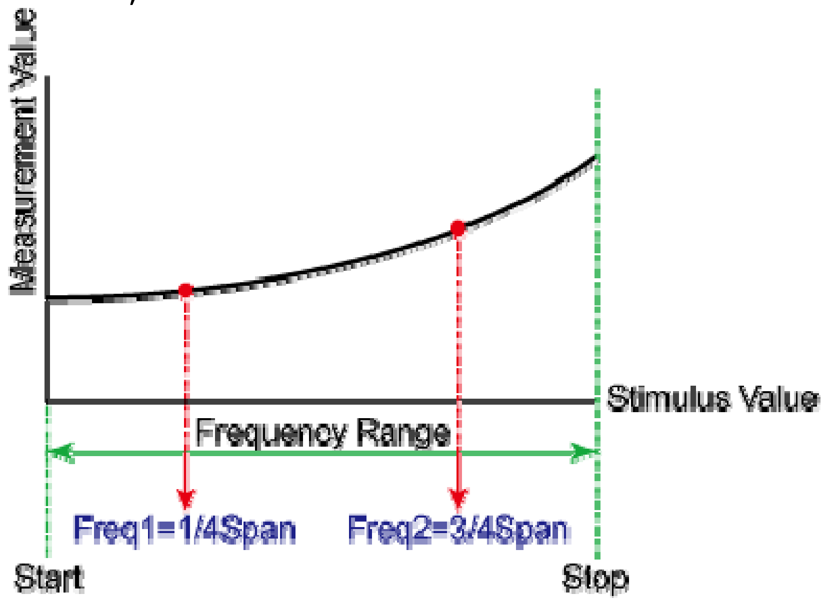
1. Press **Channel Next**/**Channel Prev** keys to activate the channel to set auto port extension.
2. Press **Cal** key.
3. Click **Port Extensions** > **Auto Port Extension** > **Select Ports** > **Method** to set the frequencies used for calculation.

Softkey	Function
Current Span	Executed using the frequency range set currently.
Active Marker	Executed using the frequency at the active marker. In this case, the result is applied to Loss1. Loss2 is ignored.
User Span	Executed using a start value and a stop value you set.

4. If you have selected **User Span**, use **User Span Start** and **User Span Stop** to set a start value and a stop value.

5. For **Current Span** and **User Span**, a frequency point at 1/4 of the frequency range is set to **Freq1**; a frequency point at 3/4 of the frequency range is set to **Freq2**.

6. If the setting is not made before starting OPEN/SHORT standard measurement, it does not affect the calculation result.



3. Specifying a loss value as a calculation target

Specify whether you want to include a loss value in the calculation result.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set auto port extension.
2. Press **Cal** key.
3. Click **Port Extensions** > **Auto Port Extension** > **Select Ports** > **Include Loss** to turn it on.

   NOTE If the setting is not made before starting the measurement of the OPEN/SHORT standard, it does not affect the calculation result.

4. Specifying a DC loss value as a calculation target

Specify whether you want to include a DC loss value in the calculation result.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to set auto port extension.
2. Press **Cal** key.
3. Click **Port Extensions** > **Auto Port Extension** > **Adjust Mismatch** to turn it on.

NOTE If the setting is not made before starting the measurement of the OPEN/SHORT standard, it does not affect the calculation result.

5. Measuring the OPEN/SHORT standard and executing calculation

Calculate port extensions and loss values based on the calculation results using the OPEN/SHORT standard.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel for which you want to want to set auto port extension.
2. Press **Cal** key.
3. Click **Port Extensions** > **Auto Port Extension**.
4. If you use the OPEN standard, click **Measure OPEN**, and select the port(s) for which you want to execute measurement. Execution is restricted to ports selected in Selecting a port(s)
5. If you use the SHORT standard, click **Measure SHORT**, and select the port(s) for which you want to execute measurement. Execution is restricted to ports selected in Selecting a port(s).

NOTE If a port extension value or loss value has been set, the value is updated to the calculated result.

NOTE If you execute both open measurement and short measurement, the average of the calculation results is reflected to the port extension and loss value.

6. Deleting the result of open/short measurement

When you exit from the softkey menu in the same level after open/short measurement, the measurement results are deleted. Note that you can use a GPIB command.

NOTE Port extension and loss values that have been calculated are not cleared.

Improving Measurement Throughput

This section explains the methods to improve measurement throughput.

- Change Sweep Mode
- Turning OFF the updating of information displayed on the LCD screen
- Turning OFF system error correction
- Turning ON the Fixed RF Range

Other topics about Optimizing Measurements

Change Sweep Mode

The E5071C provides the two sweep modes: Stepped Mode and Swept Mode.

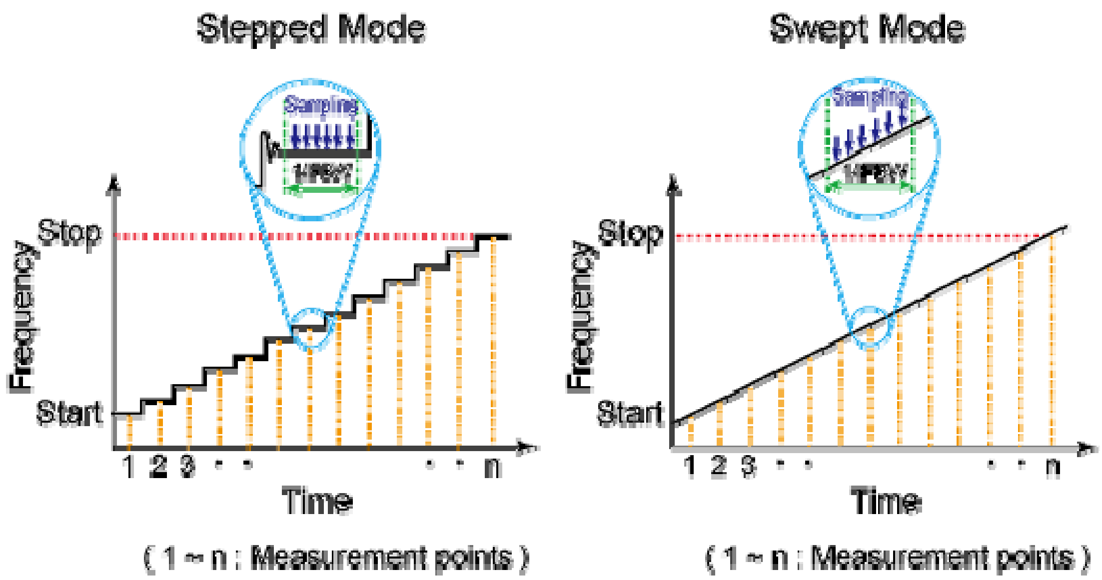
Mode	Stepped	Swept
Measurement Reliability	High	Low
Sweep Time	Long	Short
Restrictions on measurement	No Particular restriction	DUT with long electrical delay cannot be measured correctly.

Procedure to select the sweep mode

1. Press **Sweep Setup** key.
2. Click **Sweep Mode.**
3. Select **Stepped** or **Swept** for your appropriate sweep mode.

Overview

In the stepped mode, the frequency is changed stepwise and sampling is performed at a fixed frequency for each measurement point. On the other hand, in the swept mode, sampling is performed with the frequency always swept for each measurement point.



e5071c397

In the stepped mode, a certain time should be allowed until the frequency becomes stable at each measurement point, since the frequency is changed stepwise. Therefore, the sweep time in the swept mode is generally shorter than that in the stepped mode. However, if the measurement point interval is extremely large, the sweep to the next measurement point frequency cannot be completed within the sampling time due to the limitations of the instrument's frequency sweep speed. This causes a certain waiting time until the start of measurement of the next measurement point, and thus in this case the swept mode actually has a longer sweep time.

Although there is some difference depending on the IF bandwidth setting, when the measurement point interval is approximately 30 MHz or less, the sweep time in the swept mode is shorter than that in the stepped mode.

When you use the swept mode, you should confirm that there is no measurement-related problem before performing the actual measurement.

- The swept mode has the following disadvantages.

- DUTs with long electrical delay time cannot be measured correctly. For more information, refer to Notes for measuring DUTs with long electrical delay time
- Trace noise may increase compared to the stepped mode because sampling is performed while sweeping the frequency.
- The specifications are not guaranteed

Notes for measuring DUTs with long electrical delay time

When sweeping the frequency of a signal applied to the DUT (F), there is a frequency difference between the input side and output side of the DUT due to the delay time that occurs in the DUT (Delta T). This frequency difference (Delta F) becomes larger as the electrical delay time of the DUT becomes longer and the frequency sweep speed becomes faster, as shown in the following equation.

$$\Delta F = \frac{dF}{dt} \times \Delta T$$

When measuring a DUT with long electrical delay time, if you perform measurement (sampling) while sweeping the measurement signal as in the swept mode, a measurement error occurs due to the difference between the frequency outputted from the source port and the frequency actually measured at the receiver port.

Therefore, when measuring a DUT with a long electrical delay time, generally use the stepped mode to prevent the measurement error described above. However, if you need to shorten the sweep time, evaluate the measurement error as described below to determine the sweep mode that should be used.

Turning OFF the updating of information displayed on the LCD screen

If you are using the E5071C with HDD revision CFxxx or CNxxx, turning OFF the updating of information displayed on the LCD screen eliminates the processing time required to update displays within the analyzer, improving measurement throughput. If it is not necessary to check displayed information during measurements, turning OFF real-time updating is an effective means of improving throughput.

The updating of information displayed on the LCD screen can be switched using the following procedure:

Turning OFF the updating of information

1. Press **Display** key.
2. Click **Update** to switch the updating of displayed information on the LCD screen ON/OFF.

When the LCD screen update is turned OFF, **Update Off** appears on Instrument Status Bar.

For HDD revision CHxxxx, turning OFF the information update on the LCD screen does not contribute to any improvement in measurement throughput.

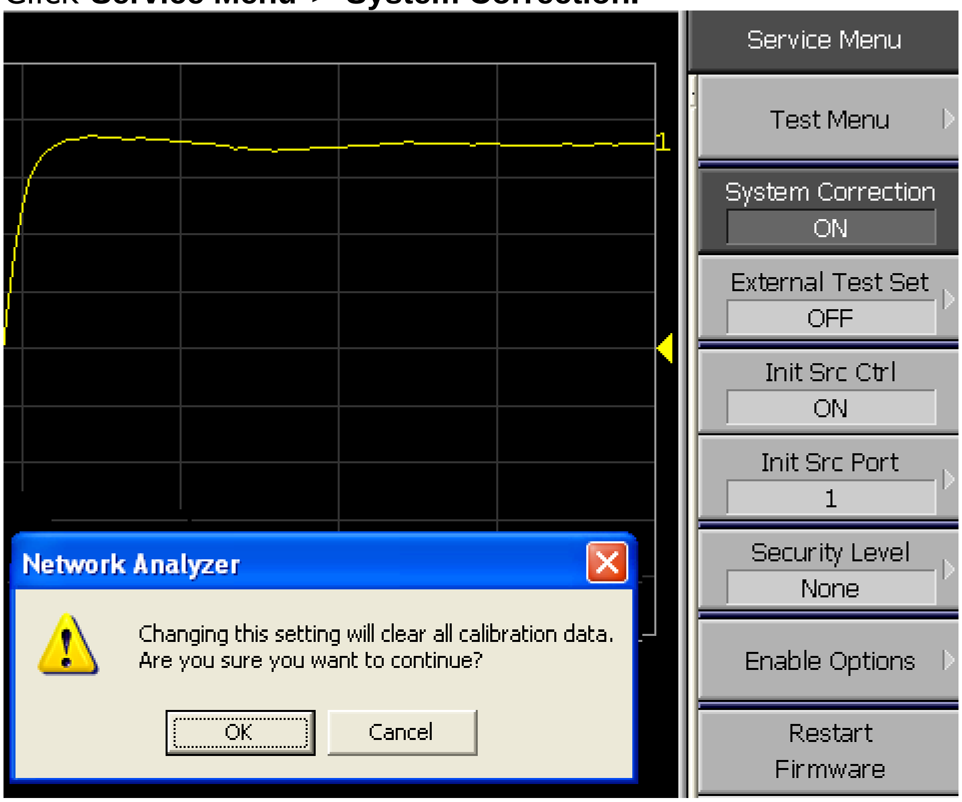
Turning OFF system error correction

The E5071C executes Port Characteristics Correction in the data processing flow shown in Data Processing Flowchart, by using the system calibration data set at the factory. This system error correction process is not required if the user performs proper calibration by using the **Cal** key and the softkeys that subsequently appear, which automatically turns on error correction.

By turning OFF system error correction, you can reduce the data processing time needed during measurement and thus improve measurement throughput.

NOTE When you turn ON/OFF system error correction, all calibration data set by user calibration is deleted.

1. Press **System** key.
2. Click **Service Menu** > **System Correction.**



e5071c074

3. Click **OK** to turn ON/OFF system error correction.

Turning ON the Fixed RF Range

The E5071C sets proper RF ranges of all test ports prior to every sweep. When you turn RF Range Fixed mode ON, the E5071C fixes RF ranges of all test ports so that it reduces 2 ms per each sweep from setting time before the measurement. This reduction will not be displayed in Sweep Time, though you can verify it by monitoring Handler I/O signals or *OPC? query.

This setting is available at the firmware revision A.11.00 or higher.

To turn ON/OFF the RF ranging, follow the below procedure:

1. Press **System** key.
2. Click **Service Menu** > **RF Range Fixed**.
3. Toggle between the **ON** and **OFF** selection. When **ON** is selected,setting time is shortened by 2ms. This setting is applied to all the channels.

NOTE This function is applicable for the E5071C with maximum frequency option of 8.5 GHz and below (option x4x/x6x/x8x).

NOTE This function does not reduce setting time when you use the E5091A/92A with the E5071C, though it is possible to turn ON/OFF this function.

NOTE If the RF Range Fixed mode status is changed after a calibration is performed, the calibration data will be useless and **C!** will be displayed. See Checking Calibration Status for details. In this case, it is strongly recommended to perform a calibration again.

NOTE Turning ON this function may degrade some analog performances such as dynamic range. See the data sheet for detail.

Performing a Segment-by-Segment Sweep

This section describes the concept of the segment sweep and how to perform it.

- Concept of Segment Sweep
- Conditions for Setting Segment Sweep
- Items that can be set for Each Segment
- Sweep Delay/Sweep Time in Segment Sweep
- Turning ON/OFF Segments Individually
- Frequency/Order Base Display
- Arbitrary Segments
- Setting Segment Sweep Procedure

Other topics about Optimizing Measurements

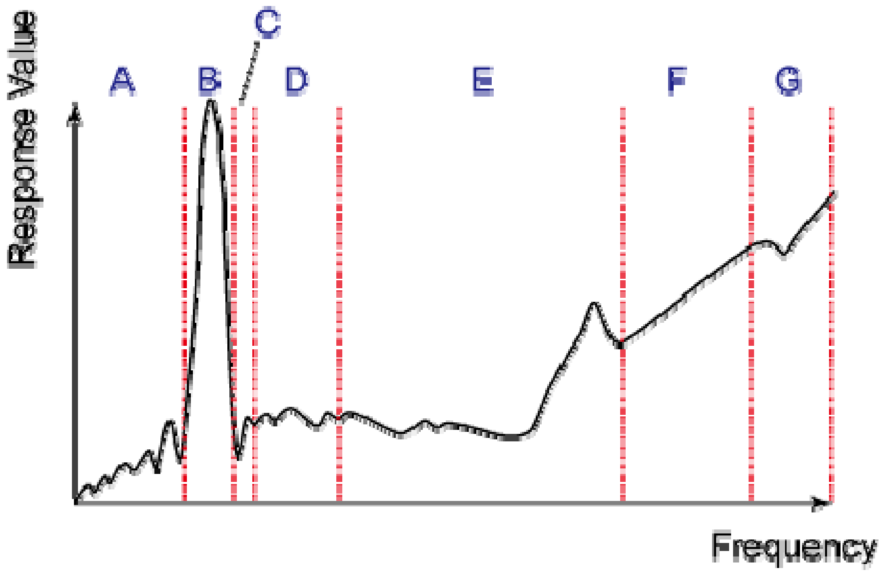
Concept of Segment Sweep

To perform a segment sweep, you must define two or more frequency ranges, called segments, and then specify the number of points, IF bandwidth, power level, sweep mode, sweep delay time, and sweep time for each segment. All segments are swept sequentially as if swept in one sweep operation.

NOTE By skipping the frequency range, which does not need to be measured, you can sweep and measure only the portions you need.

NOTE You can define the optimum measurement conditions for each of the segments you designate. For example, you can specify as many points as possible in a segment requiring high trace resolution and as few points as possible in a segment not requiring high resolution. This shortens the measurement time, enabling you to optimize the overall measurement throughput by not having to perform the entire operation under the same measurement conditions of a particular frequency range.

To evaluate a band pass filter having the transmission characteristics shown in the following figure. for example, you can select the frequency ranges you need from A through G and determine the measurement conditions shown in the table below This enables you to measure them simultaneously in one sweep operation.



e5071c393

Frequency ranges (segments) from the figure above and their measurement conditions							
	Start frequency	**Stop frequency**	**Number of points**	**IF Bandwidth**	**Sweep mode**	**State**	**Shift LO**
A	440 MHz	915 MHz	50	50 kHz	Stepped	ON	OFF
B	915 MHz	980 MHz	130	70 kHz	Stepped	ON	OFF
C	980 MHz	1.035 GHz	60	50 kHz	Stepped	ON	OFF
E	1.07 GHz	2 GHz	100	70 kHz	Swept	ON	OFF
G	2.6 GHz	3 GHz	40	70 kHz	Swept	ON	OFF

Conditions for Setting Segment Sweep

The following conditions apply when setting up a segment sweep.

- The frequency range of a segment must not overlap with that of another segment. (The start frequency of a segment must be higher than the stop frequency of the immediately preceding segment).
- The start frequency of segment 1 must be greater than lowest frequency and the stop frequency of the last segment less than

highest frequency, as per the frequency range depending on the option.

- When the start frequency and stop frequency of a segment are not the same, you can define from 2 to 1601 points in a segment.
- When the start frequency and stop frequency of a segment are the same, you can define from 1 to 1601 points in a segment.
- You can set the total number of points in the segment table from 2 to 1601.
- You can set the number of segments in the segment table to between 1 and 201.
- You can turn ON or OFF segments.
- You can turn ON or OFF the Shift LO.

Items that can be set for Each Segment

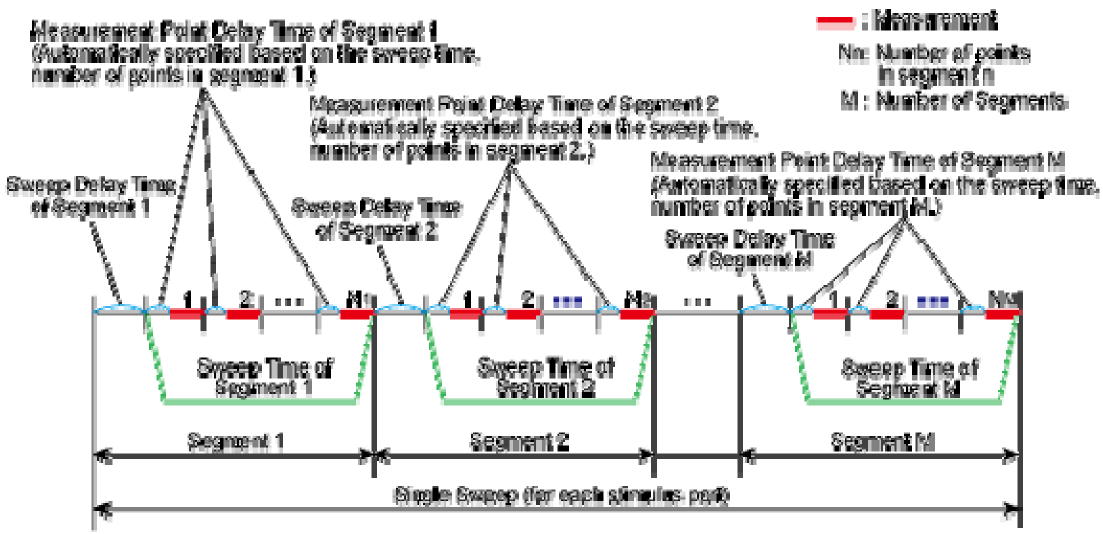
For the segment sweep, you can set the sweep range, the number of points, IF bandwidth, power level, sweep delay time, sweep mode, sweep time and segment ON/OFF for each segment.

You can set the items in the following table to ON/OFF for each segment. If you enable the segment-by-segment setting, you can make the setting for each segment in the segment table; if you disable it, the setting in the following table is used.

Item	When segment-by-segment setting is disabled
IF bandwidth	For all segments, the IF bandwidth for the linear/log sweep (set with **Avg** key > **IF Bandwidth**) is set. IF bandwidth can be set independently for each port in the segment sweep.
Power level	For all segments, the power level for the linear/log sweep (set with **Sweep Setup** key > **Power**) is set. Power level can be set independently for each port in the segment sweep.
Sweep delay time	For all segments, 0 is set.
Sweep mode	For all segments, the sweep mode for the linear/log sweep (set with **Sweep Setup** key > **Sweep Mode**) is set.
Sweep time	For all segments, the auto sweep time mode is set.
Segment ON/OFF	For all segments, ON is set.
Shift LO	For all segments, OFF is set.

Sweep Delay/Sweep Time in Segment Sweep

The definitions for sweep delay time and sweep time, which you can specify in the segment sweep, are shown in figure below.



e5071c306

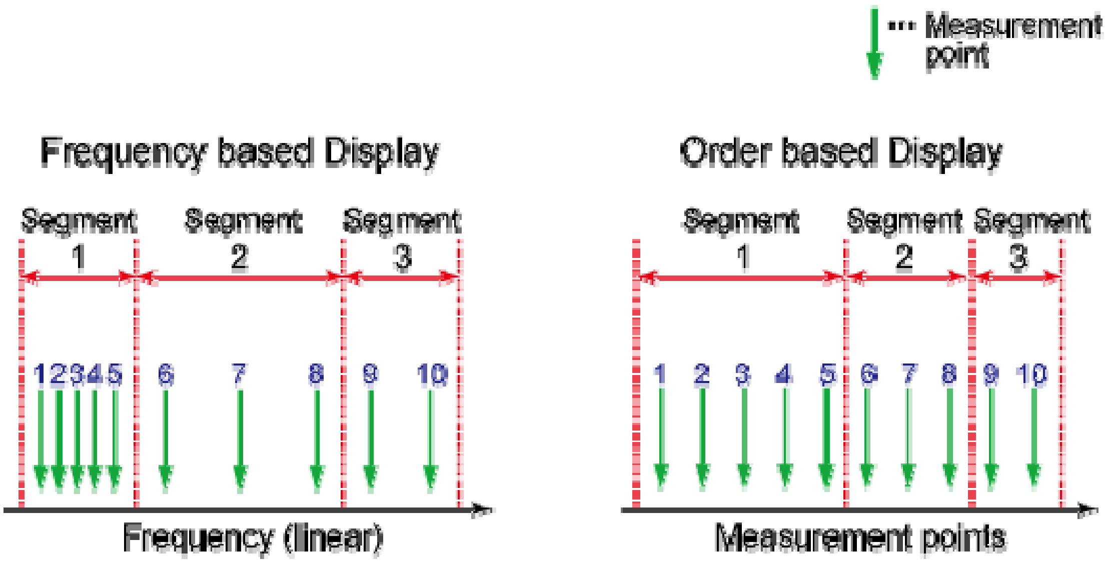
Turning ON/OFF Segments Individually

From the firmware revision A.09.60, each segment can be turned ON or OFF individually. The minimum number of points turned ON must be 2.

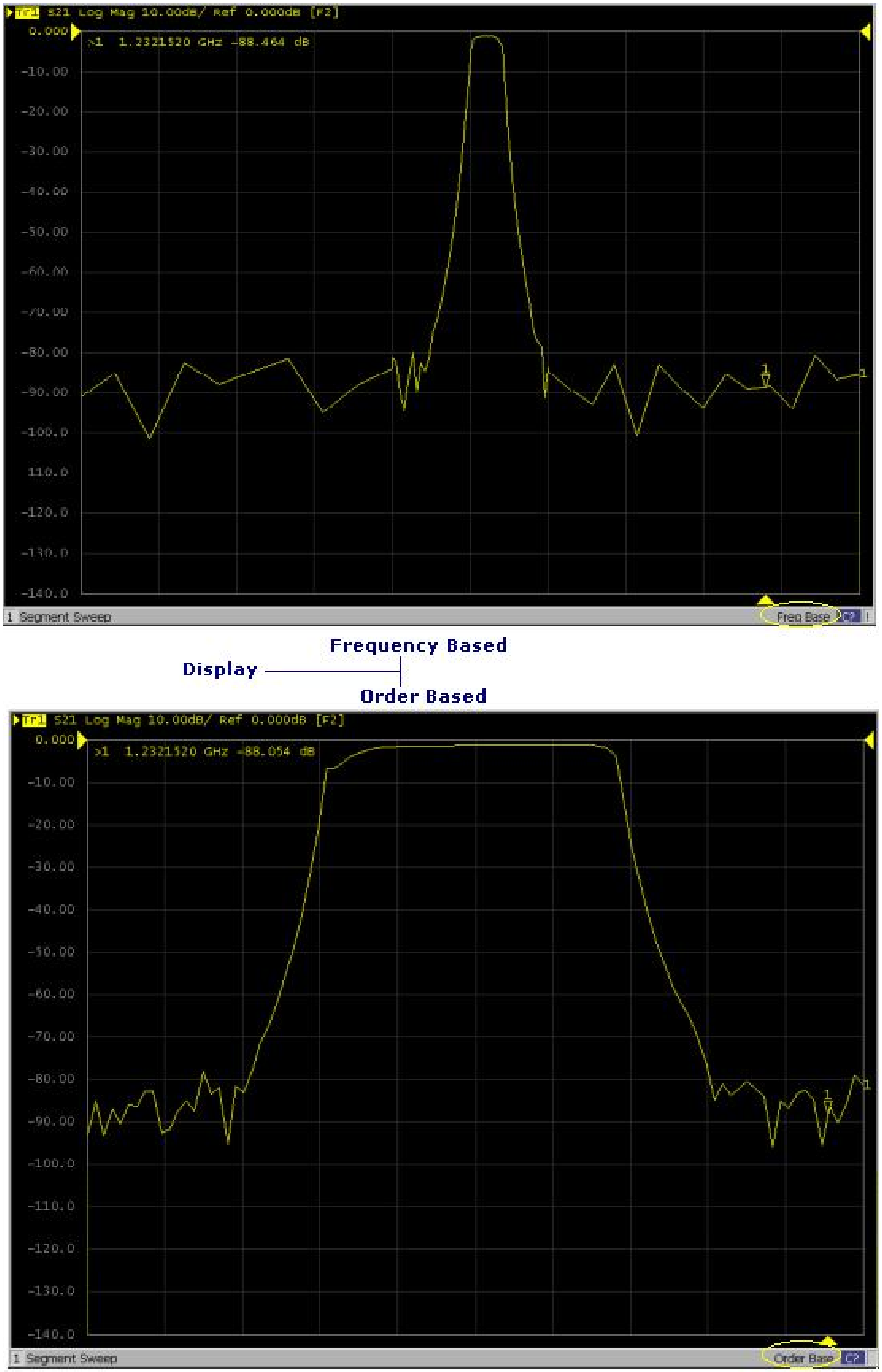
If the total number of points turned ON are less than 2, the segments with the highest frequency ranges is automatically turned ON to meet the minimum number of points.

Frequency/Order Base Display

You can choose between frequency-based and order-based display as the method of displaying traces when executing a segment sweep.


Measurement point
Frequency based Display
Segment 1
Segment 2
Segment 3
1 2 3 4 5 6 7 8 9 10
Frequency (linear)
Order based Display
Segment 1
Segment 2
Segment 3
1 2 3 4 5 6 7 8 9 10
Measurement points

e5071c394


Tr1 S21 Log Mag 10.00dB/ Ref 0.000dB [F2]
>1 1.2321520 GHz -88.464 dB
1 Segment Sweep
Freq Base
Frequency Based
Display
Order Based
Tr1 S21 Log Mag 10.00dB/ Ref 0.000dB [F2]
>1 1.2321520 GHz -88.054 dB
1 Segment Sweep
Order Base

e5071c082

Arbitrary Segments

With this function, the segments can have the below features:

- Overlapping frequencies
- The stop frequency is less than the start frequency (reverse sweep).

However, there are several limitation:

- Sweep mode: Stepped only.
- Sweep time: When reverse sweep is performed, the sweep time and the wait time before the measurement point becomes longer.
- Segment display: It is recommended to use Order Base Segment Display (**Sweep Setup** > **Segment Display** > **Order Base**) as the Frequency Base Segment Display may cause incorrect results.

You can turn ON arbitrary segments by following the steps below:

1. Press **Sweep Setup**> **Edit Segment Table** > **Arbitrary Segments** > **ON**.

NOTE Unusual results may occur when using arbitrary segments with markers, display settings, limit lines, formatting and some calibration features.

NOTE When the segment table has both forward and reverse frequency sweep, the correction interpolation may not work properly. Use the same segment table setting at both, correction and measurement. (Make a measurement at the condition where "Cor" is displayed.)

NOTE When the number of data of segment table exceeds its limitation, the error of "Segment Data Overflow" occurs. In this case, reduce the measurement data size (i.e. NOP, number of channels).

Procedure

Creating a segment table

1. Press **Channel Next/Channel Prev** keys to select the channel for which you want to create the segment table.
2. Press **Sweep Setup** key.
3. Click **Edit Segment Table**. The segment table appears in the lower part of the screen.
4. To change the frequency range setting mode or to set the IF bandwidth, power level, sweep delay time, sweep mode, sweep time for each segment, and segment ON/OFF use the following softkeys.

   NOTE When setting the segment table using the front panel keys or keyboard, you need to place focus on (select) the operation target (segment table of softkey) first. You can change the focus by pressing **Focus** key in the **ENTRY** Block. When the focus is placed on the segment table, the window frame of the segment table is displayed as bright as the window frame of the active channel. When the focus is placed on the softkey menu, the softkey menu title area is displayed in blue.

Softkey	Function
Freq Mode	Switches the frequency range setting mode (start/stop or center/span).
List IFBW	Toggles ON/OFF the IF bandwidth setting for each segment; the row for setting (**IFBW**) only appears in the segment table when this is ON.
List Power	Toggles ON/OFF the power level setting for each segment; the row for setting (**Power**) only appears in the segment table when this is ON. NOTE If you toggle **List Power** ON, the linear/log sweep (set with **Sweep Setup** > **Power**) setting is ignored.
List Port Power	Toggles ON/OFF the port power level setting for each segment; the rows for setting (**Power 1** to **Power 2** or **Power 4**) only appears in the segment table when this is ON. NOTE When both List Power and List Port Power is turned ON, the older setting is turned OFF automatically.
List Delay	Toggles ON/OFF the sweep delay time setting for each segment; the row for setting (**Delay**) only appears in the segment table when this is ON
List Sweep Mode	Toggles ON/OFF the sweep mode setting for each segment; the row for setting (**Sweep Mode**) only appears in the segment table when this is ON
List Time	Toggles ON/OFF the sweep time setting for each segment; the row for setting (**Time**) only appears in the segment table when this is ON
List State	Toggles ON/OFF the state of each segment; the row for setting (**State**) only appears in the segment table when this is ON
List Shift LO	Toggles ON/OFF the Shift LO setting for each segment; the row for setting (**Shift LO**) only appears in the segment table when this is ON
List Port IFBW	Toggles ON/OFF the IF bandwidth setting of the sweep stimulated from port X (X = 1 to 2, or to 4) for each segment. For example, "IFBW 2" refers to the IF bandwidth of the sweep stimulated from Port 2. The rows for setting (**IFBW 1** to **IFBW 2** or **IFBW 4**) only

	appears in the segment table when this is ON. The below conditions should be met. In the event the conditions are not met, the conditions will be changed automatically: Sweep Mode: Stepped Sweep Time: Auto Minimum Frequency: 6.26 MHz NOTE When both List IFBW and List Port IFBW is turned ON, the older setting is turned OFF automatically.

5. Enter each item in the following table for each added segment (line) to create the segment table. To create the segment table, use the hardkeys and softkeys.

Start	Sets the start value of the sweep range
Stop	Sets the stop value of the sweep range
Center	Sets the center value of the sweep range
Span	Sets the span value of the sweep range
Points	Sets the number of points
IFBW	Sets the IF bandwidth
Power	Sets the power level; the power range is common to the settings for the linear/log sweep (**Sweep Setup** key > **Power Range**)
Port Power	Sets the power level for Port 1, Port 2, Port 3 and Port 4 independently
Delay	Sets the sweep delay time
Sweep Mode	Sets the sweep mode; you need to select one of the following items: **STEPPED**: Stepped mode **SWEPT**: Swept mode
Time	Sets the sweep time; to specify the auto setting (**AUTO**), enter 0 as the sweep time
State	Sets the state of each segment
Shift LO	Sets the state of Shift LO.
Port IFBW	Sets the IF BW for Port 1, Port 2, Port 3 and Port 4 independently

Useful functions when using a mouse

By right-clicking on the selected cell, you can use the following shortcut menu.

Shortcut	Function
Copy	Copies the value in the selected cell into the clipboard (internal temporary storage memory)
Paste	Pastes the value data in the clipboard to a newly selected cell
Insert	Adds a new line above the selected cell
Delete	Deletes the line containing the selected cell

In the character-by-character edit mode, you can also use the following shortcut menu.

Shortcut	Function
Undo	Undoes the change and restore the value before the change
Cut	Cuts the selected string and store it into the clipboard (temporary memory)
Copy	Copies the selected string into the clipboard
Paste	Pastes the string in the clipboard to a newly selected cell
Delete	Deletes the selected string
Select All	Selects the entire string in the cell

Executing segment sweep

To execute a segment sweep by using the segment table you have created, you must specify the sweep type for that sweep operation by following the steps below.

1. Press **Channel Next/Channel Prev** keys to select the channel on which you will execute the segment sweep operation.
2. Press **Sweep Setup** Key.
3. Press **Sweep Type** > **Segment**.

Setting up the segment display

Define the method of displaying traces when the segment sweep is executed by following the steps described below.

1. Press **Channel Next/Channel Prev** keys to select the channel on which you will define the segment display.
2. Press **Sweep Setup** Key.
3. Click **Segment Display**.

4. Select the segment display.

Softkey	Function
Freq Base	Displays the X-axis as the axis for linear frequencies (frequency-based display)
Order Base	Displays the X-axis as the axis for the points (order-based display)

Saving a newly created segment table in CSV format

As discussed in Creating a segment table, you can export the newly created segment table as a CSV (comma-separated value) formatted file (so it can be used easily in software that requires a different format).

1. Press **Sweep Setup** Key.
2. Click **Edit Segment Table** > **Export to CSV File** to open the Save As dialog box. Note that CSV Files (***.csv**) will already be selected as the file type when the dialog box first opens.
3. Type the file name in the File Name area and Click **Save** to save the segment table.

Calling a segment table saved in CSV Format

By importing a segment table file saved by E5071C, you can set up the segment table.

NOTE It is possible to recall a file from a different channel where it was saved.

1. Press **Sweep Setup** Key.
2. Click **Edit Segment Table** > **Import from CSV File** to open the Open dialog box. Note that CSV Files (***.csv**) will already be selected as the file type when the dialog box first opens.
3. Select the CSV format file to be imported, and click **Open** to call up the segment table.

NOTE You cannot import a CSV-formatted file created/edited in spreadsheet software into the E5071C.

NOTE You cannot import CSV-formatted files created in firmware revision A.09.60 or later to firmware revisions before revision A.09.60.

Measurement Examples

Measurement Examples

- Measuring the SAW Bandpass Filter Using the Segment Sweep
- Evaluating a Duplexer
- Measuring the Deviation from a Linear Phase
- Measuring an Unbalanced and Balanced Bandpass Filter
- Measuring Parameters with Cable
- Evaluating Transmission Characteristics of a Front End Module
- Executing Power Calibration

Measuring SAW Bandpass Filter using Segment Sweep

- Overview
- Procedure

Other Measurement Examples

Overview

This section illustrates how to use the segment sweep function to evaluate a SAW bandpass filter with a center frequency of 947.5 MHz.

Procedure

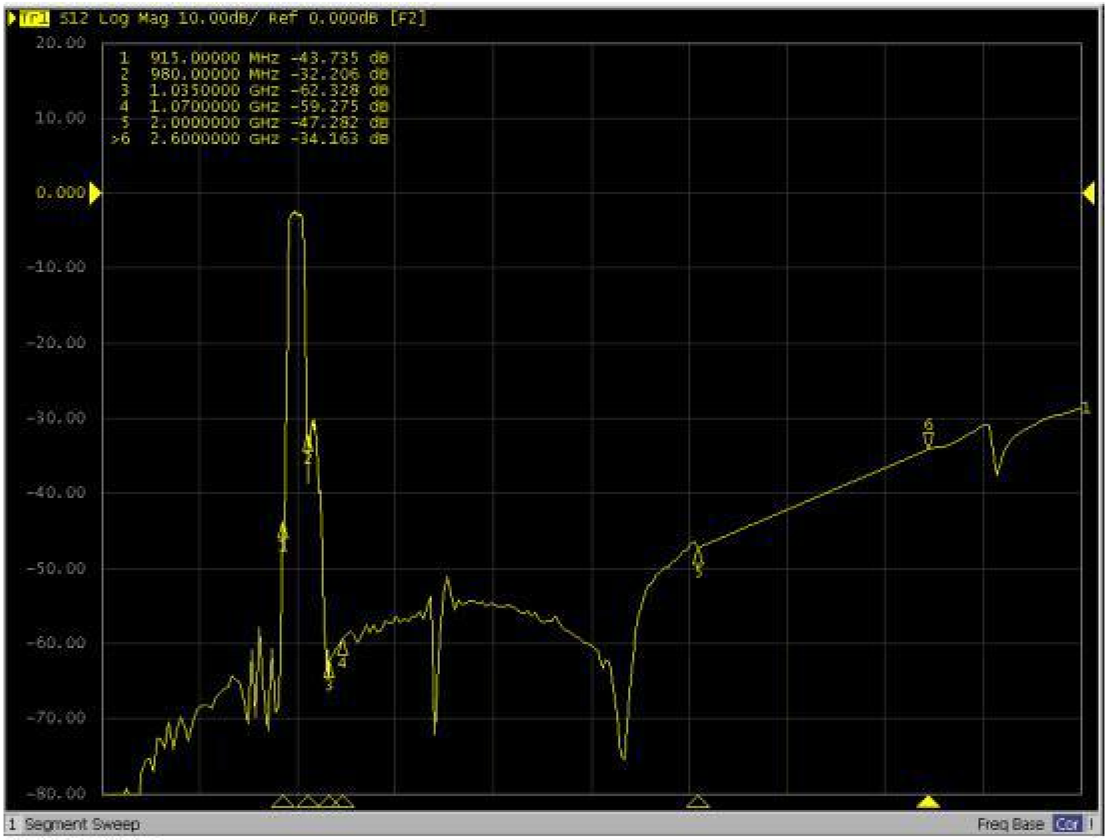
Here, the DUT is evaluated by following the steps.

Step	Description
1. Determine the Segment Sweep Conditions	The segment sweep conditions are determined according to the characteristics of the DUT.
2. Create a Segment Sweep Table	The segment sweep conditions are entered in the E5071C.
3. Select the Segment Sweep as the Sweep Type	The segment sweep is selected as the sweep type.
4. Execute the Calibration	A 2-port ECal is performed between the test ports connecting the DUT.
5. Connect the DUT	The DUT is connected.
6. Execute the Measurement	A trigger is applied to perform the measurement.
7. Define the Setup for Display	The choice is made between frequency base and order base as the method of displaying segments.

1. Determine the Segment Sweep Conditions

The following figure shows the results of evaluating the transmission characteristics of the SAW bandpass filter in the range of 440 MHz to 3 GHz by using the linear sweep.

Transmission characteristics of SAW bandpass filter (440 MHz to 3 GHz, linear sweep)



e5071c101

The measurement conditions are determined for each frequency range. Here, the segment sweep is performed following the sweep conditions shown in the following table.

Frequency Range		**Measurement Conditions**	
Start	**Stop**	**Number of Points**	**IF Bandwidth**
440 MHz	915 MHz (Marker 1)	47	70 kHz
915 MHz (Marker 1)	980 MHz (Marker 2)	130	100 kHz
980 MHz (Marker 2)	1.035 GHz (Marker 3)	55	70 kHz
1.07 GHz (Marker 4)	2 GHz (Marker 5)	93	70 kHz
2.6 GHz (Marker 6)	3 GHz	41	70 kHz

2. Create a Segment Sweep Table

Follow the steps below to make entries in the segment sweep table.

1. Display the segment table.

Setup Description	Key Operation
Presetting	**Preset** > **OK**
Displaying the segment table	**Sweep Setup** > **Edit Segment Table**

2. Display the IF bandwidth setting column in the segment table.

Setup Description	Key Operation
Moving the focus to the softkey menu	**Focus**
Display of IF bandwidth setting column: ON	**List IFBW** (turn it **ON**)

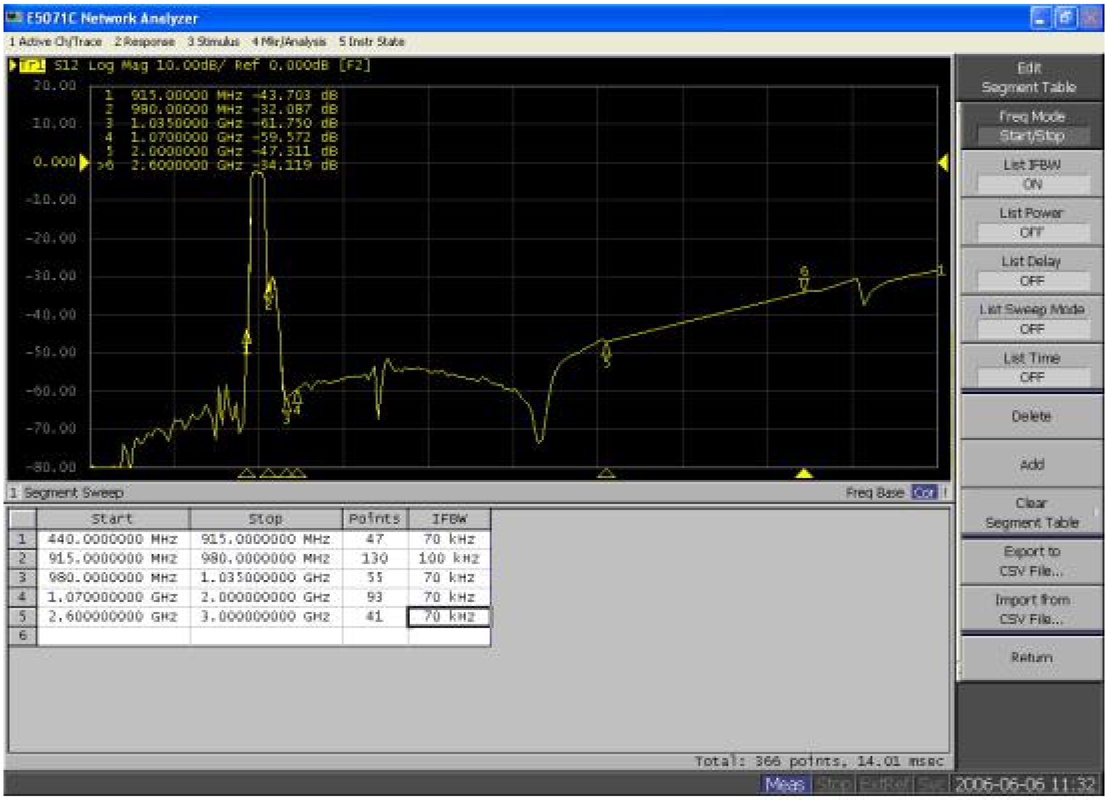
When setup items (power level, delay time, sweep mode, and sweep time in this case) are not displayed in the segment table, the setting for the channel in use applies to all segments.

3. Enter the setup data in the segment table.

Setup Description	Key Operation
Moving the focus to the segment table (select)	**Focus**
Segment 1	
Start frequency: 440 MHz	**4** > **4** > **0** > **M/μ**
Stop frequency: 915 MHz	**9** > **1** > **5** > **M/μ**
Number of points: 47	**4** > **7** > **x1**
IF bandwidth: 70 kHz	**7** > **0** > **k/m**
Segment 2	
Start frequency: 915 MHz	**9** > **1** > **5** > **M/μ**
Stop frequency: 980 MHz	**9** > **8** > **0** > **M/μ**
Number of points: 130	**1** > **3** > **0** > **x1**
IF bandwidth: 100 kHz	**1** > **0** > **0** > **k/m**

Segment 3	
Start frequency: 980 MHz	**9** > **8** > **0** > **M/µ**
Stop frequency: 1.035 GHz	**1** > **.** > **0** > **3** > **5** > **G/n**
Number of points: 55	**5** > **5** > **x1**
IF bandwidth: 70 kHz	**7** > **0** > **k/m**
Segment 4	
Start frequency: 1.07 GHz	**1** > **.** > **0** > **7** > **G/n**
Stop frequency: 2 GHz	**2** > **G/n**
Number of points: 93	**9** > **3** > **x1**
IF bandwidth: 70 kHz	**7** > **0** > **k/m**
Segment 5	
Start frequency: 2.6 GHz	**2** > **.** > **6** > **G/n**
Stop frequency: 3 GHz	**3** > **G/n**
Number of points: 41	**4** > **1** > **x1**
IF bandwidth: 70 kHz	**7** > **0** > **k/m**

Completed segment table



e5071c102

3. Select the Segment Sweep as the Sweep Type

The segment sweep is selected as the sweep type.

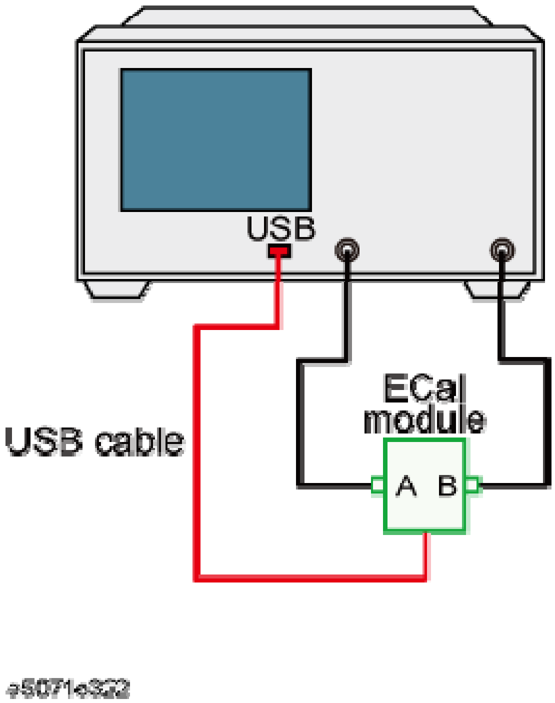
Setup Description	Key Operation
Sweep type: Segment sweep	Sweep Setup > **Sweep Type** > **Segment**

4. Execute the Calibration

In this step, a 2-port ECal is executed on the two ports to be used.

1. Connect the ECal module across test ports 1 and 2.

Connecting the ECal module



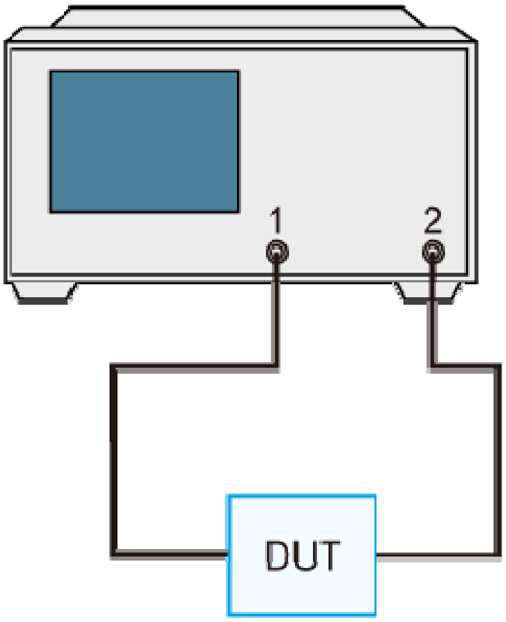
2. Execute the 2-port ECal.

Setup Description	Key Operation
Executing a 2-port ECal between test ports 1 and 2	**Cal** > **ECal** > **2 Port ECal** > **Port 1-2**

5. Connect the DUT

The DUT is connected across test ports 1 and 2.

Connecting the DUT



e5071c320

6. Execute the Measurement

A trigger is applied to perform the measurement.

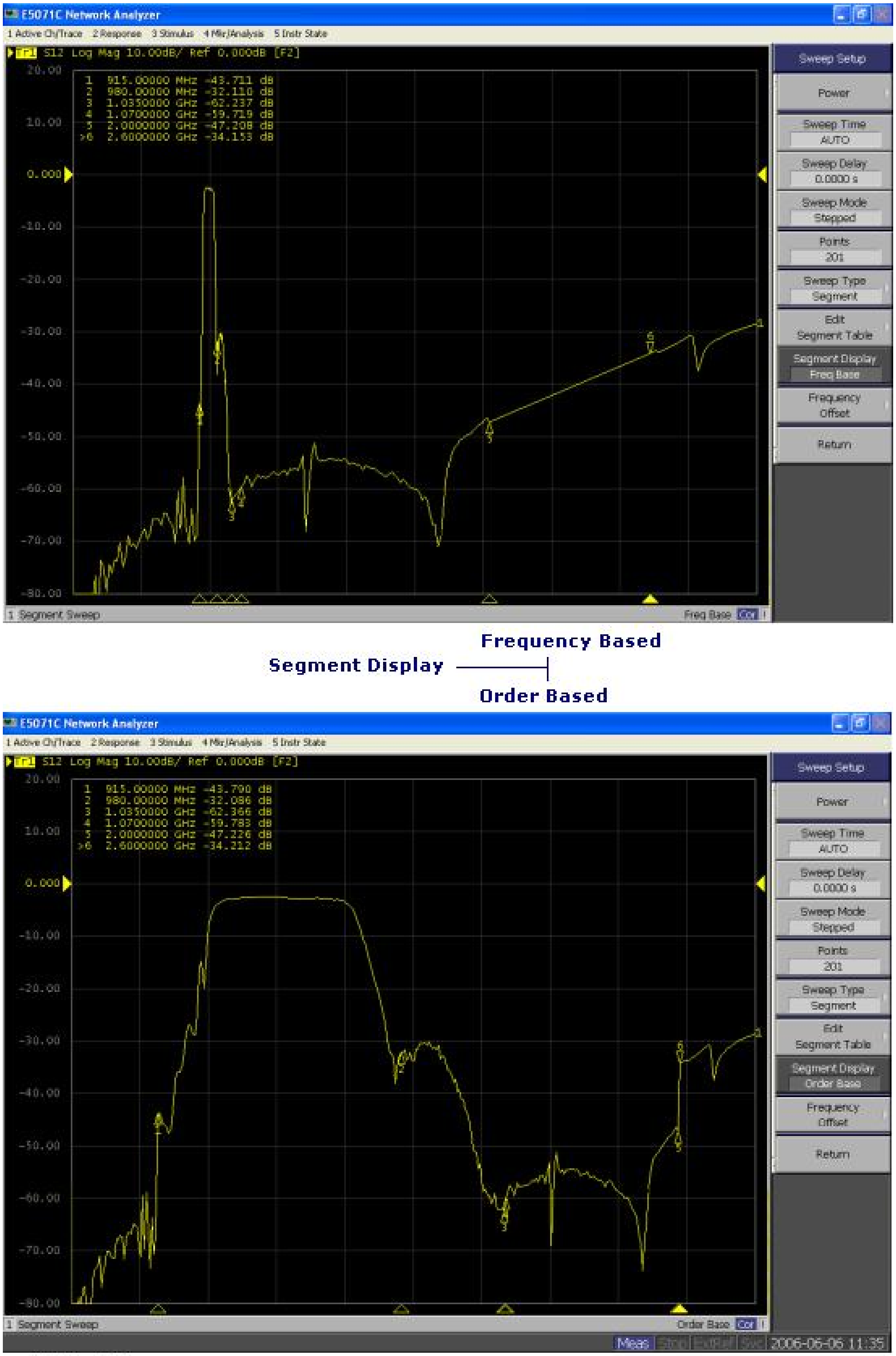
Setup Description	Key Operation
Trigger mode: Single	Trigger > **Single** (Or **Continuous**)

7. Define the Setup for Display

The choice is made between frequency base and order base as the segment display mode.

Setup Description	Key Operation
Segment display: Frequency base or order base	Sweep Setup > **Segment Display** > **Frequency Base** | **Order Base**

Segment display: Frequency/Order base


E5071C Network Analyzer
1 Active Ch/Trace 2 Response 3 Stimulus 4 Mkr/Analysis 5 Instr State
Tr1 S12 Log Mag 10.00dB/ Ref 0.000dB [F2]
1 915.00000 MHz -43.711 dB
2 980.00000 MHz -32.110 dB
3 1.0350000 GHz -62.237 dB
4 1.0700000 GHz -59.719 dB
5 2.0000000 GHz -47.208 dB
>6 2.6000000 GHz -34.153 dB
Sweep Setup
Power
Sweep Time AUTO
Sweep Delay 0.0000 s
Sweep Mode Stepped
Points 201
Sweep Type Segment
Edit Segment Table
Segment Display Freq Base
Frequency Offset
Return
1 Segment Sweep
Freq Base
Frequency Based
Segment Display
Order Based
E5071C Network Analyzer
1 Active Ch/Trace 2 Response 3 Stimulus 4 Mkr/Analysis 5 Instr State
Tr1 S12 Log Mag 10.00dB/ Ref 0.000dB [F2]
1 915.00000 MHz -43.790 dB
2 980.00000 MHz -32.086 dB
3 1.0350000 GHz -62.366 dB
4 1.0700000 GHz -59.783 dB
5 2.0000000 GHz -47.226 dB
>6 2.6000000 GHz -34.212 dB
Sweep Setup
Power
Sweep Time AUTO
Sweep Delay 0.0000 s
Sweep Mode Stepped
Points 201
Sweep Type Segment
Edit Segment Table
Segment Display Order Base
Frequency Offset
Return
1 Segment Sweep
Order Base
Meas
2006-06-06 11:35

e5071c103

Evaluating a Duplexer

- Overview
- Procedure

Other Measurement Examples

Overview

This section illustrates how to evaluate a duplexer (Tx center frequency: 1.88 GHz, Rx center frequency: 1.96 GHz).

Procedure

Here, the DUT is evaluated by following the steps described in table below.

Steps	Description
1. Determine the Segment Sweep Conditions	Segment sweep conditions are determined according to the characteristics of the DUT.
2. Create a Segment Sweep Table	The segment sweep conditions are entered in the E5071C.
3. Select the Segment Sweep as the Sweep Type	The segment sweep is selected as the sweep type.
4. Execute the Calibration	A full 3-port calibration is executed by using the 2-port ECal module.
5. Connect the DUT	The DUT is connected.
6. Define the Setup for Display	The number of traces to be displayed, split display, and measurement parameters are specified.
7. Execute the Measurement	A trigger is applied to execute the measurement.
8. Define the Setup for the Segment Display and Scale	The setup for segment display and for the scale are defined.
9. Analyze the Parameters	The evaluation parameters for the duplexer are determined.
10. Define the Setup for a Limit Table	The setup for the limit table is defined.
11. Execute the Limit Test	The limit test is executed.

1. Determine the Segment Sweep Conditions

A segment sweep is performed by following the sweep conditions shown in table below.

Start	Stop	Number of Points
1.73 GHz	1.83 GHz	50
1.83 GHz	2.03 GHz	400
2.03 GHz	2.13 GHz	50
3.65 GHz	4.03 GHz	38
5.5 GHz	6.02 GHz	52

2. Create a Segment Sweep Table

Entries are made in the segment sweep table following the steps described below.

1. Display the segment table.

Setup Description	Key Operation
Presetting	**Preset** > **OK**
Displaying the segment table	**Sweep Setup** > **Edit Segment Table**

2. Enter the setup data in the segment sweep table.

In this step, the IF Bandwidth, power level, delay time, and sweep time are not entered segment by segment. By turning off the display of those parameters on the segment table, you can use, in each segment without making a change:

- the IF Bandwidth (preset value: 70 kHz) of the channel specified by using **Avg** > **IF Bandwidth**
- Power level (preset value: 0 dBm) of the channel specified by using **Sweep Setup** > **Power**
- Sweep delay time (preset value: 0 sec) of the channel specified by using **Sweep Setup** > **Sweep Delay**
- Sweep mode (preset value: Stepped mode) of the channel specified by using **Sweep Setup** > **Sweep Mode**
- Sweep time (preset value: Automatic) of the channel specified by using **Sweep Setup** > **Sweep Time**.

e5071c083

3. Select the Segment Sweep as the Sweep Type

The segment sweep is selected as the sweep type.

Setup Description	Key Operation
Sweep type: Segment sweep	**Focus** (Moves the focus to the softkey menu) > **Return** > **Sweep Type** > **Segment** (or **Sweep Setup** > **Sweep Type** > **Segment**)

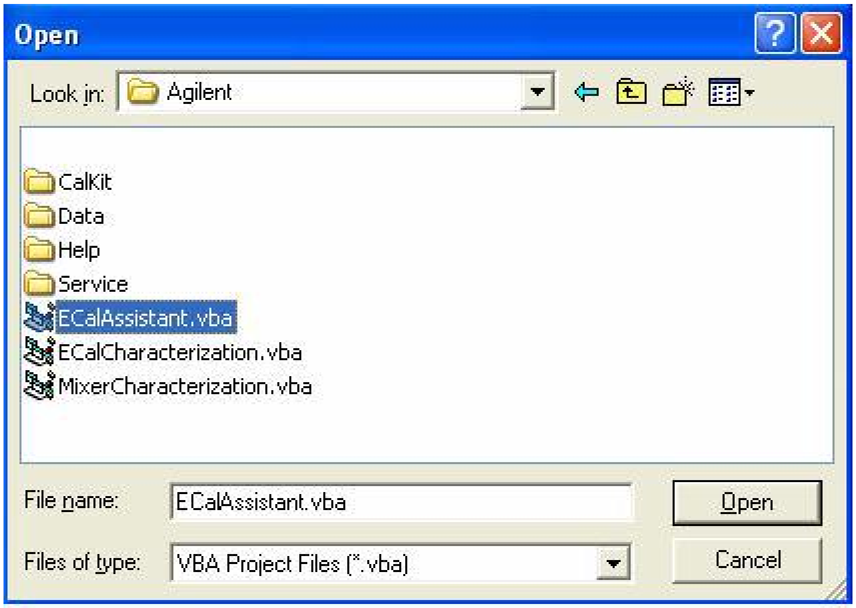
4. Execute the Calibration

In this step, a 2-port ECal module and 3-/4-port module installed in the E5071C are used to execute calibration on the three ports used in the measurement.

1. Connect the USB port of the 2-port ECal module and the USB port of the E5071C with a USB cable. The connection may be made while the unit is powered.
2. Load and execute the 3-/4-port ECal programs.

Setup Description	Key Operation
Opening the VBA Project Open dialog box	**Macro Setup** > **Load Project**
Loading **ECalAssistant.VBA**	Select **D:\Agilent\ECalAssistant.VBA** and press the **Open** button.
Executing the program	**Macro Run**

3. The EcalAssistant (start) dialog box appears.



e5071c084

4. Click **Next** to display the EcalAssistant (port/channel selection) dialog box.

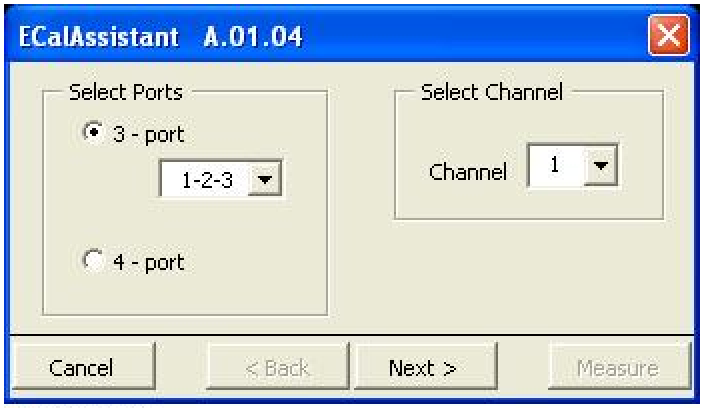
e5071c085

5. Following the instructions in the dialog box, select the type of ECal, test ports, and the channel.

Setup Description	Key Operation
Select Ports	
ECal type: Full 3-port calibration	**3 Port**

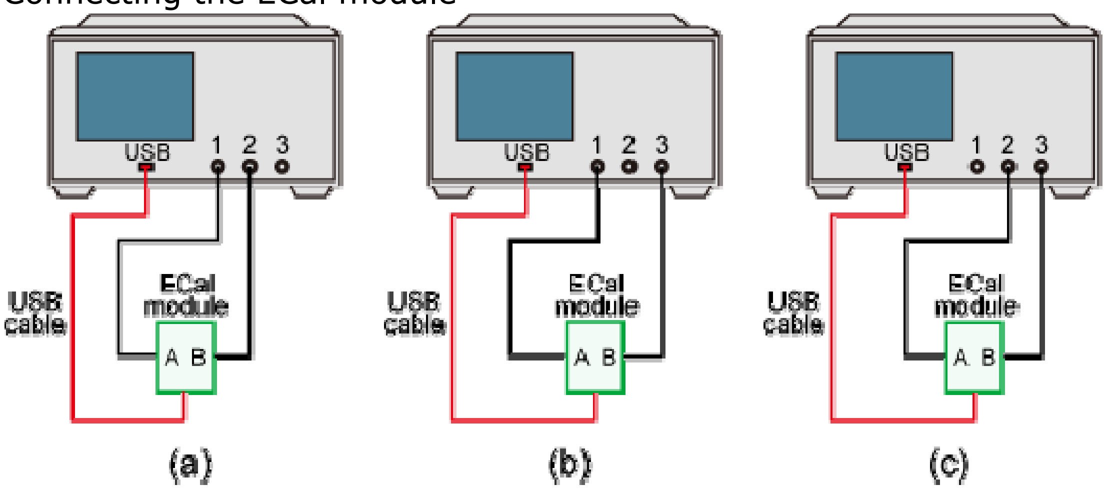
Test ports to be used for ECal: 1, 2, and 3	**1, 2, 3**
Select Channel	
Channel on which ECal is to be executed: Channel 1	**Channel: 1**

6. Click **Next**. The EcalAssistant (connection) dialog box appears. Select 3-port/Channel:1.



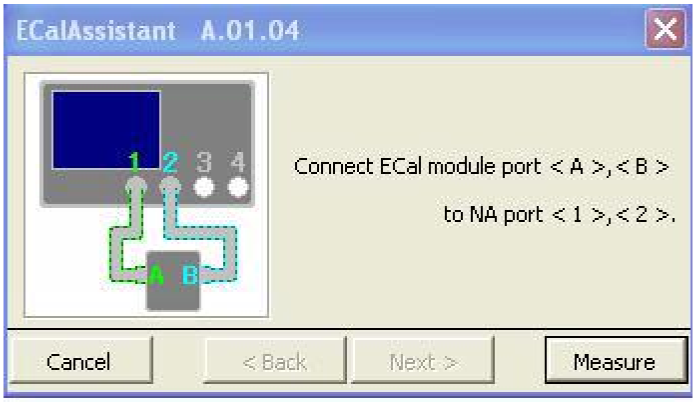
e5071c086

Connecting the ECal module



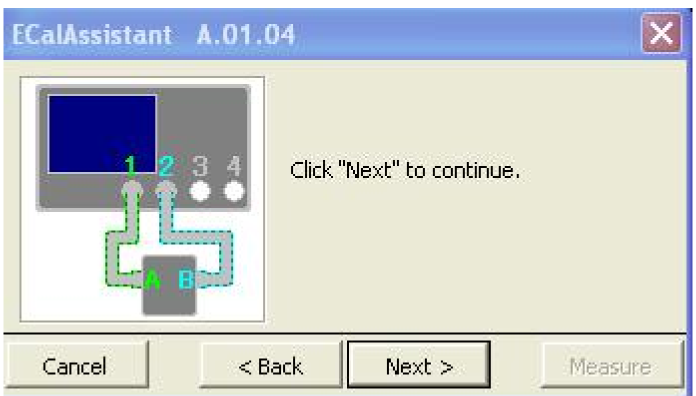
e5071c361

7. Click **Next.** The calibration dialog box for Port 1-2 appears.



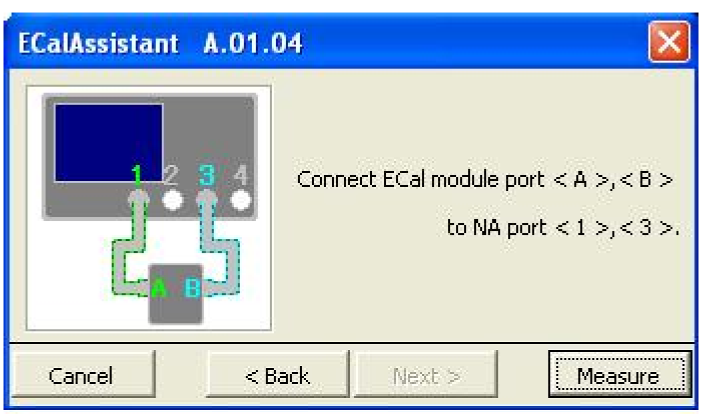
e5071c087

8. Click **Measure** to Calibrate Port 1-2 settings. After successful calibration of Port 1-2, a new dialog box appears.



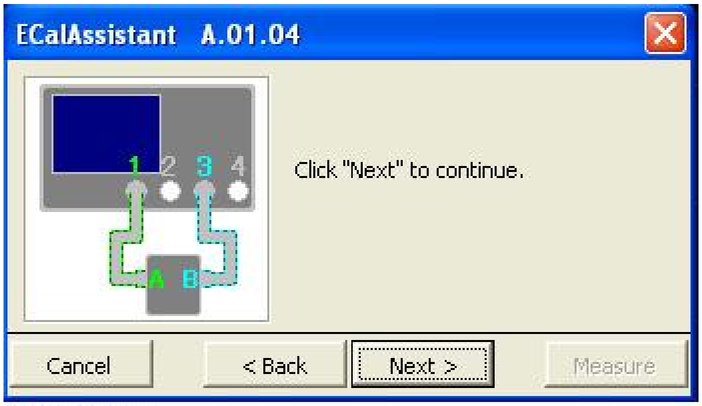
e5071c088

9. Click **Next.** The dialog box for the calibration of port 1-3 appears.



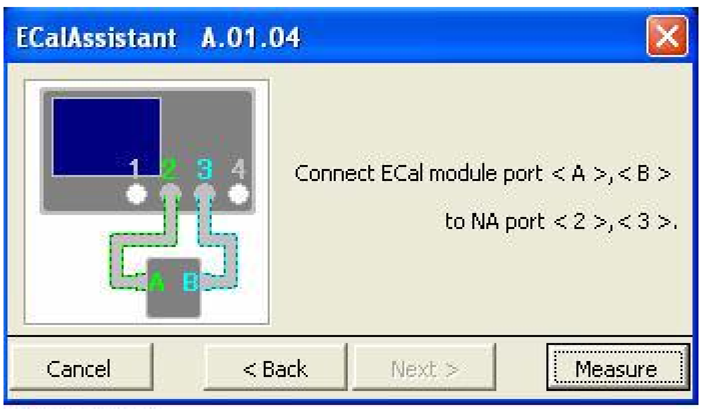
e5071c089

10. Click **Measure** to Calibrate Port 1-3 settings. After successful calibration of Port 1-3, a new dialog box appears.



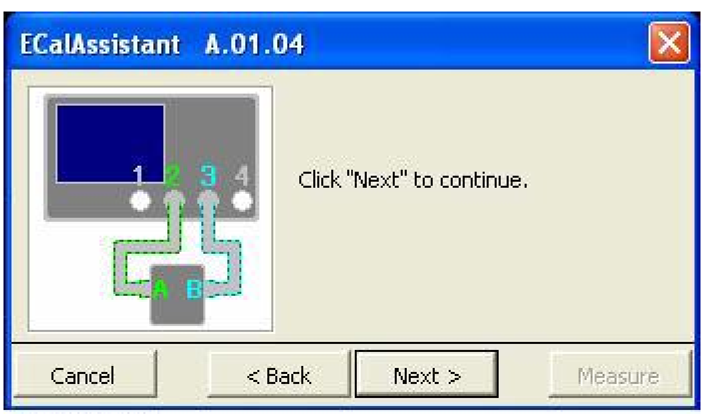
e5071c090

11. Click **Next**. The dialog box for the calibration of port 2-3 appears.



e5071c100

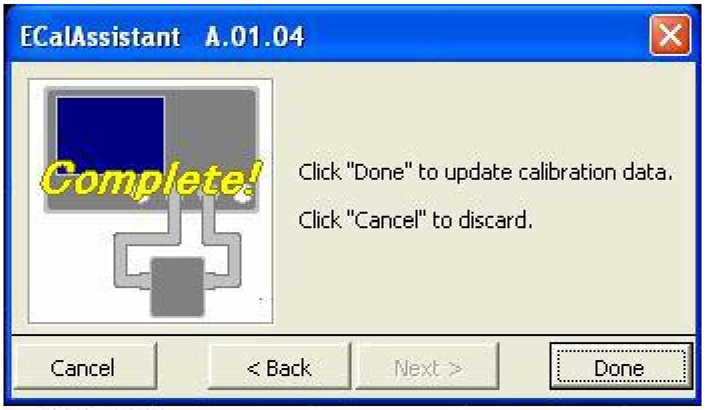
12. Click **Measure** to Calibrate Port 2-3 settings. After successful calibration of Port 2-3, a new dialog box appears.



e5071c098

13. The EcalAssistant (complete) dialog box appears. Click **Done** to terminate the calibration.

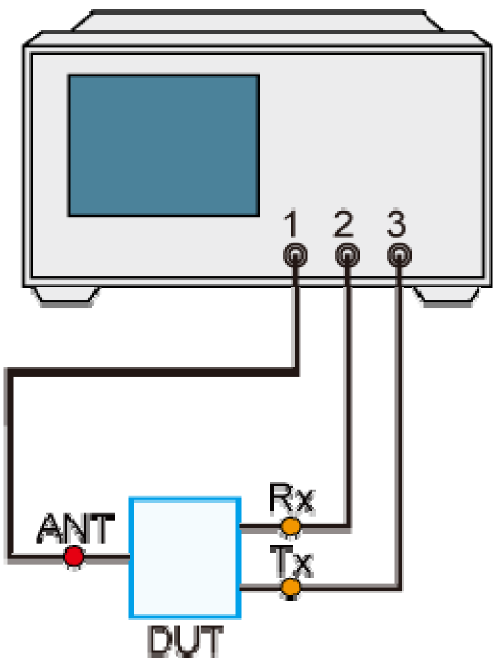
ECalAssistant (complete) dialog box



e5071c099

5. Connect the DUT

The DUT is connected to test ports 1, 2, and 3.

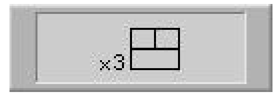


e5071c359

6. Define the Setup for Display

The setup for display is defined.

Setup Description	Key Operation
Number of traces to be displayed: 5	**Display** > **Number of Traces** > **5**

Trace placement: Trisected	**Allocate Traces** > 
Measurement Parameter	
Trace 1: S13	**Meas** - **S13**
Trace 2: S21	**Trace Next** > **Meas** > **S21**
Trace 3: S23	**Trace Next** > **Meas** > **S23**
Trace 4: S33	**Trace Next** > **Meas** > **S33**
Trace 5: S11	**Trace Next** > **Meas** > **S11**

7. Execute the Measurement

A trigger is applied to execute the measurement.

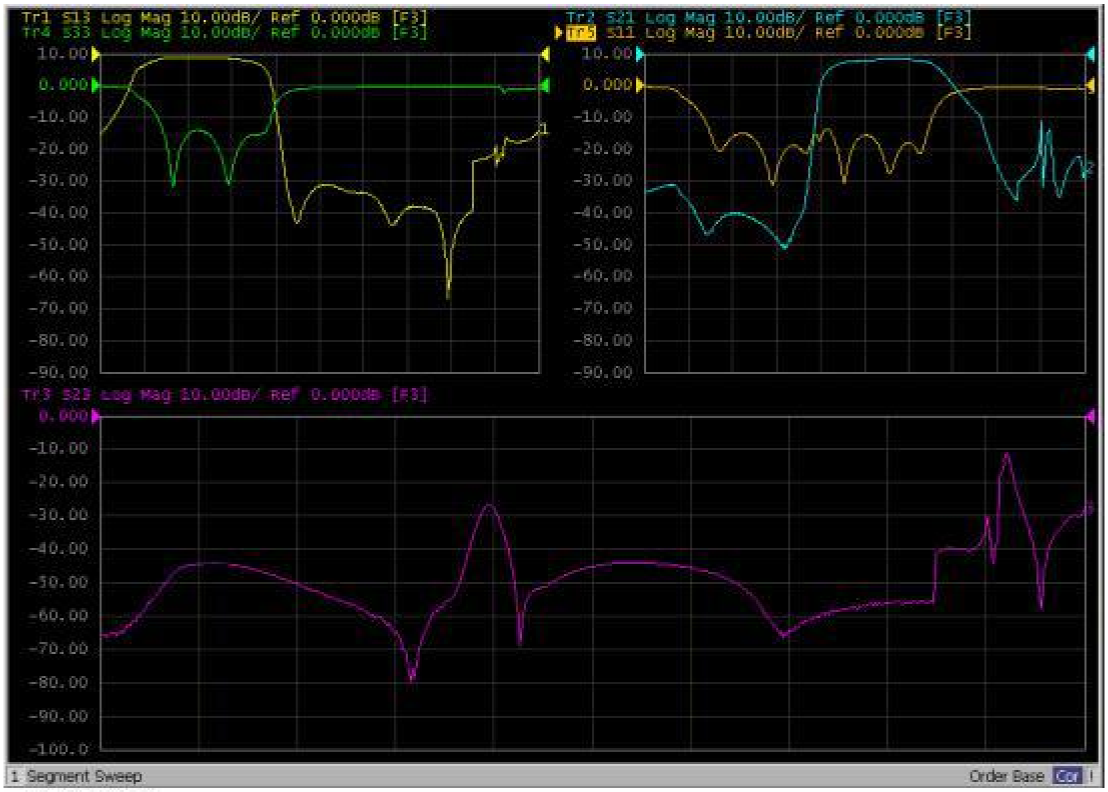
Setup Description	Key Operation
Trigger mode: Single (or continuous)	**Trigger** > **Single** (or **Continuous**)

8. Define the Setup for the Segment Display and Scale

The setup for the scale is defined.

Setup Description	Key Operation
Segment display: Order base	**Sweep Setup** > **Segment Display** > **Order Base**
Reference Line Position	
Trace 1: 10	**Trace Next** > **Scale** > **Reference Position** > **1** > **0** > **x1**
Trace 2: 10	**Trace Next** > **Reference Position** > **1** > **0** > **x1**
Trace 3: 10	**Trace Next** > **Reference Position** > **1** > **0** > **x1**
Trace 4: 9	**Trace Next** > **Reference Position** > **9** > **x1**
Trace 5: 9	**Trace Next** > **Reference Position** > **9** > **x1**

Measurement result (segment display: order base)



e5071c091

9. Analyze the Parameters

The parameters for the duplexer are determined.

1. Determine the insertion loss and 3-dB bandwidth for Tx.

Setup Description	Key Operation
Marker coupling: OFF	**Marker Fctn** > **Couple** (Turn it **OFF**)
Activating Trace 1	**Trace Next**
Marker 1: ON	**Marker**
Search/Tracking: ON	**Marker Search** > **Tracking** (Turn it **ON**)
Moving the marker 1 to the trace maximum	**Max**
Bandwidth search: ON	**Bandwidth** (Turn it **ON**)

The insertion loss (**loss**) in this example is -1.243 dB, and the 3-dB bandwidth (**BW**) is 85.53 MHz.

2. Determine the insertion loss and 3-dB bandwidth for the Rx.

Setup Description	Key Operation
Activating Trace 2	**Trace Next**
Marker 1: ON	**Marker**
Search/Tracking: ON	**Marker Search** > **Tracking** (Turn it **ON.**)
Moving Marker 1 to the trace maximum	**Max**
Bandwidth search: ON	**Bandwidth** (Turn it **ON**)

The insertion loss (**loss**) in this example is -1.627 dB and the 3-dB bandwidth (**BW**) is 71.04 MHz.

3. Determine the isolation between Tx and Rx.

Setup Description	Key Operation
Activating Trace 3	**Trace Next**
Marker 1: ON	**Marker**
Search/Tracking: ON	**Marker Search** > **Tracking** (Turn it **ON**)
Moving Marker 1 to the peak near 1.92 GHz	**Peak** - **Search Left** or **Search Right** (press as many times as necessary)

The isolation (response value of marker 1) in this example is -6.612 dB.

4. Determine the return loss of Tx.

Setup Description	Key Operation
Activating Trace 4	**Trace Next**
Marker 1: ON	**Marker**
Search/Tracking: ON	**Marker Search** > **Tracking** (Turn it **ON**)
Moving Marker 1 to the peak in the pass band	**Peak** > **Search Left** or **Search Right** (press as many times as necessary)

The return loss (response value of Marker 1) in this example is 12.65 dB.

5. Determine the return loss of Rx.

Setup Description	Key Operation
Activating Trace 5	**Trace Next**
Marker 1: ON	**Marker**

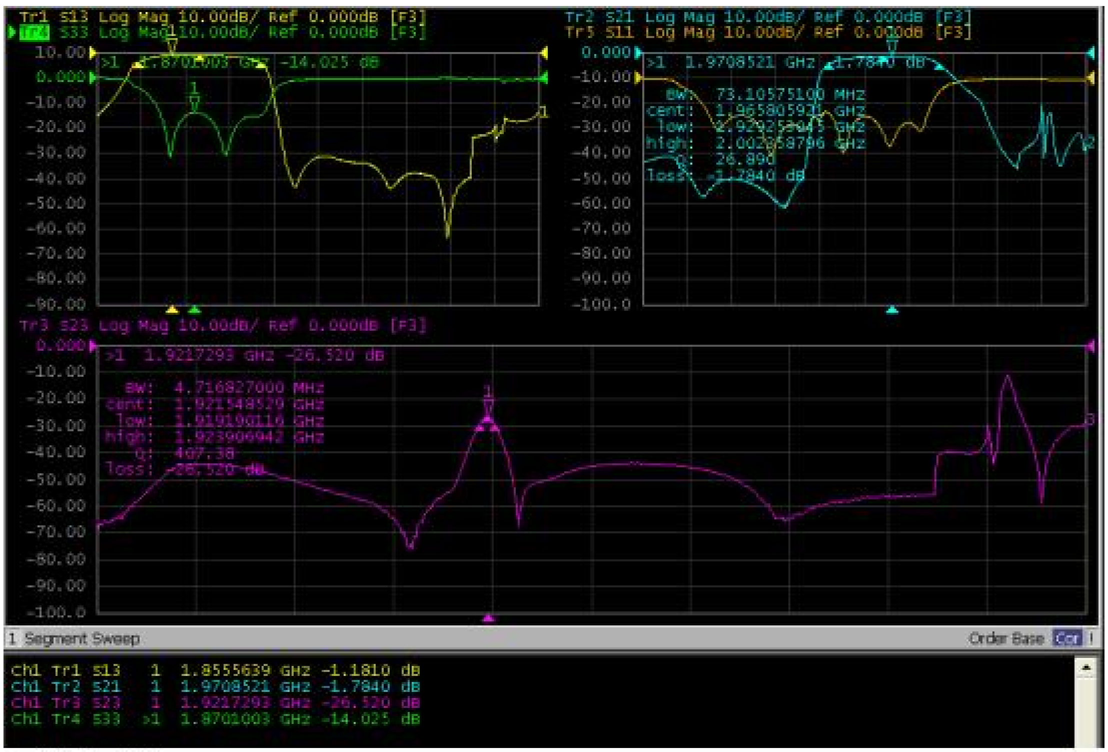
Search/Tracking: ON	**Marker Search** > **Tracking** (Turn it **ON**)
Moving Marker 1 to the peak in the pass band	**Peak** > **Search Left**|**Search Right** (press as many times as necessary)

The return loss (response value of Marker 1) in this example is 13.80 dB.

6. Turn on the marker table display.

Setup Description	Key Operation
Marker table display: ON	**Marker Fctn** > **Marker Table** (Turn it **ON**)

Activating Marker Table



e5071c092

10. Define the Setup for a Limit Table

Follow the steps below to make entries in the limit table.

1. Display the limit table for Trace 1 (S13).

Setup Description	Key Operation
Activating Trace 1	**Trace Next**
Displaying a limit table	**Analysis** > **Limit Test** > **Edit Limit Line**

2. Enter the setup data in the limit table for trace 1.

Completed limit table for trace 1

	Type	Begin Stimulus	End Stimulus	Begin Response	End Response
1	MAX	1.730000000 GHz	1.940000000 GHz	0 dB	0 dB
2	MIN	1.850000000 GHz	1.910000000 GHz	-4 dB	-4 dB
3	MAX	1.930000000 GHz	1.990000000 GHz	-35 dB	-35 dB
4	MAX	1.990000000 GHz	2.130000000 GHz	-30 dB	-30 dB
5	MAX	2.130000000 GHz	6.020000000 GHz	-20 dB	-20 dB
6					

e5071c093

3. Display the limit table for Trace 2 (S21).

Setup Description	Key Operation
Activating Trace 2	**Trace Next**

4. Enter the setup data in the limit table for trace 2.

Completed limit table for trace 2

	Type	Begin Stimulus	End Stimulus	Begin Response	End Response
1	MAX	1.730000000 GHz	1.850000000 GHz	-40 dB	-40 dB
2	MAX	1.850000000 GHz	1.910000000 GHz	-45 dB	-45 dB
3	MAX	1.910000000 GHz	6.020000000 GHz	0 dB	0 dB
4	MIN	1.930000000 GHz	1.990000000 GHz	-5 dB	-5 dB
5					

e5071c094

11. Execute the Limit Test

The limit test is executed.

1. Turn on the limit line and limit test for Trace 1.

Setup Description	Key Operation
Activating Trace 1	**Trace Prev**
Limit Line: ON	**Analysis** > **Limit Test** - **Limit Line** (Turn it **ON**)
Limit Test: ON	**Limit Test** (Turn it **ON**)

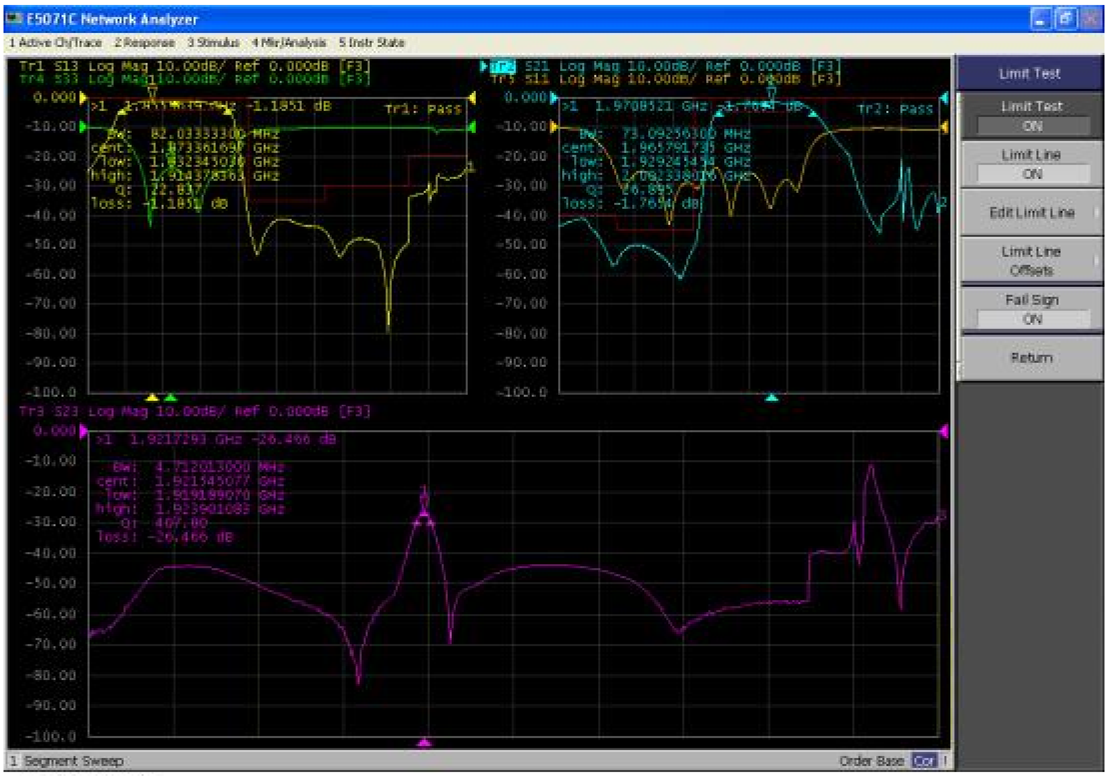
2. Turn on the limit line and limit test for Trace 2.

Setup Description	Key Operation
Activating Trace 2	**Trace Next**
Limit Line: ON	**Limit Line** (Turn it **ON**)
Limit Test: ON	**Limit Test** (Turn it **ON**)

3. Apply a trigger to execute the measurement.

Setup Description	Key Operation
Trigger Mode: Single	**Trigger** > **Single** (or **Continuous**)

Limit test results

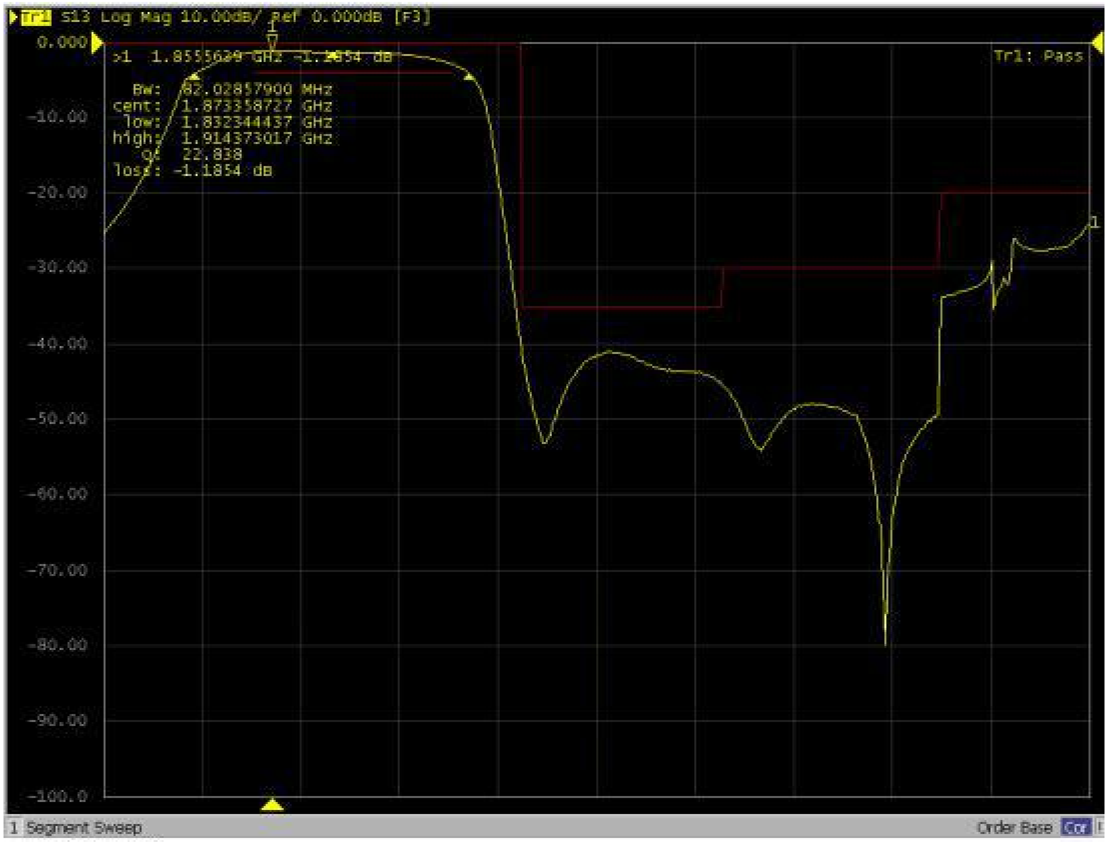


e5071c095

4. Maximize the screen display of Trace 1 to examine its details.

Setup Description	Key Operation
Activating Trace 1	**Trace Prev**
Maximizing the display of Trace 1	**Trace Max**

Enlarged display of trace 1

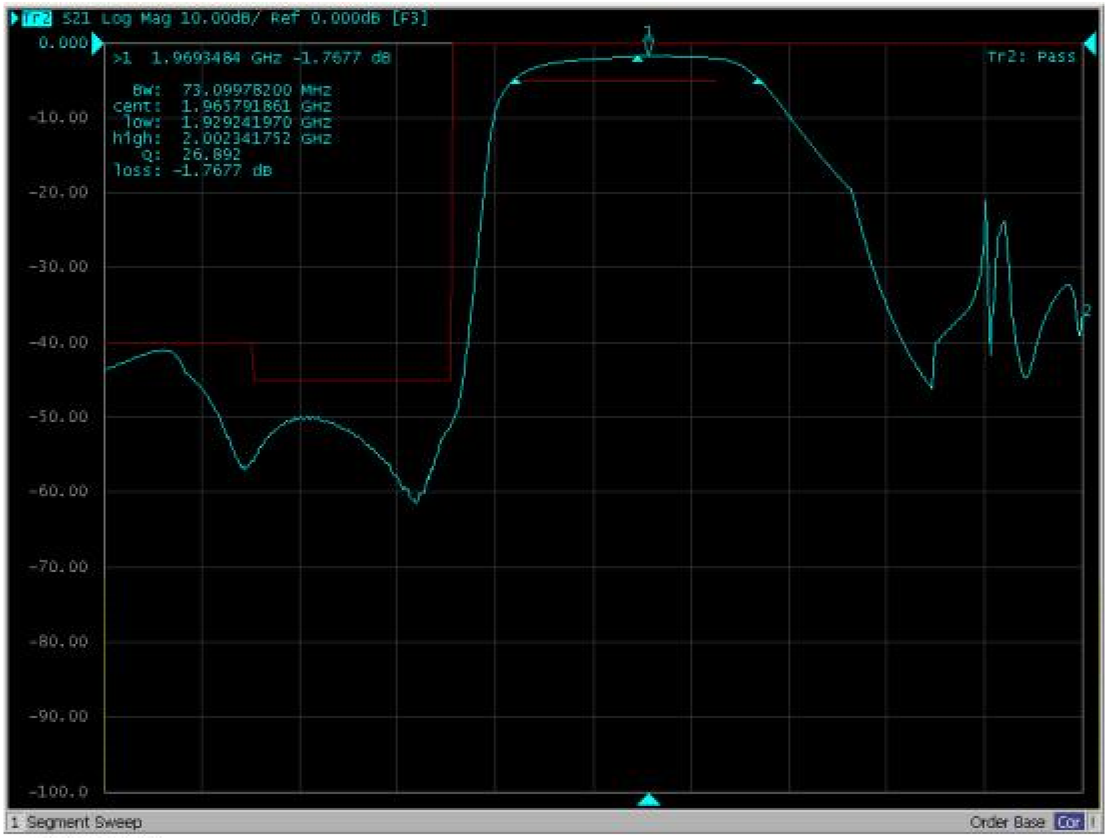


e5071c096

5. Maximize the screen display of Trace 2 to examine its details.

Setup Description	Key Operation
Activating Trace 2	**Trace Prev** (The display of Trace 2 is maximized.)

Enlarged display of trace 2


Tr2 S21 Log Mag 10.00dB/ Ref 0.000dB [F3]
>1 1.9693484 GHz -1.7677 dB
BW: 73.09978200 MHz
cent: 1.965791861 GHz
low: 1.929241970 GHz
high: 2.002341752 GHz
Q: 26.892
loss: -1.7677 dB
Tr2: Pass
0.000
-10.00
-20.00
-30.00
-40.00
-50.00
-60.00
-70.00
-80.00
-90.00
-100.0
1 Segment Sweep
Order Base
Cor

e5071c097

Measuring the Deviation from a Linear Phase

- Overview
- Procedure

Other Measurement Examples

Overview

This section illustrates how to determine the deviation from a linear phase in the pass band of a 1.09-GHz bandpass filter.

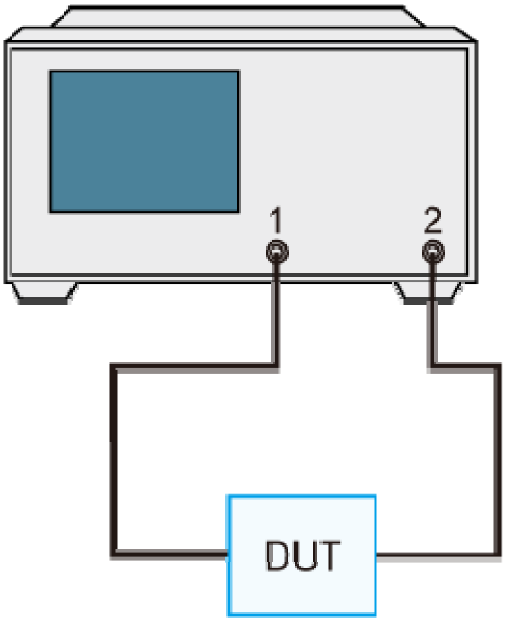
Procedure

Here, the DUT is evaluated by following the steps.

Step	Description
1. Connect the DUT	The DUT is connected.
2. Define the Measurement Conditions	The measurement conditions are defined.
3. Execute the Calibration	The calibration is executed.
4. Connect the DUT and Execute the Auto Scale	The DUT is connected again to execute the auto scale function.
5. Specify the Electrical Delay	The electrical delay is specified.
6. Measure the Deviation from a Linear Phase	The statistics data function (peak-to-peak) is used to determine the deviation from a linear phase.

1. Connect the DUT

Connect the DUT as shown below.



e5071c320

2. Define the Measurement Conditions

The measurement conditions are defined by following the steps described below.

Setup Description	Key Operation
Presetting	**Preset** > **OK**
Center frequency: 1.09 GHz	**Center** > **1** > **.** > **0** > **9** > **G/n**
Frequency span: 20 MHz	**Span** > **2** > **0** > **M/u**
Measurement parameter: S21	**Meas** > **S21**
Data format: Expand Phase	**Format** > **Expand Phase**
Executing the Auto Scale	**Scale** > **Auto Scale**

3. Execute the Calibration

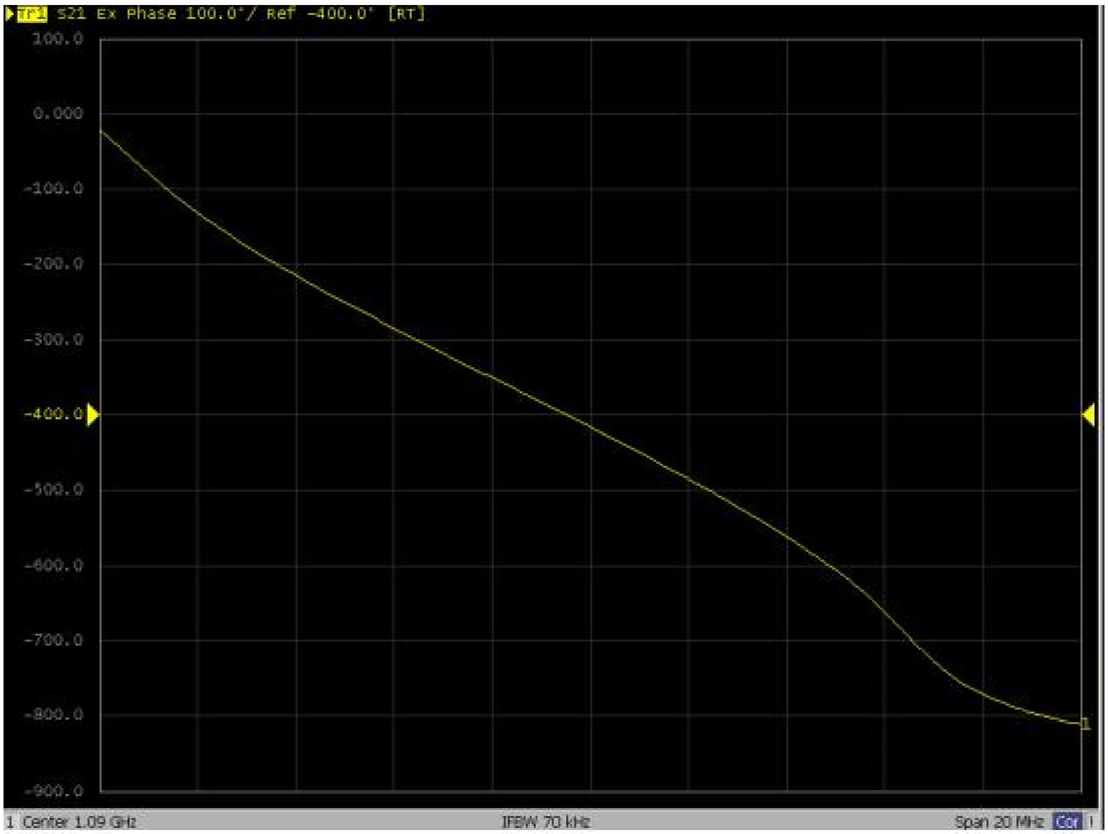
The THRU response calibration is executed.

Setup Description	Key Operation
Executing the THRU response	(A THRU standard is connected instead of a DUT) **Cal** > **Calibrate** > **Response (Thru)** > **Thru** > **Done**

4. Connect the DUT and Execute the Auto Scale

The DUT is connected again as shown in Connect the DUT to execute the auto scale.

Setup Description	Key Operation
Executing the auto scale	**Scale** > **Auto Scale**



e5071c113

5. Specify the Electrical Delay

The electrical delay is entered to flatten the phase trace.

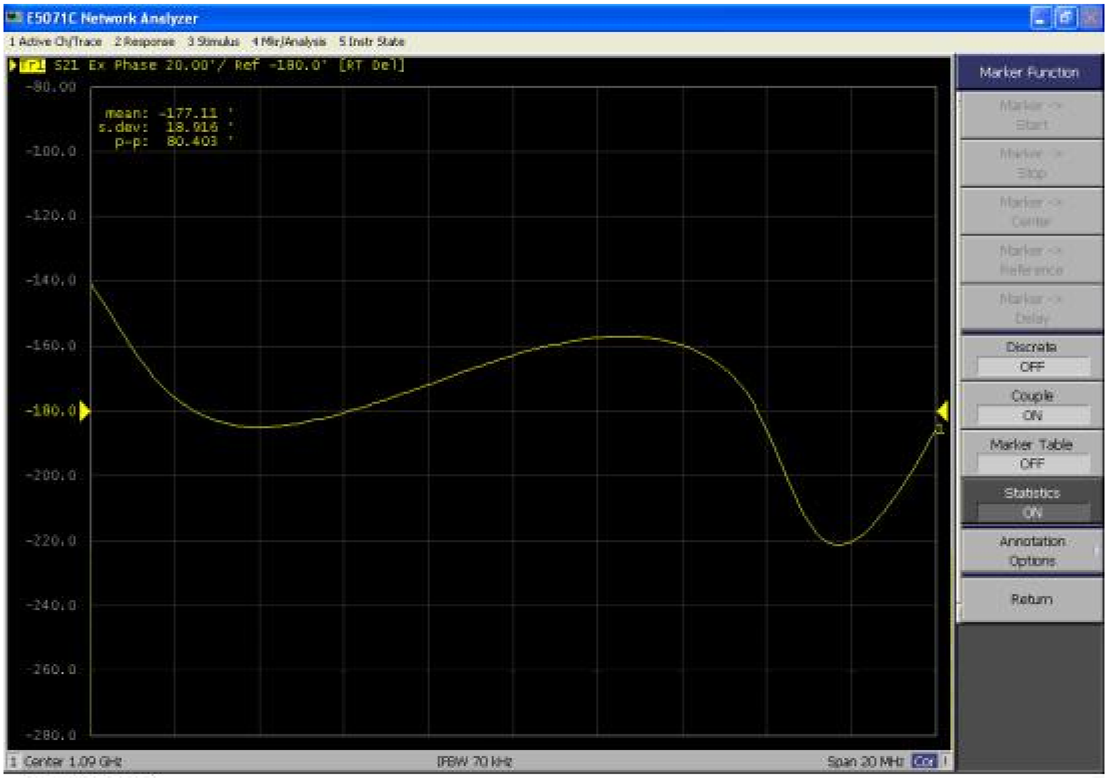
Setup Description	Key Operation
Entering the electrical delay	**Scale** > **Electrical Delay** > **Up**/**Down Arrow** keys or Rotary Knob (Flattening a trace)

6. Measure the Deviation from a Linear Phase

The statistics data is used to read the deviation from a linear phase (peak-to-peak).

Setup Description	Key Operation
Executing the auto scale	**Scale** > **Auto Scale**

Displaying the statistics data	**Marker Function** > **Statistics** (Turn it **ON.**)



e5071c114

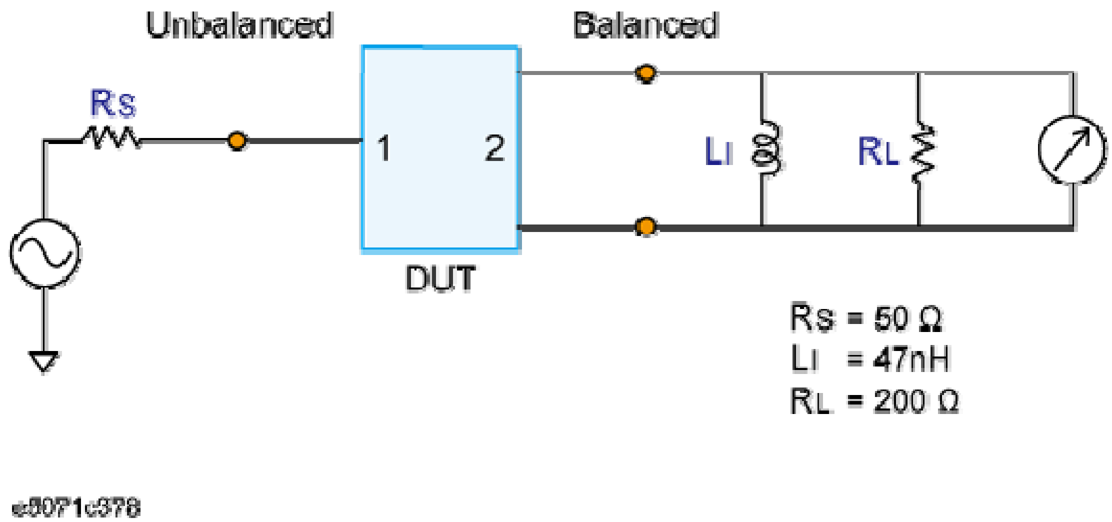
Measuring Unbalanced-Balanced Bandpass Filter

- Overview
- Procedure

Other Measurement Examples

Overview

This section introduces an example of actually evaluating the unbalanced and balanced SAW bandpass filter with a center frequency of 942.5 MHz. The following figure shows the measurement circuit in the condition for evaluating a DUT.



Procedure

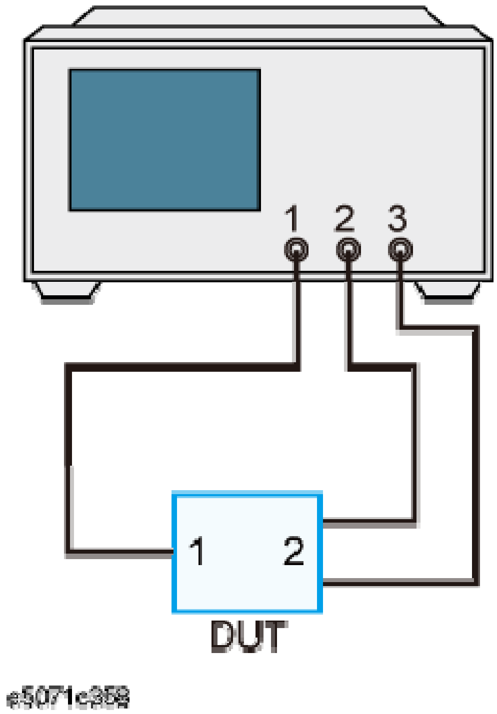
Here, the DUT is evaluated by following the steps.

Step	Description
1. Connecting the DUT	The DUT is connected.
2. Setting the Measurement Conditions	The measurement conditions are defined.
3. Performing Calibration	The full 3-port calibration is executed.
4. Setting a Balance Conversion Topology	The balance conversion topology is specified.
5. Selecting Measurement Parameters	The mixed-mode S-parameters are selected.

6. Extending the Calibration Plane (removing the cause of error)	The calibration reference plane is extended.
7. Setting the Port Reference Impedances	The port reference impedances are specified.
8. Adding a Matching Circuit	A matching circuit is added.

1. Connecting the DUT

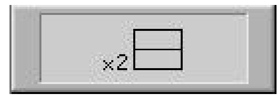
Connect the DUT to the E5071C by using the instrument's three test ports.

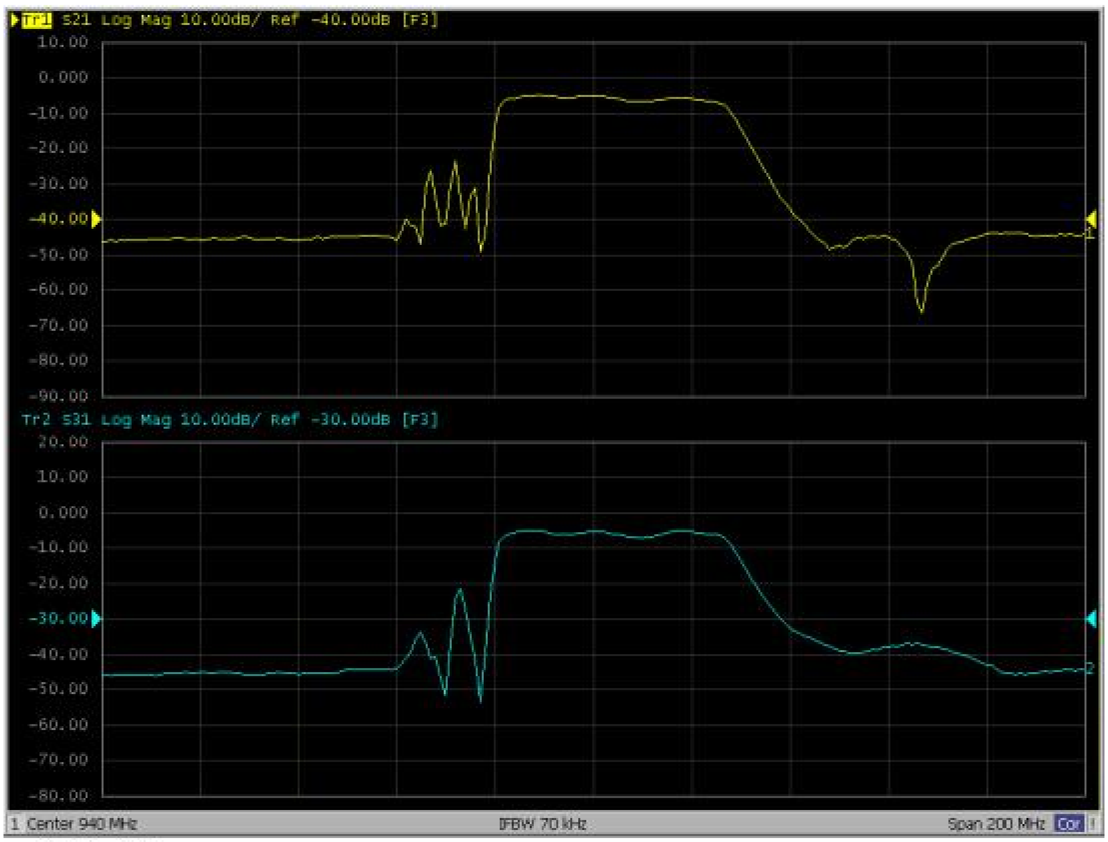


2. Setting the Measurement Conditions

Follow the procedure below to set the measurement conditions. The measurement parameters for balanced measurements should be set after unbalanced-balanced conversion. Here, set the measurement parameters for observing the characteristics achieved during unbalanced measurements.

Setting Description	Key Operation
Preset for setting	**Preset** > **OK**
Center frequency: 940 MHz	**Center** > **9** > **4** > **0** > **M/u**
Frequency span: 200 MHz	**Span** > **2** > **0** > **0** > **M/u**

Number of traces: 2	**Display** > **Num of Traces** > **2**
Trace-1 measurement parameter: S21	**Meas** > **S21**
Trace-2 measurement parameter: S31	**Trace Next** > **Meas** > **S31**
Allocate a trace to upper and lower displays	**Display** > **Allocate Traces** > 
Auto-scale all traces	**Scale** > **Auto Scale All**



e5071c104

3. Performing Calibration

Perform a full three-port calibration for the three ports to be used.

1. Set the type and conditions of calibration.

Setting Description	Key Operation

Calibration kit to use: 85033D	**Cal** > **Cal Kit** > **85033D**
Type of calibration: Full three port calibration	**Calibrate** > **3-Port Cal**
Test ports to calibrate: 1, 2, 3	**Select Ports** > **1-2-3** (check only)

2. Perform a reflection calibration.

Setting Description	Key Operation
Select reflection calibration	**Reflection**
Perform Port 1 calibration	(With the OPEN connected) **Port 1 OPEN** (With the SHORT connected) **Port 1 SHORT** (With the LOAD connected) **Port 1 LOAD**
Perform Port 2 calibration	(With the OPEN connected) **Port 2 OPEN** (With the SHORT connected) **Port 2 SHORT** (With the LOAD connected) **Port 2 LOAD**
Perform Port 3 calibration	(With the OPEN connected) **Port 3 OPEN** (With the SHORT connected) **Port 3 SHORT** (With the LOAD connected) **Port 3 LOAD**

3. Perform a transmission calibration.

Setting Description	Key Operation
Select transmission calibration	**Return** > **Reflection**
Perform a Port 1-to-Port 2 calibration	(With thru connection) **Port 1-2 Thru**
Perform a Port 1-to-Port 3 calibration	(With thru connection) **Port 1-3 Thru**

Perform a Port 2-to-Port 3 calibration	(With thru connection) **Port 2-3 Thru**

4. Finish the calibration.

Setting Description	**Key Operation**
Complete the calibration and then calculate and store calibration coefficients.	**Return** > **Done** (This causes **Correction** to turn **ON.**)
Calibration property display: ON	**Return** > **Return > Property** (Turns it **ON.**)

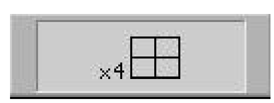
4. Setting a Balance Conversion Topology

Follow the procedure below to set the balanced conversion topology.

Setting Description	**Key Operation**
Set port 1 on the DUT to unbalanced and port 2 on the DUT to balanced.	**Analysis** > **Fixture Simulator > Topology > Device > SE-Bal** (check only)
Set the connecting destination of port 1 on the DUT (unbalanced) to test port 1 of the analyzer.	**Port 1 (se)** > **1** (check only)
Set the connecting destination of port 2 on the DUT (balanced) to test ports 2 and 3 of the analyzer.	**Port 2 (bal)** > **2-3** (check only)

5. Selecting Measurement Parameters

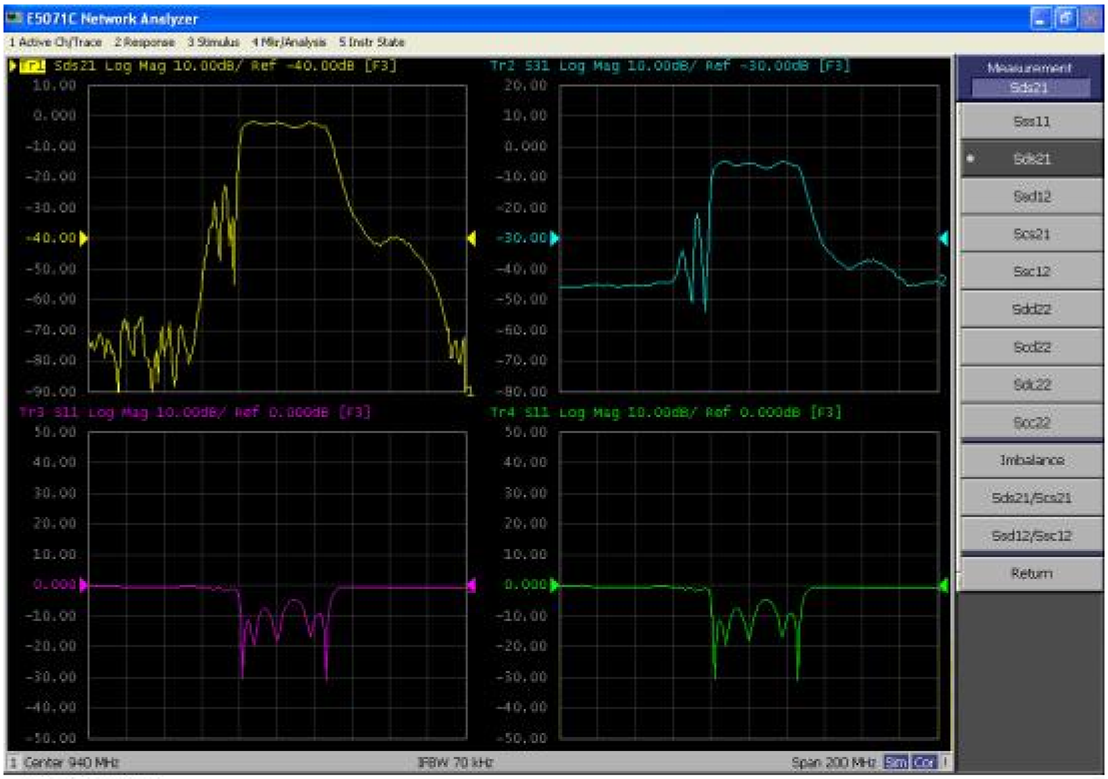
1. Display four traces.

Setting Description	Key Operation
Number of traces: 4	**Display** > **Number of Traces > 4**
Trace allocation: 4-part split	**Allocate Traces** > 

2. Set the measurement parameter (mixed mode S-parameter) and data format for trace 1.

Setting Description	**Key Operation**
Fixture simulator: ON	**Analysis** > **Fixture Simulator** > **Fixture Simulator** (turns it **ON**)

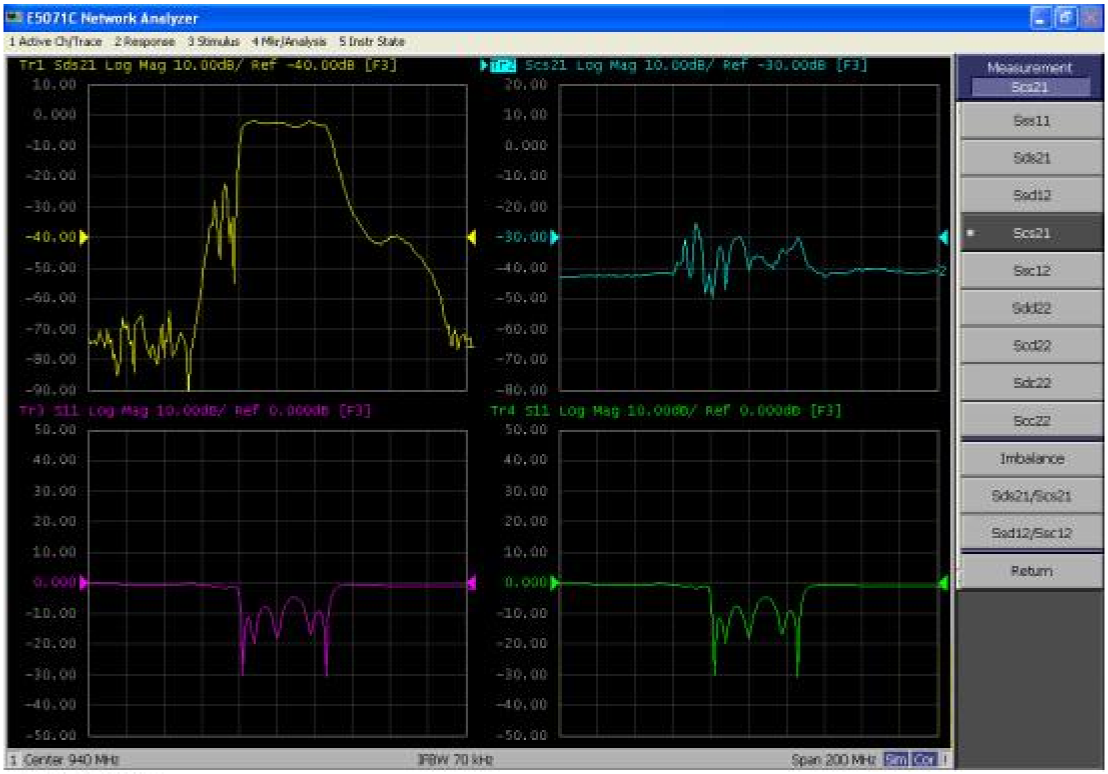
Unbalanced-balanced conversion of trace 1: ON	**BalUn** (turns it **ON**)
Measurement parameter: Sds21	**Meas** > **Sds21**



e5071c105

3. Set the measurement parameter (mixed mode S-parameter) and data format for trace 2.

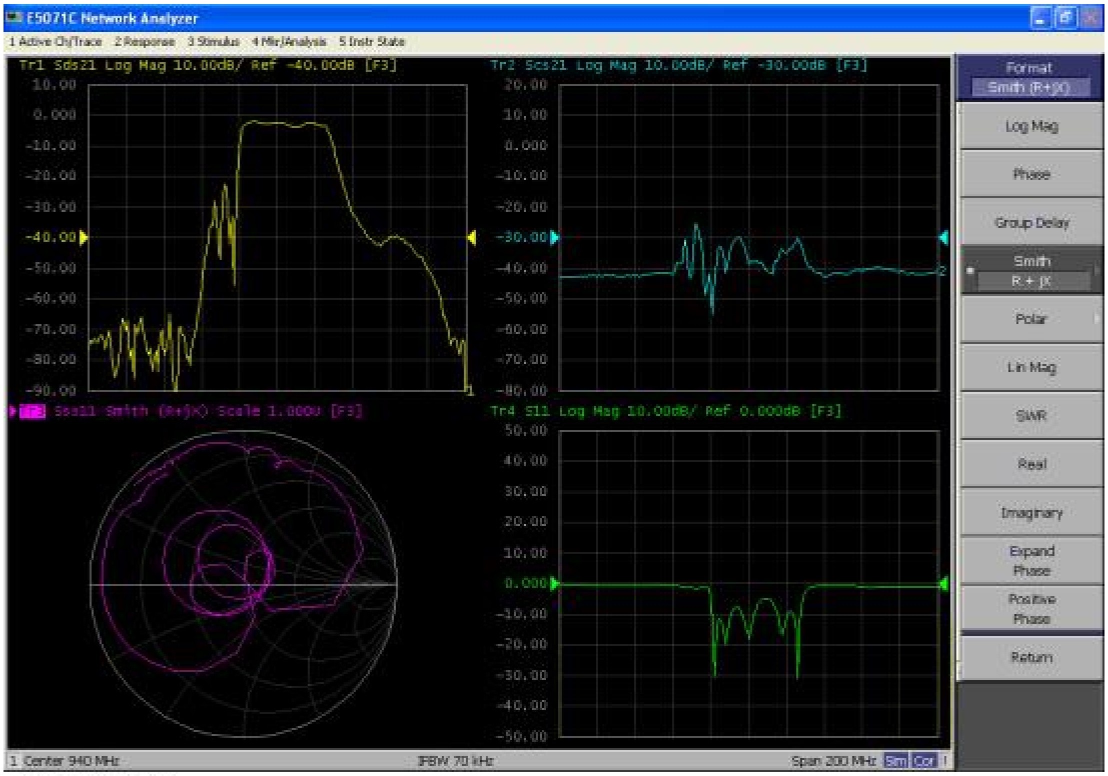
Setting Description	Key Operation
Unbalanced-balanced conversion of trace 2: ON	**Trace next** > **Analysis** > **Fixture Simulator** > **BalUn** (turns it **ON**)
Measurement parameter: Scs21	**Meas** > **Scs21**



e5071c106

4. Set the measurement parameter (mixed mode S-parameter) and data format for trace 3.

Setting Description	Key Operation
Unbalanced-balanced conversion of trace 3: ON	**Trace next** > **Analysis** > **Fixture Simulator** > **BalUn** (turns it **ON**)
Measurement parameter: Sss11	**Meas** > **Sss11**
Data format: Smith chart (marker display: R+jX)	**Format** > **Smith** > **R + jX**

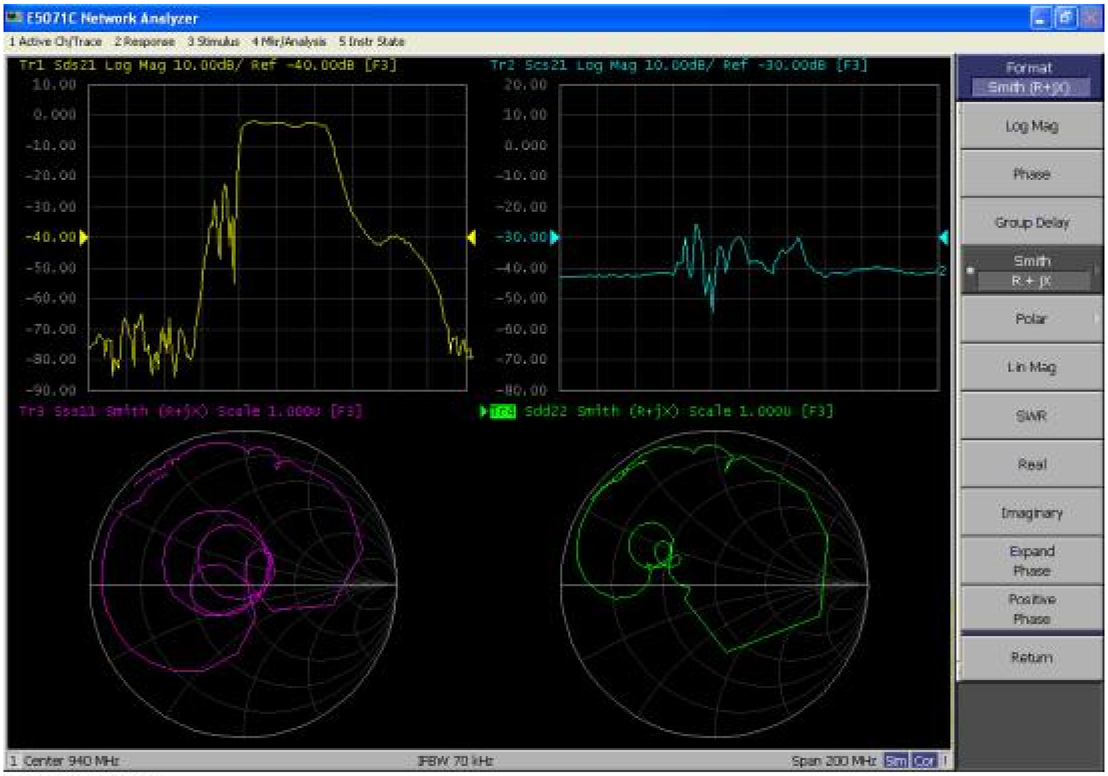


e5071c107

5. Set the measurement parameter (mixed mode S-parameter) and data format for trace 4.

Setting Description	Key Operation
Unbalanced-balanced conversion of trace 4: ON	**Trace next** > **Analysis** > **Fixture Simulator** > **BalUn** (turns it **ON**)
Measurement parameter: Sdd22	**Meas** > **Sdd22**
Data format: Smith chart (marker display: R+jX)	**Format** > **Smith** > **R + jX**

The following figure shows the setting results for each parameter.



e5071c108

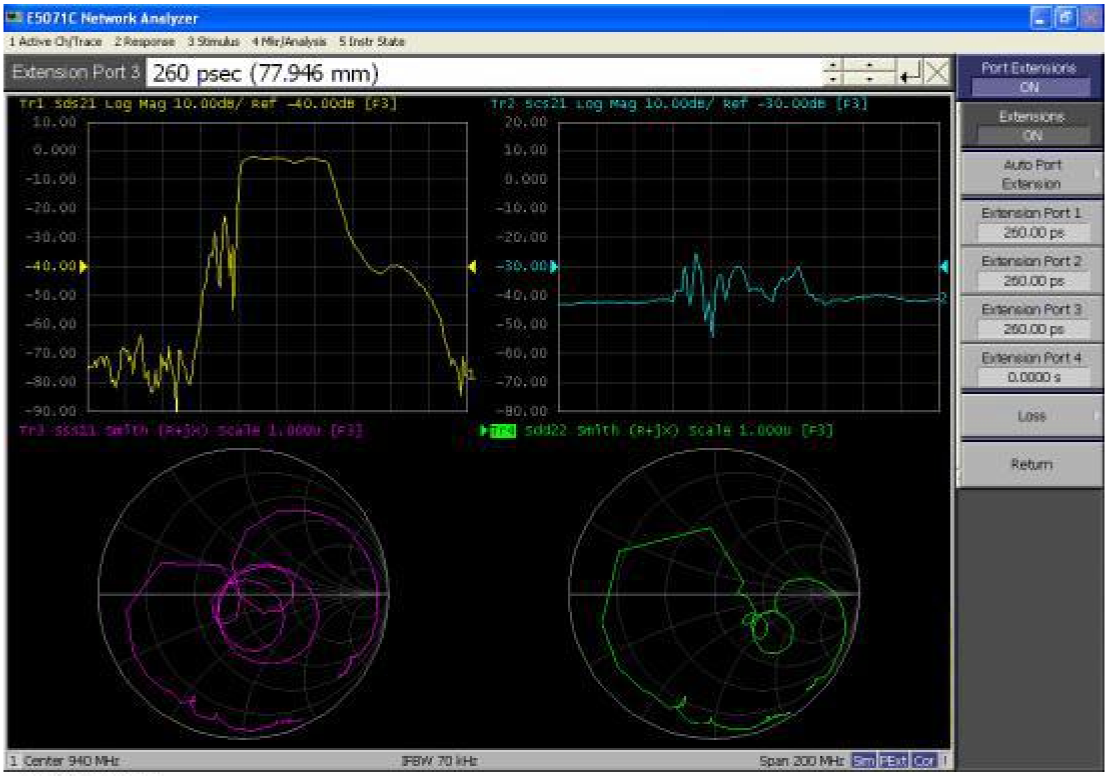
6. Extending the Calibration Plane (removing the cause of error)

In this section you will use the port extension function to remove an electrical delay caused by cables or fixtures located between the calibration reference plane and the DUT to be evaluated. If you can provide a two-port Touchstone data file representing the characteristics of the network to be removed, the network removal function allows you to remove the network and extend the calibration reference plane.

Follow the procedure below to set port extension for each test port.

Setting Description	Key Operation
Port extension of test port 1: 260 ps	**Cal** > **Port Extensions** > **Extension Port 1** > **.** > **2** > **6** > **G/n**
Port extension of test port 2: 260 ps	**Extension Port 2** > **.** > **2** > **6** > **G/n**
Port extension of test port 3: 260 ps	**Extension Port 3** > **.** > **2** > **6** > **G/n**
Port extension: ON	**Extensions** (turns it **ON**)

The following figure shows the results of extending the calibration reference plane.



e5071c109

7. Setting the Port Reference Impedances

With the reference impedances of two test ports in unbalanced measurements set to Z0, conversion of those ports into balanced ports permits the impedance of the balanced ports' common mode to be automatically set to Z0/2 and the impedance of their differential mode to be automatically set to 2Z0.

1. Set the port reference impedance of port 1 on the DUT (unbalanced) to 50 ohm.

Setting Description	Key Operation
Reference impedance of test port 1: 50 ohm	Analysis > **Fixture Simulator** > **Port Z conversion** > **Port 1 Z0 Real** > 5 > 0 > x1

2. In order to set the impedance of the differential mode of port 2 on the DUT (balanced) to 200 ohm , set the impedances of two unbalanced ports before conversion each to 100 ohm.

Setting Description	Key Operation
Reference impedance of test port 2: 100 ohm	**Port 2 Z0 Real** > 1 > 0 > 0 > x1

Reference impedance of test port 3: 100 ohm	**Port 3 Z0 Real** > **1** > **0** > **0** > **x1**

3. Always set the reference impedances of the two test ports before balanced conversion to the same value.
4. Turn on the port reference impedance conversion function.

Setting Description	Key Operation
Port reference impedance conversion: ON	**Port Z Conversion** (turns it **ON**)

The reference impedance of the command mode of port 2 on the DUT is set to 50 W . The impedance of the differential mode of that port may be set and modified independently of setting the two-port reference impedances before balanced conversion. For more information, see Converting Reference Impedance of Balanced Port.

8. Adding a Matching Circuit

Here, add an inductance of 47 nH in parallel to port 2 on the DUT (balanced). It is also possible to add a matching circuit to the port before unbalanced-balanced conversion. For more information, see Determining the Characteristics that Result from Adding a Matching Circuit to a Differential Port.

NOTE The balance matching circuit (Differential matching circuit embedding) is not applied to single-ended S-parameter results. For example, the balance matching circuit is not applied to the imbalance parameter as it is derived from single-ended S-parameters.

Setting Description	Key Operation
Selecting a matching circuit: Shunt L - Shunt C	**Return** (or **Analysis** > **Fixture Simulator**) > **Diff. Matching** > **Select Circuit** > **Shunt L-Shunt C**

	 e5071c110
Inductance: 47 nH	**L** > **4** > **7** > **G/n**
C=0, G=0, R=0	(checks that **C**, **G**, and **R** have been set to **0.**)
Differential matching circuit function: ON	**Diff. Matching** (turns it **ON**)

Measuring Parameters with Cable

- Overview
- Procedure

Other Measurement Examples

Overview

This section introduces an example of how to detect the location of a mismatch that occurs in a cable by using the time domain function.

Procedure

In this example, a DUT is evaluated according to the steps.

Step	Description
1. Setting the Measurement Conditions	Set the measurement conditions.
2. Executing Calibration	Execute calibration.
3. Connecting the DUT	Connect the DUT.
4. Auto Scale	Execute auto scale.
5. Setting the Time Domain Function	Set the time domain function.

1. Setting the Measurement Conditions

Follow these steps to set the measurement conditions:

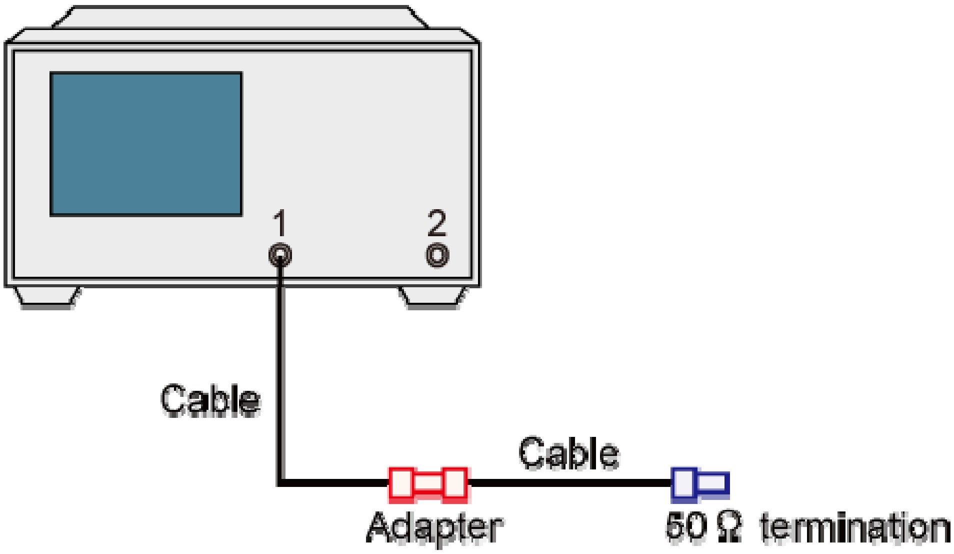
Setting Description	Key Operation
Presetting	**Preset** > **OK**
Stop frequency: 3 GHz	**Stop** > **3** > **G/n**
Number of points: 201	**Sweep Setup** > **Points** > **2** > **0** > **1** > **x1**
Specifying the low-pass mode sweep condition	**Analysis** > **Transform** > **Set Freq Low Pass**
Measurement parameter: S11	**Meas** > **S11**

2. Executing Calibration

By following 1-Port Calibration (reflection test), execute 1-port calibration on port 1.

3. Connecting the DUT

Connect the DUT as shown below.

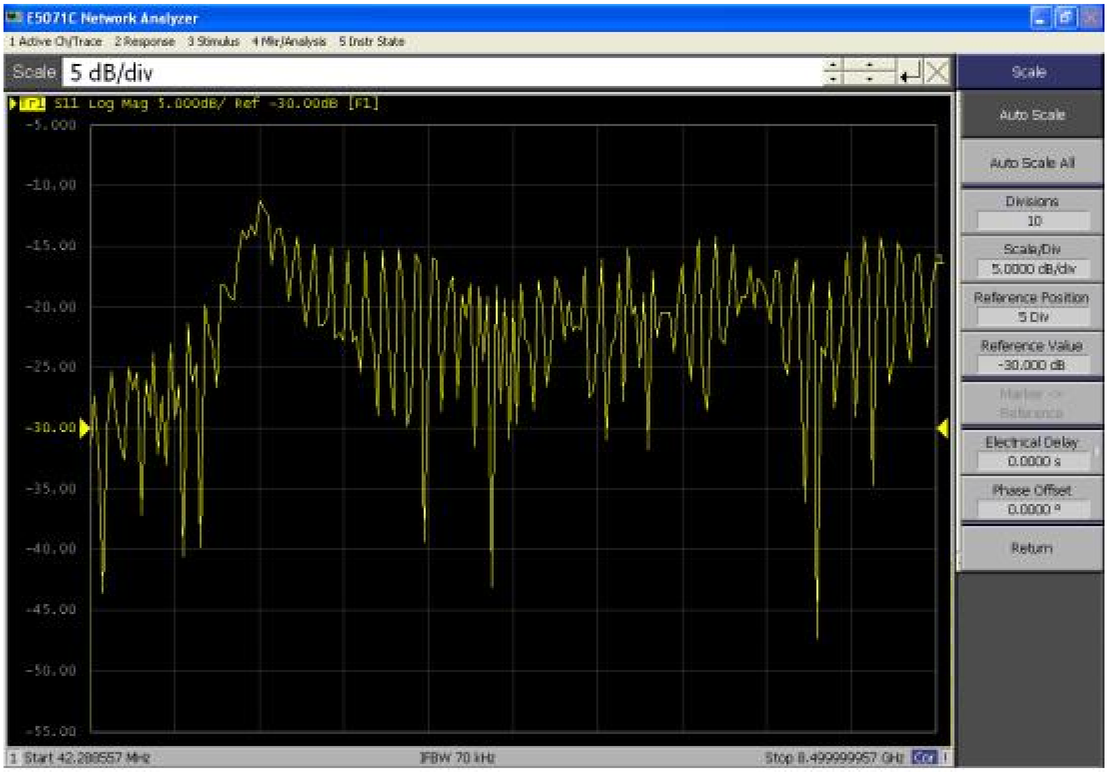


e5071c327

4. Auto Scale

Execute the auto scale function.

Setting Description	Key Operation
Executing auto scale	Scale > **Auto Scale**



e5071c111

5. Setting the Time Domain Function

Set the conversion function to display the response in the time domain. If you enable this setting, the response in time domain is displayed as shown figure below. A peak indicating a small mismatch appears at the location of the connector.

Setting Description	Key Operation
Data format: real	**Format** > **Real**
Setting the transformation type to low-pass impulse	**Analysis** > **Transform** > **Type** > **Lowpass Imp**
Setting the window type to maximum.	**Window** > **Maximum**
Setting the display range: from 0 s to 20 ns	**Start** > **0** > **x1**
	Stop > **2** > **0** > **G/n**
Enabling the transformation function	**Transform** (set to **ON**)
Executing auto scale	**Scale** > **Auto Scale**


E5071C Network Analyzer
1 Active Ch/Trace 2 Response 3 Stimulus 4 Mkr/Analysis 5 Instr State
Stop 20 nsec (5.9958 m)
Tr1 S11 Real 10.00mU/ Ref 0.000U [F1]
Transform
Transform ON
Start 0.0000 s
Stop 20.000 ns
Center 10.000 ns
Span 20.000 ns
Type Lowpass Imp
Window
Return
1 Start 0 s
IFBW 70 kHz
Stop 20 ns

e5071c112

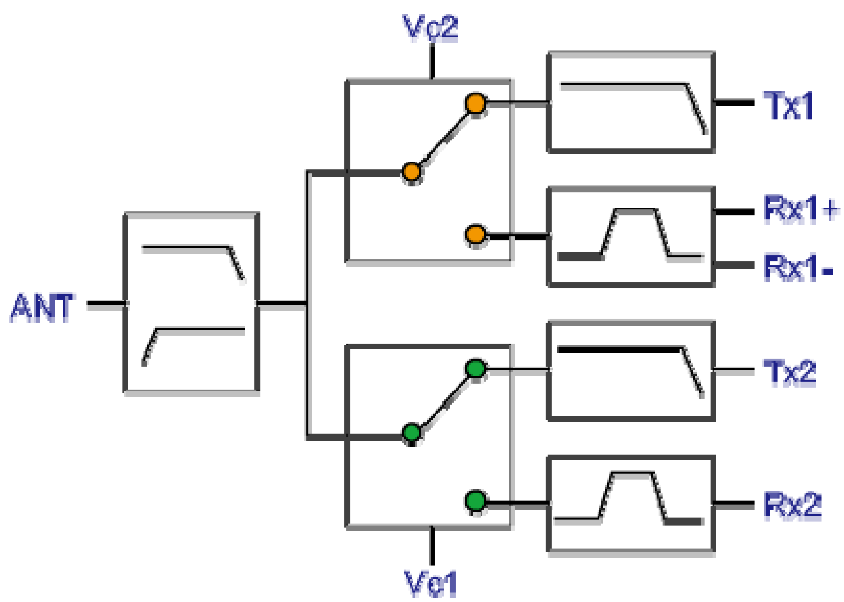
Evaluating Transmission Characteristics of a Front End Module

- Overview
- Procedure

Other Measurement Examples

Overview

This example shows how to measure the transmission characteristics of a 6-port front end module, as shown in the following figure, by using the E5071C and the E5091A.



e5071c472

Procedure

In this example, a DUT is evaluated according to the procedure shown in table below.

Procedure	Description
1. Determining Measurement Conditions	Determine the measurement conditions such as the sweep conditions and measurement ports.
2. Setting Channel Window Allocation	Set the allocation of the channel windows on the screen.
3. Setting the Test Ports	Determine the test port assignment for each channel.

4. Setting Control Line	Set the E5091A's control line.
5. Setting Sweep Conditions	Set the sweep range and the number of points.
6. Setting Balance Conversion Topology	Set the balance port and unbalance port assignment.
7. Selecting Measurement Parameter	Set the measurement parameter.
8. Executing Calibration	Perform calibration using 4-port ECal.
9. Connecting DUT	Connect the DUT.
10. Executing Measurement	Execute the measurement and perform auto scale.

1. Determining Measurement Conditions

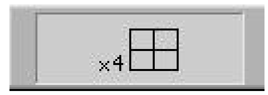
In this example, perform measurement under the measurement conditions in table below.

Ch.	Start frequency	Stop frequency	NOP	Test port assignment	Control line	Meas. parameter	Calibration	
							Type	Port
1	400 MHz	1.4 GHz	201	Port 1 - Port A Port 2 - Port T1 Port 3 - Port R1+ Port 4 - Port R1-	Line 1: Low Line 2: High	S12	Full 2-Port	1,2
2	880 MHz	1 GHz	101		Line 1: Low Line 2: Low	Sds21	Full 3-Port	1,3,4
3	1.34 GHz	2.34 GHz	201	Port 1 - Port A Port 2 - Port T2 Port 3 - Port R2+ Port 4 - Port R1-	Line 1: High Line 2: Low	S12	Full 2-Port	1,2
4	1.665 GHz	2.015 GHz	101		Line 1:	S31	Full 2-	1,3

					Low Line 2: Low		Por t	

2. Setting Channel Window Allocation

Set the screen to split into 2 rows and 2 columns to assign channel windows after preset.

Setting Description	Key Operation
Execute preset	**Preset > OK**
Allocate channel windows	**Display** > **Allocate Channels** > 

3. Setting the Test Ports

1. Display the E5091A setup menu and select the 9-port model for ID1.

Setting Description	Key Operation
Display the E5091A setup menu	**System** > **Multiport Test Set Setup**
Select the 9-port model for ID1	**Test Set 1 > Select Test Set > E5091_9**

2. Display the E5091A properties.

Setting Description	Key Operation
Display the E5091A properties	**Property**

3. Select the test ports assigned to ports 1 to 4 for channel 1.

Setting Description	Key Operation
Assign test port A to port 1.	**Port1** > **A**
Assign test port T1 to port 2.	**Port2** > **T1**
Assign test port R1+ to port 3.	**Port3** > **R1+**
Assign test port R1- to port 4.	**Port4** > **R1-**

4. Assign test ports for channels 2, 3, and 4. Press the **Channel Next** key to switch the active channel and then make the same setting of channel 1.

5. Enable the control of the E5091A.

Setting Description	Key Operation
Enable the control of the E5091A.	**Control** (set it to **ON**)

4. Setting Control Line

1. Set the bits of the control line for channel 1.

Setting Description	Key Operation
Set channel 1 to the active channel.	**Channel Next**
Line 1: Low	**Control Lines** (check only)
Line 2: High	**Control Lines** > **Line 1** (set it to **High**)

2. Set the control line for channels 2, 3, and 4 according to the same procedure of channel 1.

5. Setting Sweep Conditions

1. Set the sweep conditions for channel 1.

Setting Description	Key Operation
Set channel 1 to the active channel.	**Channel Next**
Start frequency: 400 MHz	**Start** > **4** > **0** > **0** > **M/u**
Stop frequency: 1.4 GHz	**Stop** > **1** > **.** > **4** > **G/n**
Number of points: 201	**Sweep Setup** > **Point** > **2** > **0** > **1** > **x1**

2. Set sweep conditions for channels 2, 3, and 4 according to the same setting of channel 1.

6. Setting Balance Conversion Topology

For channel 2, set the balance conversion topology in order to perform measurement that includes the balanced port.

Setting Description	Key Operation
Set channel 2 to the active channel.	**Channel Next** (press it until channel 2 is activated)
Set DUT port 1 to unbalance and DUT port 2 to balance.	**Analysis** > **Fixture Simulator > Topology > Device > SE-Bal**
Set the destination to which DUT port 1 (unbalance) is connected to test port 1 of the analyzer.	**Port 1 (se)** > **1**
Set the destination to which DUT port 2 (balance) is connected to test ports 3 and 4 of the analyzer.	**Port 2 (bal)** > **3-4**
Unbalance-balance conversion for trace 1: ON	**BalUn** (set it to **ON**)
Fixture simulator: ON	**Fixture Simulator** (set it to **ON**)

7. Selecting Measurement Parameter

1. Select the measurement parameter for trace 1 of channel 1.

Setting Description	Key Operation
Set channel 1 to the active channel.	**Channel Next** (press it until channel 1 is activated)
Measurement parameter for trace 1: S12	**Meas** > **S12**

NOTE The subscript of the measurement parameter means the test port of the E5071C. Check the test port assignment and select the measurement parameter.

2. Set measurement parameter for channels 2, 3, and 4 according to the same setting of channel 1.

8. Executing Calibration

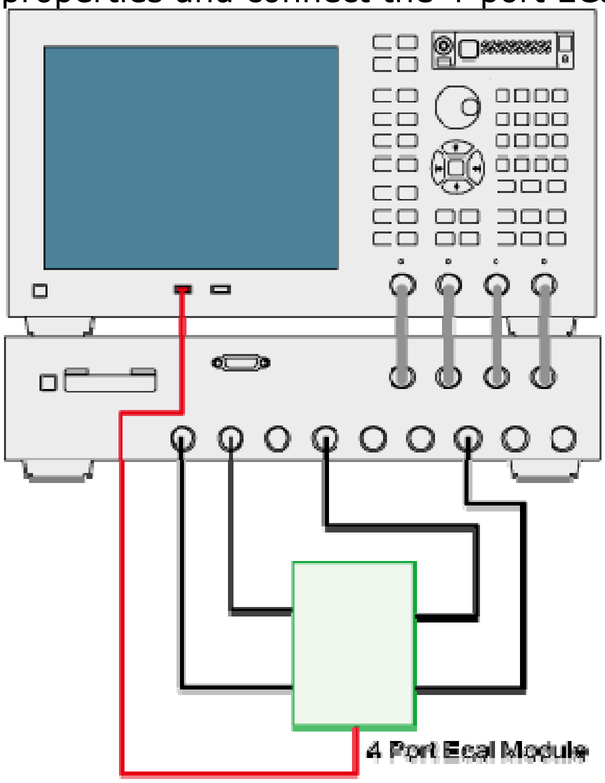
1. Display the Ecal menu.

Setting Description	Key Operation
Display the ECal menu.	**Cal** > **ECal**

2. Set channel 1 to the active channel.

Setting Description	Key Operation
Switch the active channel.	**Channel Next**

3. Check the test ports assigned to ports 1 to 4 in the E5091A properties and connect the 4-port ECal module to those port.



e5071c413

4. Execute the calibration.

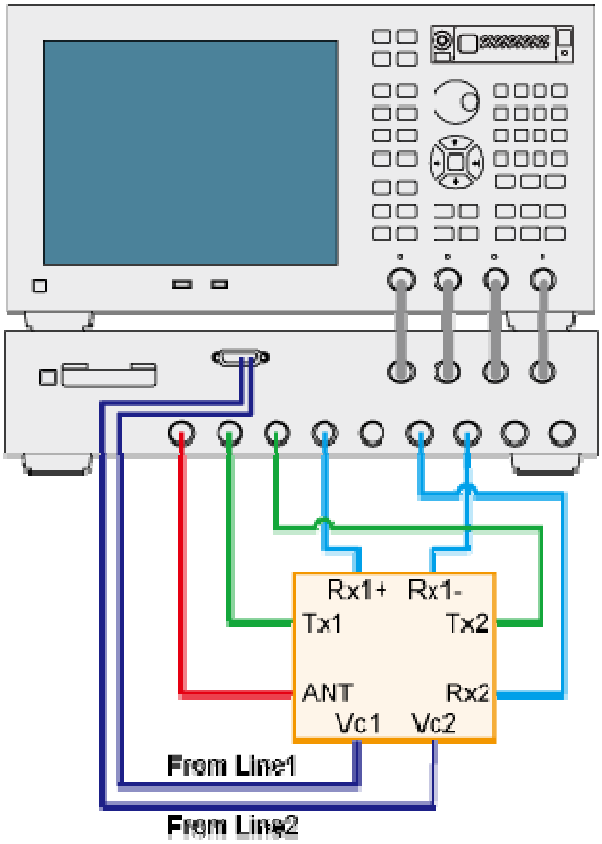
Setting Description	Key Operation
Select full 2-port calibration.	**2-Port ECal**
Select the port and execute the calibration.	**1-2**

5. Perform calibration for channels 2, 3, and 4 according to the same procedure as in Step 2 to Step 4.

NOTE Because the test port assignment setting for channels 1 and 2 and that for channels 3 and 4 are the same, you need not change the ECal connection.

9. Connecting DUT

Connect the DUT as shown below.



e5071c412

10. Executing Measurement

1. Display the trigger menu.

Description	Key Operation
Display the trigger menu.	**Trigger**

2. Set the trigger source to "manual."

Setting Description	Key Operation
Set the trigger source to "manual."	**Trigger Source** > **Manual**

3. Set the trigger mode for channel 1 to "continuous."

Setting Description	Key Operation
Set channel 1 to the active channel.	**Channel Next**

Set the trigger mode to "continuous."	**Continuous**

4. Set the trigger mode for channels 2, 3, and 4 to "continuous" according to the same procedure as in Step 3.
5. Execute the measurement.

Setting Description	Key Operation
Generate a trigger event.	**Trigger**

6. Repeat the following procedure to execute auto scale for all of the channels.

Setting Description	Key Operation
Set the active channel.	**Channel Next**
Execute auto scale.	**Scale** > **Auto Scale**

Executing Power Calibration

- Overview
- Procedure

Other Measurement Examples

Overview

This section shows an example of executing power calibration using the E4418B power meter and the E4412A power sensor.

NOTE Power calibration may not be executed when the specified frequency and level is out of range of the power sensor specification.

Procedure

In this example, power calibration is executed according to the steps shown in table below.

Step	Description
1. Connecting Power Meter	Connects the power meter to the E5071C.
2. Selecting Power Meter a. Setting Address of GPIB Power Meter b. Setting USB Power Meter	Configures the power meter's USB/GPIB interface and GPIB address with the E5071C.
3. Setting Stimulus Condition	Sets conditions such as the power level and frequency.
4. Executing Zero Adjustment and Calibration of Power Sensor	Executes zero adjustment and calibration of the power sensor.
5. Setting Calibration Data Measurement Conditions	Selects the port, selects the power sensor, sets the number of measurements at one point, and sets the tolerance during power calibration.
6. Connecting Power Sensor	Connects the power sensor.
7. Measuring Calibration Data	Executes the measurement of calibration data.

1. Connecting Power Meter

Connect the GPIB connector on the E4418B (GPIB address: 14) to the E5071C through USB/GPIB Interface.

NOTE From Firmware revision 9.2, E5071C supports USB enabled power sensors. When you connect USB enabled power sensor for the first time, new hardware wizard runs. For more information, refer to Setting system controller (USB/GPIB interface)

2. Selecting Power Meter

Follow the steps to select the power meter

Setting Description	Key Operation
USB/GPIB	**System** > **Misc Setup** > **Power Meter Setup** > **Select Type** > **USB/GPIB**

Setting Address of GPIB Power Meter

Follow these steps to configure the power meter's GPIB address.

Setting Description	Key Operation
GPIB address of the power meter: 14	**System** > **Misc Setup** > **GPIB Setup** > **Power Meter Address** > **1** > **4** > **x1**

NOTE You can also set the GPIB address of power meter by pressing **System** > **Misc Setup** > **Power Meter Setup** > **GPIB Address**

Setting USB Power Meter

Follow these steps to configure the USB power meter

Setting Description	Key Operation
USB power meter	**System** > **Misc Setup** > **Power Meter Setup** > **USB**

3. Setting Stimulus Condition

Follow these steps to configure the stimulus conditions:

Setting Description	Key Operation
Presetting	**Preset** > **OK**
Sweep type: Power	**Sweep Setup** > **Sweep Type** > **Power Sweep**
Fixed frequency: 1 GHz	**Sweep Setup** > **Power** > **CW Freq** > **1** > **G/n**
Start value: -40 dBm	**Start** > **+/-** > **4** > **0** > **x1**
Stop value: -10 dBm	**Stop** > **+/-** > **1** > **0** > **x1**
Number of points: 61	**Sweep Setup** > **Points** > **6** > **1** > **x1**

4. Executing Zero Adjustment and Calibration of Power Sensor

Execute the zero adjustment and calibration of the power sensor according to the *E4418B Power Meter User's Guide*.

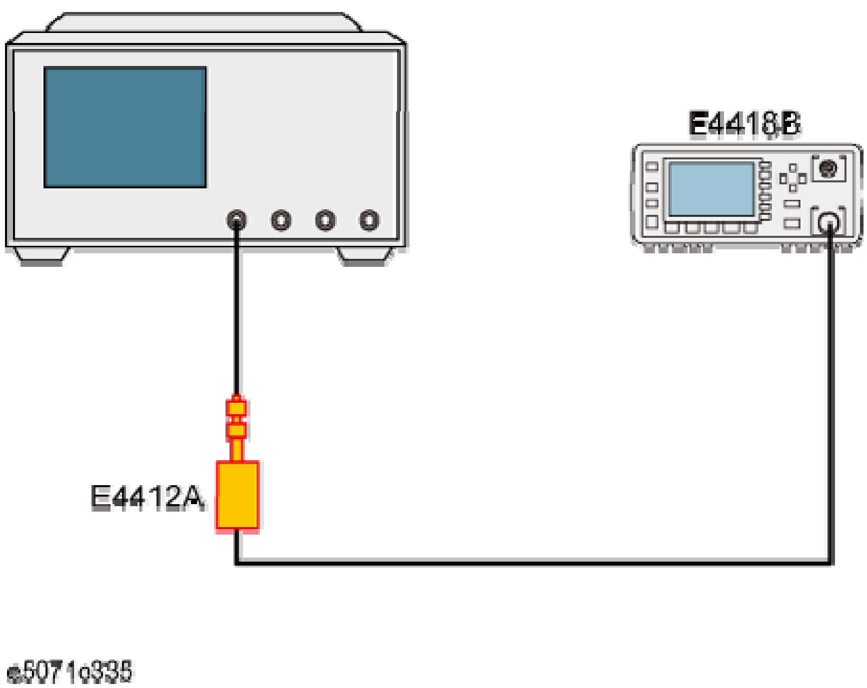
5. Setting Calibration Data Measurement Conditions

Follow these steps to set the calibration data measurement conditions:

Setting Description	Key Operation
Selecting the test port: 1	**Cal** > **Power Calibration** > **Select Port** > **1**
Selection of power sensor: A	**Cal** > **Power Calibration** > **Use Sensor [A]**
Number of measurements at one measurement point: 4	**Cal** > **Power Calibration** > **Num of Readings** > **4** > **x1**
Tolerance during power calibration: 5 dB	**Cal** > **Power Calibration** > **Tolerance** > **5** > **x1**

6. Connecting Power Sensor

Connect the power sensor as shown below.



7. Measuring Calibration Data

Follow this step to measure the calibration data:

Setting Description	Key Operation
Measuring calibration data	**Cal** > **Power Calibration** > **Take Cal Sweep**

Fixture Simulator

Fixture Simulator

- Overview of Fixture Simulator
- Extending the Calibration Plane using Network De-embedding
- Converting the Port Impedance of the Measurement Result
- Determining Characteristics After Adding a Matching Circuit
- Obtaining Characteristics After Embedding/De-embedding 4/6/8-port Network
- Evaluating Balanced Devices (balance-unbalance conversion function)
- Converting Reference Impedance of Balanced Port
- Determining the Characteristics that Result from Adding a Matching Circuit to a Differential Port
- Example of Using Fixture Simulator

Overview of Fixture Simulator

- Overview
- Functions for Single-Ended (Unbalanced) Port

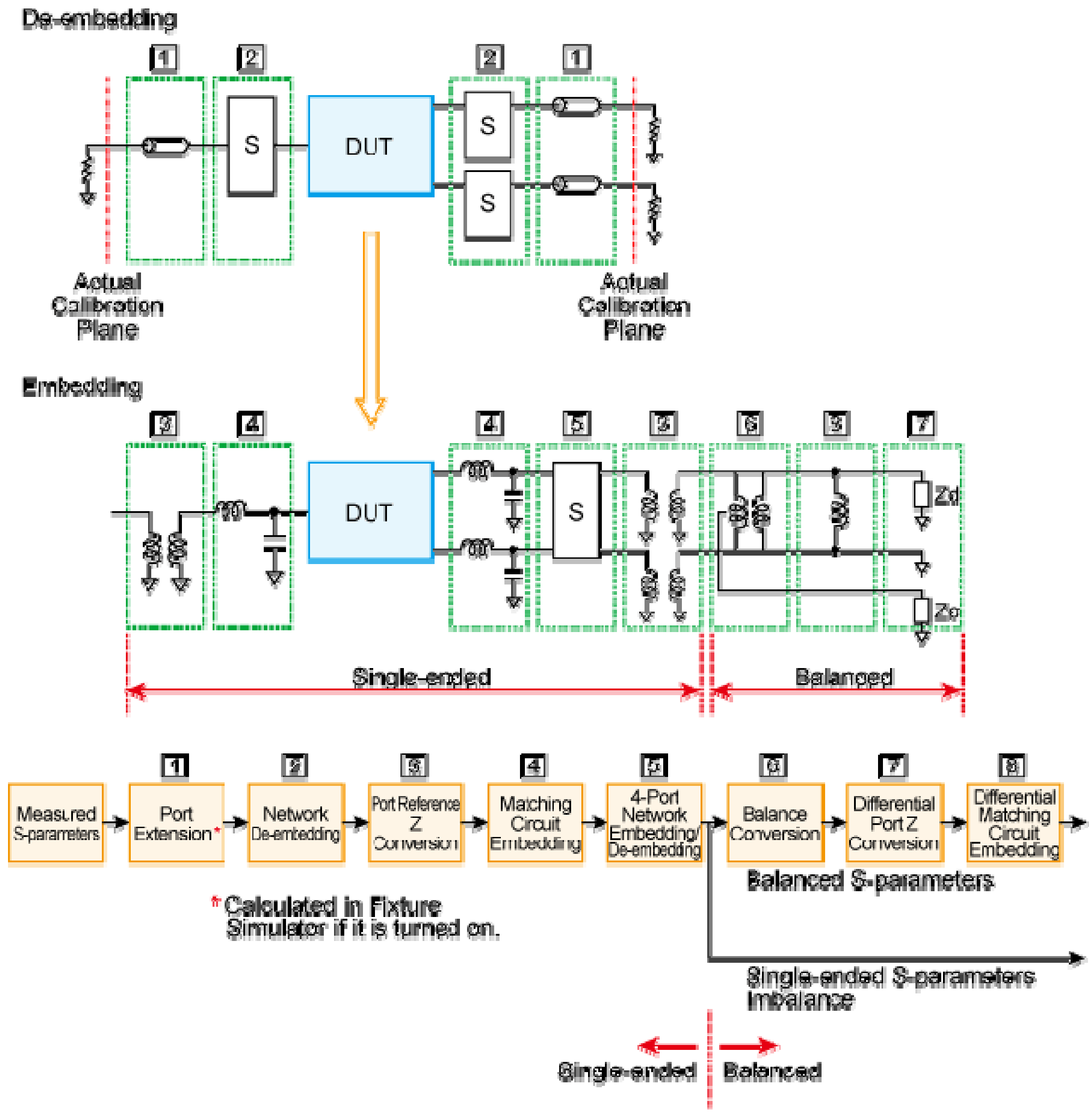
Other topics about Fixture Simulator

Overview

The Fixture Simulator is a function that uses software in the E5071C to simulate various measurement conditions based on the measurement results. The functions available in Fixture Simulator are as follows:

- Network de-embedding
- Port reference impedance conversion
- Matching circuit embedding
- 4-port network embedding/de-embedding
- Balance-unbalance conversion
- Differential/Common port reference impedance conversion
- Differential matching circuit embedding

Data processing flow diagram of fixture simulator



e5071c545

NOTE The E5071C adopts the power wave definition for its S-parameter calculation. Reference document: "Power waves and Scattering Matrix. K. Kurokawa"

Port extension is an independent function from the fixture simulator, but if the fixture simulator function is on, data processing is automatically executed as a function of the fixture simulator to improve the data processing efficiency. (Measurement result is the same as when the fixture simulator is turned off.) Port extension moves the calibration reference location by setting an electrical delay for a single-ended port. Port extension can eliminate only electrical delay (phase shift) for each single-ended port. Loss or mismatch cannot be eliminated by this function.

Functions for Single-Ended (Unbalanced) Port

The following three functions are applied to single-ended ports (unbalanced ports). Balance-unbalance conversion can additionally be applied to single-ended ports.

Network de-embedding

A function that uses software to remove an arbitrary network (50 ohm system) defined by a two-port Touchstone data file from each test port (single-ended) and to extend the calibration plane. This makes it possible to remove networks that create error elements between the calibration plane and the DUT, thereby enabling a more realistic evaluation of the DUT.

For the setup procedure of the network de-embedding function, see Extending the Calibration Plane Using Network De-embedding.

Port reference impedance conversion

A function that uses software to convert an S-parameter measured with a 50 ohm port reference impedance into a value measured with an arbitrary impedance.

For the setup procedure of port reference impedance conversion, see Converting the Port Impedance of the Measurement Result.

Matching circuit embedding

A function for converting an original measurement result into a characteristic determined under the condition of inserting a matching circuit between the DUT and the test port (single-ended). The matching circuit to be inserted is either selected from the five predetermined circuit models or provided by a designated arbitrary circuit defined in a two-port Touchstone file.

For the setup procedure used for matching circuit embedding, see Determining Characteristics After Adding a Matching Circuit.

4-port network embedding/de-embedding

This is a feature to embed (in terms of numerical calculation) your desired network that you have defined in a 4-port Touchstone data file into measurement results or to de-embed it from them.

When you use this as de-embedding, the embedded matching circuit and a part of networks in the DUT are removed.

For information on how to operate this function, refer to Obtaining Characteristics After Embedding/De-embedding 4-port Network.

NOTE When the 4-port network embedding/de-embedding feature reads a 4-port Touchstone data file, it does not automatically convert the file's normalized impedance value to adapt to the port reference impedance setting value of the analyzer.

Balance-unbalance conversion (4 ports option only)

A function that uses software to convert the measurement results in an unbalanced DUT state, which are obtained by connecting the DUT to the test port of the E5071C, into measurement results in a balanced state. Two test ports of the E5071C are connected to one balanced port of the DUT.

For the setup procedure used for balance-unbalance conversion, see Evaluating Balanced Devices (balance-unbalance conversion function).

Functions for balanced port (4 ports option only)

The following two functions are applied to a balanced (differential) port converted by balance-unbalance conversion.

Differential/Common port impedance conversion (4 ports option only)

A function for converting the differential mode port impedance of a balanced port after an balance-unbalance conversion. Balance-unbalance conversion automatically converts the differential mode port impedance at the balanced port into 2Z0 and the common mode port impedance into Z0/2, compared with the two pre-conversion port impedances of Z0. Differential port impedance conversion further converts a differential port impedance after balance-unbalance conversion into an arbitrary port impedance.

For the setup procedure used for differential port impedance conversion, see Converting Reference Impedance of Balanced Port.

Differential matching circuit embedding (4 ports option only)

A function for converting the measurement results obtained from balance-unbalance conversion into a characteristic under the condition of inserting a matching circuit in the balanced port.

For setup procedure of differential matching circuit embedding, see Determining the Characteristics that Result from Adding a Matching Circuit to a Differential Port.

Extending the Calibration Plane Using Network De-embedding

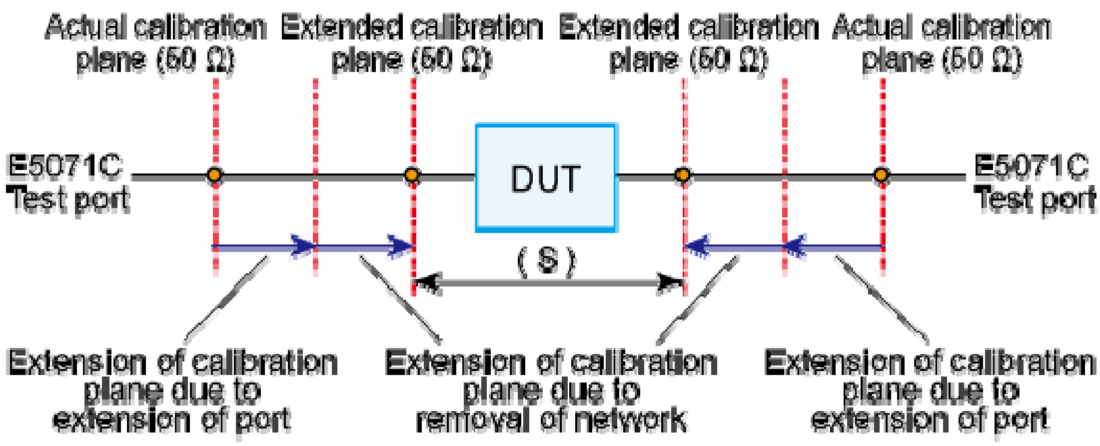
- Overview
- Using Network De-embedding Function

Other topics about Fixture Simulator

Overview

Network de-embedding is a function for performing measurements, test port by test port, by removing the characteristics of an arbitrary network defined by a Touchstone data file. By removing the characteristics of the cable, test fixtures, etc. between the actual calibration plane and the DUT, the calibration plane can be correspondingly extended. The network de-embedding function can be used together with the port extension function.

Port extension and calibration plane extension using network de-embedding



e5071c454

Using Network De-embedding Function

1. Prepare a two-port Touchstone data file (.s2p format) corresponding to the network to be removed.

   NOTE A Touchstone data file is defined for a single normalized impedance value (real). Thus, when preparing a Touchstone data file with the 2-port network de-embedding feature, be sure to specify the same port reference impedance value (real) to both 2-port network in the file.

   NOTE For a network defined in the user file, it is assumed that port 1 is connected to the test port and port 2 is connected to the DUT.

2. Press **Analysis** key, then click **Fixture Simulator** > **De-Embedding**.
3. Click **Select Port.**
4. Click **1**, **2**, **3**, or **4** to select the test port from which the network de-embedding is performed.
5. Click **User File**.
6. Using the dialog box that appears, select the Touchstone data file defining the characteristics of the network to be removed. Once the file is selected, the selection of **Select Type** automatically changes to **User**. To cancel a user-defined file that has been set up, click **Select Type** > **None**.
7. Repeat the procedure to set up the Touchstone data file for each port from which a network is to be removed.
8. Click **De-Embedding** to turn the network de-embedding function **ON**.
9. Click **Return**.
10. If **Fixture Simulator** is **OFF**, press the key again to turn it **ON**.

Converting the Port Impedance of the Measurement Result

- Overview
- Converting Port Impedance

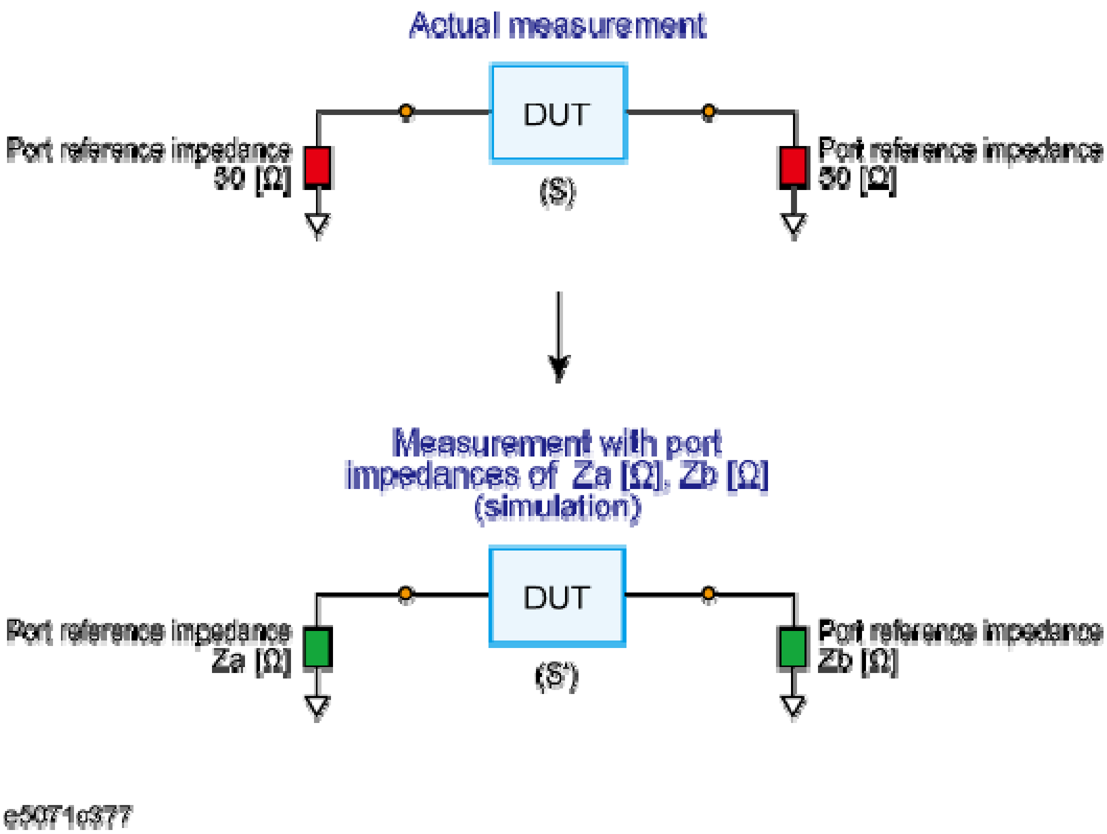
Other topics about Fixture Simulator

Overview

The measured value obtained by using a port impedance of 50 ohm can be converted into a measured value at an arbitrary port impedance.

NOTE When using balance conversion, be sure to set the impedances of the two unbalanced ports equal to each other.

Port impedance conversion function



Converting Port Impedance

1. Press **Analysis** key, the click **Fixture Simulator** > **Port Z Conversion.**
2. Specify the port reference impedance for the required port.
    - Select **Port 1 Z0 Real**, **Port 2 Z0 Real**, **Port 3 Z0 Real**, or **Port 4 Z0 Real** to set the port reference impedance in Real format.

- Select **Port 1 Z0 Real** and **Port 1 Z0 Imag**, **Port 2 Z0 Real** and **Port2 Z0 Imag**, **Port 3 Z0 Real** and **Port 3 Z0 Imag**, or **Port 4 Z0 Real** and **Port 4 Z0 Imag** to set the port reference impedance in Complex format.

3. Click **Port Z Conversion** to change the port impedance conversion function to the **ON** state.
4. Click **Return**.
5. If **Fixture Simulator** is **OFF**, click the key again to turn it **ON**.

Determining Characteristics After Adding a Matching Circuit

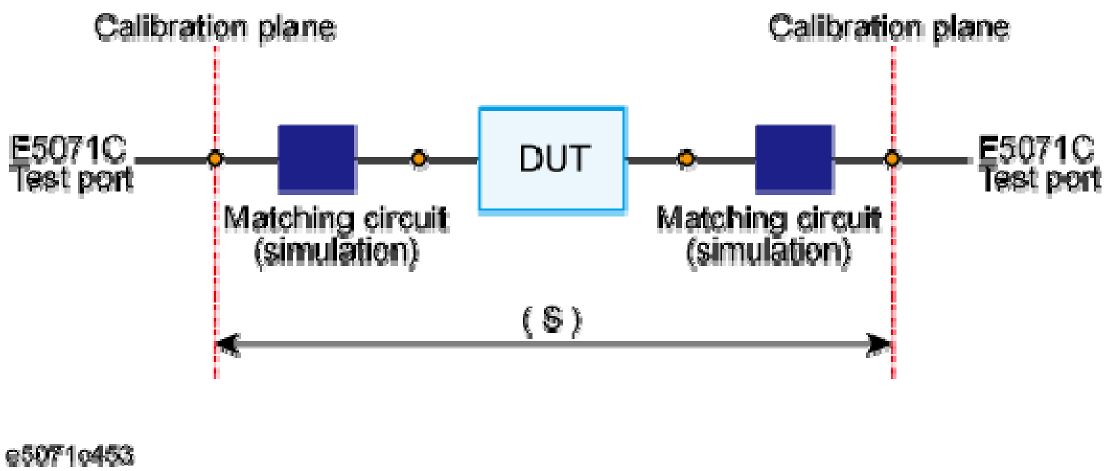
- Overview
- Using Matching Circuit Function

Other topics about Fixture Simulator

Overview

Using the matching circuit embedding function, you can easily obtain the resulting characteristics after adding a matching circuit for each test port.

Matching circuit function



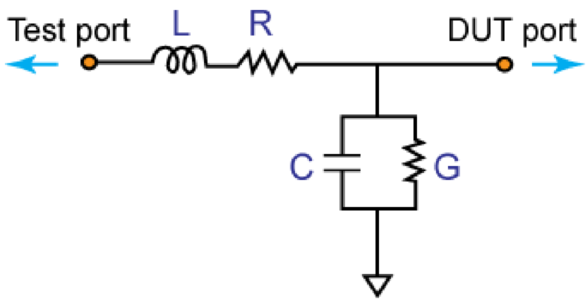
Define the matching circuit to be added by one of the following methods:

- Select one of the nine predetermined circuit models and specify the values for the elements in the circuit model.
- Use a user file (in two-port Touchstone data format) that defines the matching circuit to be added.

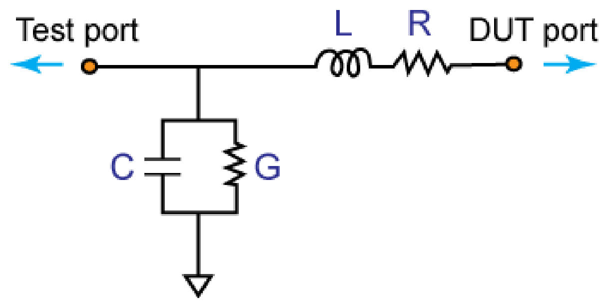
The circuit models used for defining matching circuits are shown below.

Circuit models for defining matching circuits

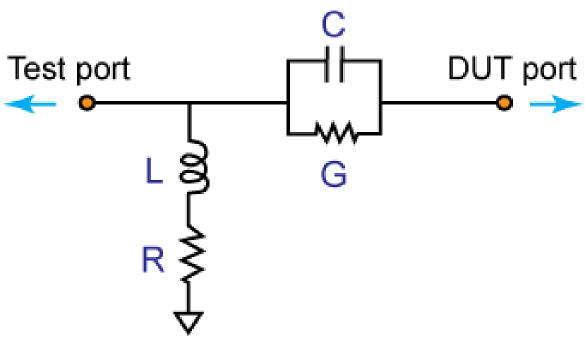
Series L - Shunt C



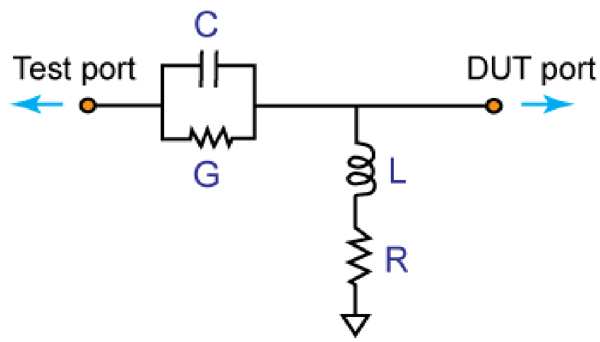
Shunt C - Series L



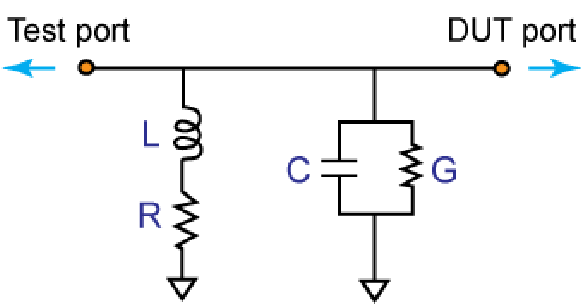
Shunt L - Series C



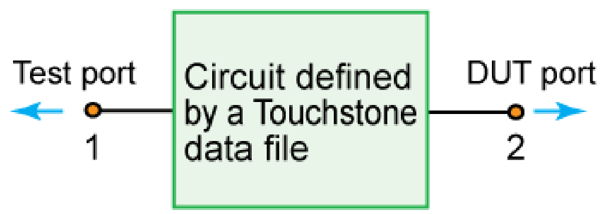
Series C - Shunt L



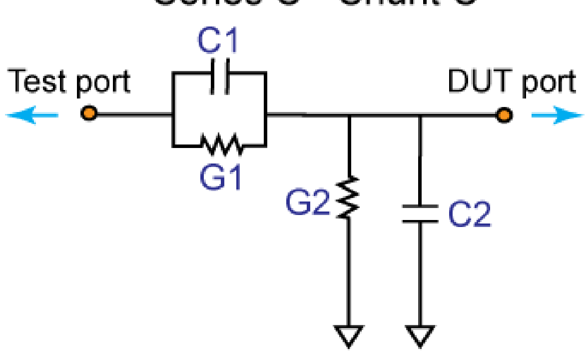
Shunt L - Shunt C



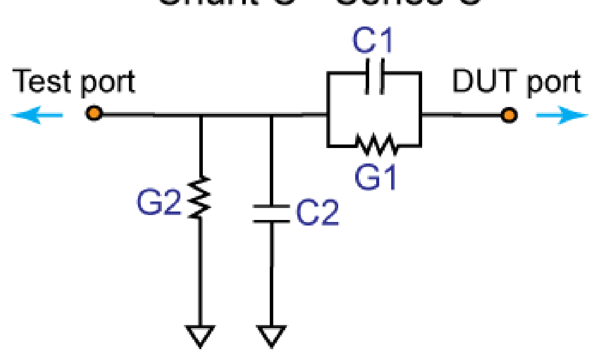
User



Series C - Shunt C



Shunt C - Series C



NOTE In the 2-port matching circuit embedding or the 2-port network de-embedding feature, if the normalized impedance value specified in the user file (2-port Touchstone data file) is different from the port reference impedance setting value of the analyzer, it is automatically converted to adapt to the analyzer setting.

When a 2-port Touchstone data file is read in, data for up to 3202 frequency points are read in using interpolation according to the measurement frequency point setting of the analyzer. If the number of frequency points in the file is greater than 3202, excess data are ignored.

For a network defined in the user file, it is assumed that port 1 is connected to the test port and port 2 is connected to the DUT.

Using Matching Circuit Function

1. Press **Analysis** key.
2. Click **Fixture Simulator** > **Port Matching** > **Select Port.**
3. Click **1**, **2**, **3**, or **4** to select the port to which a matching circuit is to be added.
4. Click **Select Circuit.**
5. Select a matching circuit model.

Softkey	Function
None	No matching circuit is added.
Series L - Shunt C	Selects a circuit model consisting of a series inductor and a shunt capacitor
Shunt C - Series L	Selects a circuit model consisting of a shunt capacitor and a series inductor
Shunt L - Series C	Selects a circuit model consisting of a shunt inductor and a series capacitor
Series C - Shunt L	Selects a circuit model consisting of a series capacitor and a shunt inductor
Shunt L - Shunt C	Selects a circuit model consisting of a shunt inductor and a shunt capacitor
Series C - Shunt C	Selects a circuit model consisting of a series capacitor and a shunt capacitor

Shunt C - Series C	Selects a circuit model consisting of a shunt capacitor and a series capacitor
Series L - Shunt L	Selects a circuit model consisting of a series inductor and a shunt inductor
Shunt L - Series L	Selects a circuit model consisting of a shunt inductor and a series inductor
User	Selects the circuit model defined in the user file imported. To add a matching circuit defined in a user file, execute the following operations before selecting this softkey: 1. Press **User File.** 2. In the dialog box that appears, select the two-port Touchstone data file (.s2p format) for the matching circuit to be added. Once a user file is specified, the selection of **Select Circuit** automatically changes to **User**.

6. For the circuit models, see Circuit models for defining matching circuits.
7. Specify the values of the elements in the selected circuit model.

Softkey	Function
C (C1)	Specifies capacitance [F]
G (G1)	Specifies conductance [S]
L (L1)	Specifies inductance [H]
R (R1)	Specifies resistance [ohm]
C2	Specifies capacitance [F]
G2	Specifies conductance [S]
L2	Specifies inductance [H]
R2	Specifies resistance [ohm]

8. NOTE When either of the two components of Series-C, "C" and "G", is set to 0, the component is defined as "Open"; when both components are set to 0, they are defined as "Short."
   When either of the two components of Shunt-L, "L" and "R", is set to 0, the component is defined as "Short";

when both components are set to 0, they are defined as "Open."

9. Repeat the procedure above to set up the matching circuit for each port used.
10. Click **Port Matching** to turn the matching circuit function **ON.**
11. Click **Return.**
12. If **Fixture Simulator** is **OFF**, press the key again to turn it **ON.**

Obtaining Characteristics After Embedding/De-embedding 4/6/8-port Network

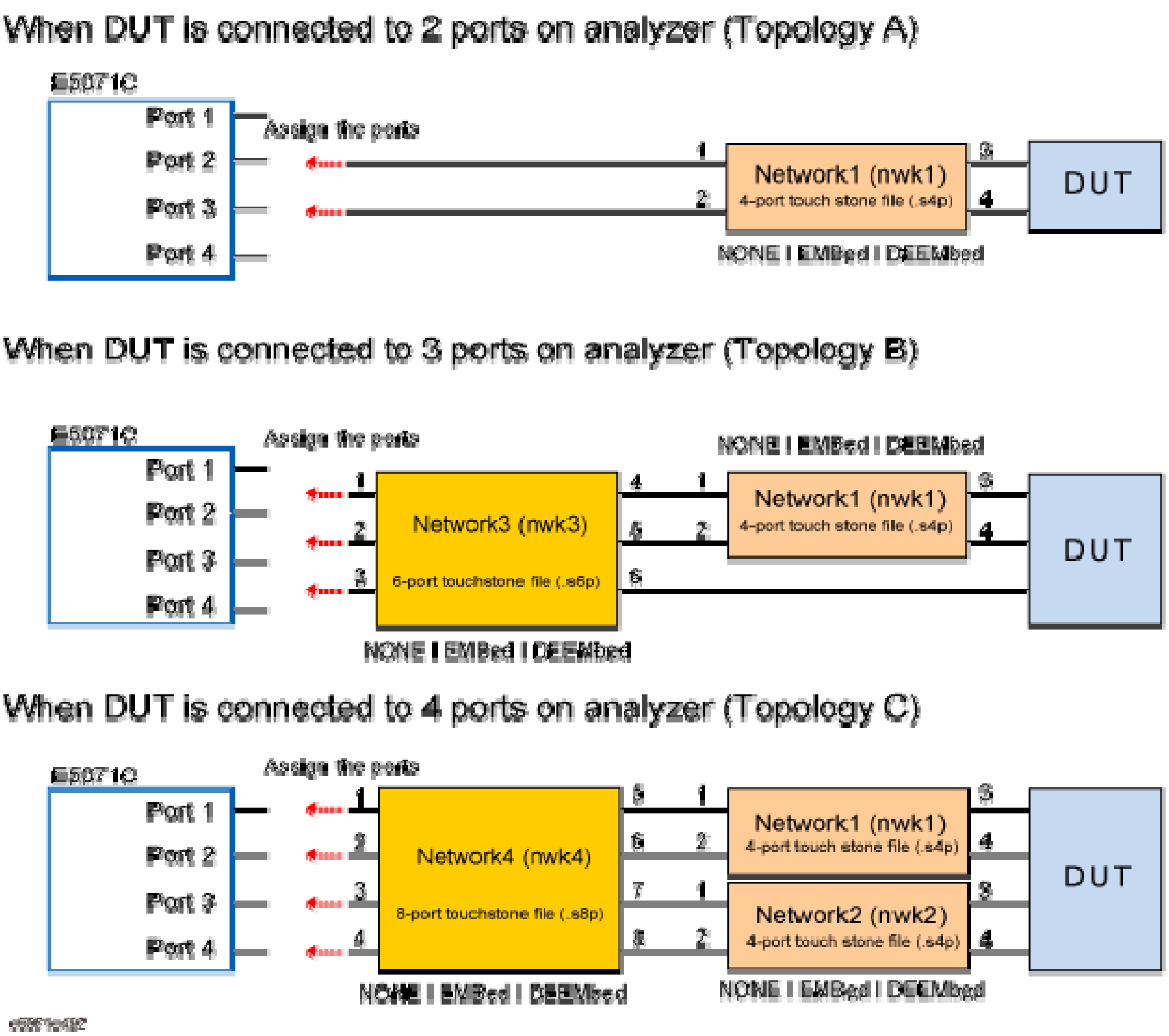
- Overview
- Procedure

Other topics about Fixture Simulator

Overview

The 4/6/8-port network embedding/de-embedding feature allows you to add (embed) or remove (de-embed) data of your desired 4/6/8-port network defined in a 4/6/8-port Touchstone data file to or from measurement values through software-based processing. The embedding/de-embedding supports three types of connection as shown below.

Connection that enables embedding/de-embedding network (4/6/8-port touchstone data)



Touchstone Data File Definition

When defined in the 4-port Touchstone data file, a 4-port network assumes that ports 1 and 2 are connected to the analyzer and that ports 3 and 4 are connected to the DUT.

When defined in the 6-port Touchstone data file, a 6-port network assumes that ports 1, 2 and 3 are connected to the analyzer and that ports 4, 5 and 6 are connected to the DUT.

When defined in the 8-port Touchstone data file, a 8-port network assumes that ports 1 to 4 are connected to the analyzer and that ports 5 to 8 are connected to the DUT.

Topology Selection

For measuring a 4-port DUT with a 4-port network embed/de-embed only on one side, select topology C instead of A, set embedding/de-embedding on the desired side, and specify no processing (None) on the other side.

For measuring a 3-port DUT with a 4-port network embed/de-embed on one side and a 2-port network on the other side, select topology B instead of A, embed/de-embed a 4-port network on one side, and embed/de-embed a 2-port network on the other side using the 2-port matching circuit embedding feature or the 2-port network de-embedding feature. Selecting topology A will result in unexpected measurement.

Procedure

1. Prepare a 4/6/8-port touchstone data file (.s4p/.s6p/.s8p format) corresponding to a network you want to embed/de-embed.
2. Press **Analysis key**.
3. Click **Fixture Simulator** > **De-Embedding SnP** > **Topology** > **Select Topology**.
4. Select a network connection type (topology) from A, B, or C.
5. Click Ports to select measurement port numbers of the analyzer to which you want to connect ports 1 and 2 of the 4-port network.
    - For topology A: Select measurement port numbers of the analyzer to which you want to connect nwk1-1 and nwk1-2. For example, when you connect nwk1-1 and nwk1-2 to measurement ports 1 and 2, respectively, specify Ports ="1-2".
    - For topology B: Select measurement port numbers of the analyzer to which you want to connect nwk3-1, nwk3-2, and nwk3-3. For example, when you connect nwk3-1, nwk3-2, and nwk3-3 to measurement ports 1, 2, and 3, respectively, specify Ports="1-2-3".
    - For topology C: Select measurement port numbers of the analyzer to which you want to connect nwk4-1, nwk4-2, nwk4-

3, and nwk4-4. For example, when you connect nwk4-1, nwk4-2, nwk4-3, and nwk4-4 to measurement ports 1, 2, 3, and 4, respectively, specify Ports="1-2-3-4".

6. Click **User File (nwk*n*)** and select a Touchstone data file you want to use for network n. For nwk1 and nwk2, a 4-port touchstone data file should be selected. a 6-port and 8-port touchstone data file should be selected for nwk3 and nwk4, respectively.
7. Click **Type (nwk*n*)** and select a processing type for network 1.
8. Click **De-Embedding SnP** to turn **ON** the 4/6/8-port network embedding/de-embedding feature.
9. Click **Return**.
10. If **Fixture Simulator** is **OFF**, click the key to turn it **ON**.

At the firmware A.9.61 and above, when the 4/6/8-port network embedding/de-embedding feature reads a 4/6/8-port Touchstone data file,it automatically convert the file's normalized impedance value to adapt to the port reference impedance setting value of the analyzer.

At the firmware A.9.61 and above, the data in the 4/6/8-port Touchstone data file has to be defined with a single normalized impedance value (real) written in the option line.

When a 4/6/8-port Touchstone data file is read in, data for up to 3202 frequency points are read in by using interpolation according to the measurement frequency point setting of the analyzer. If the number of frequency points in the file is greater than 3202, excess data are ignored.

Evaluating Balanced Devices (Balance-Unbalance Conversion Function)

- Overview
- Measurement Parameters of Balanced Devices
- Procedure for Balance-Unbalance Conversion
- Measurement Parameter Setup
- Checking Device Type and Port Assignment

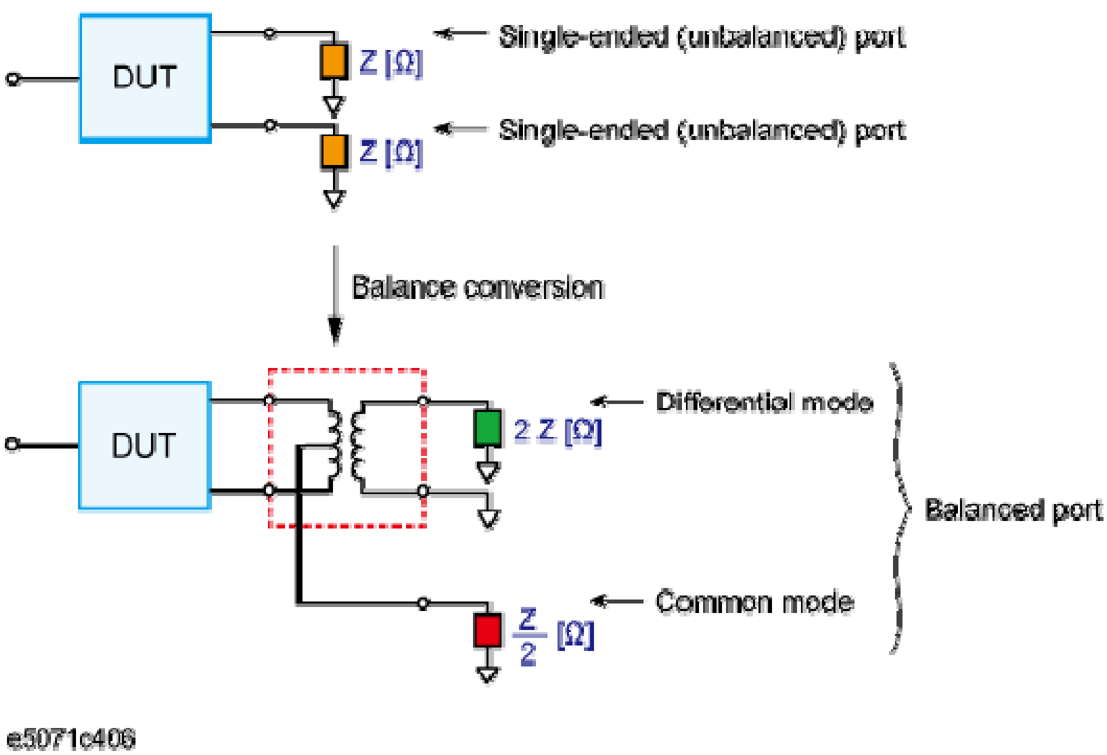
Other topics about Fixture Simulator

Overview

The balance-unbalance conversion function simulates a measurement under a balanced state based on measurement results obtained in an unbalanced state. This function enables you to evaluate devices with balanced ports.

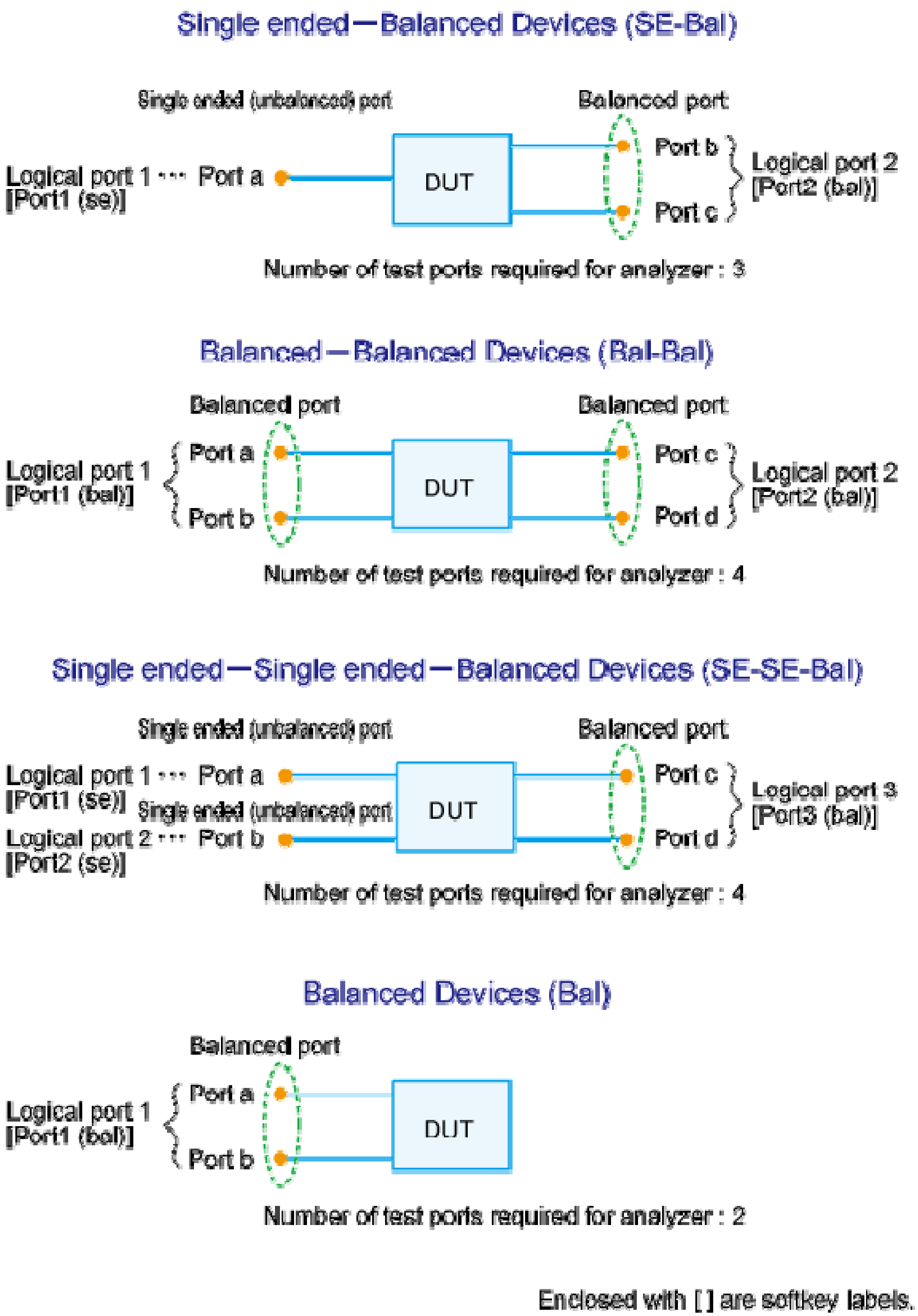
NOTE Be sure to set the impedances of the two unbalanced ports equal to each other. For more details on setting up port impedance for unbalanced ports, refer to Converting the Port Impedance of the Measurement Result.

Balance-unbalance conversion



The types of devices that can be evaluated using the E5071C are shown below. To evaluate a balanced device, an E5071C with at least three test ports is required.

Types of balanced devices that can be evaluated with E5071C



Single ended—Balanced Devices (SE-Bal)
Single ended (unbalanced) port
Balanced port
Logical port 1 ··· Port a
[Port1 (se)]
DUT
Port b
Port c
Logical port 2
[Port2 (bal)]
Number of test ports required for analyzer : 3
Balanced—Balanced Devices (Bal-Bal)
Balanced port
Balanced port
Logical port 1
[Port1 (bal)]
Port a
Port b
DUT
Port c
Port d
Logical port 2
[Port2 (bal)]
Number of test ports required for analyzer : 4
Single ended—Single ended—Balanced Devices (SE-SE-Bal)
Single ended (unbalanced) port
Balanced port
Logical port 1 ··· Port a
[Port1 (se)]
Single ended (unbalanced) port
Logical port 2 ··· Port b
[Port2 (se)]
DUT
Port c
Port d
Logical port 3
[Port3 (bal)]
Number of test ports required for analyzer : 4
Balanced Devices (Bal)
Balanced port
Logical port 1
[Port1 (bal)]
Port a
Port b
DUT
Number of test ports required for analyzer : 2
Enclosed with [ ] are softkey labels.

e5071c375

In the terminology of the E5071C, ports after the balance conversion are called logical ports (or DUT ports).

You can freely assign the test ports of the E5071C to logical ports (ports a to d in the figure above).

Measurement Parameters of Balanced Devices

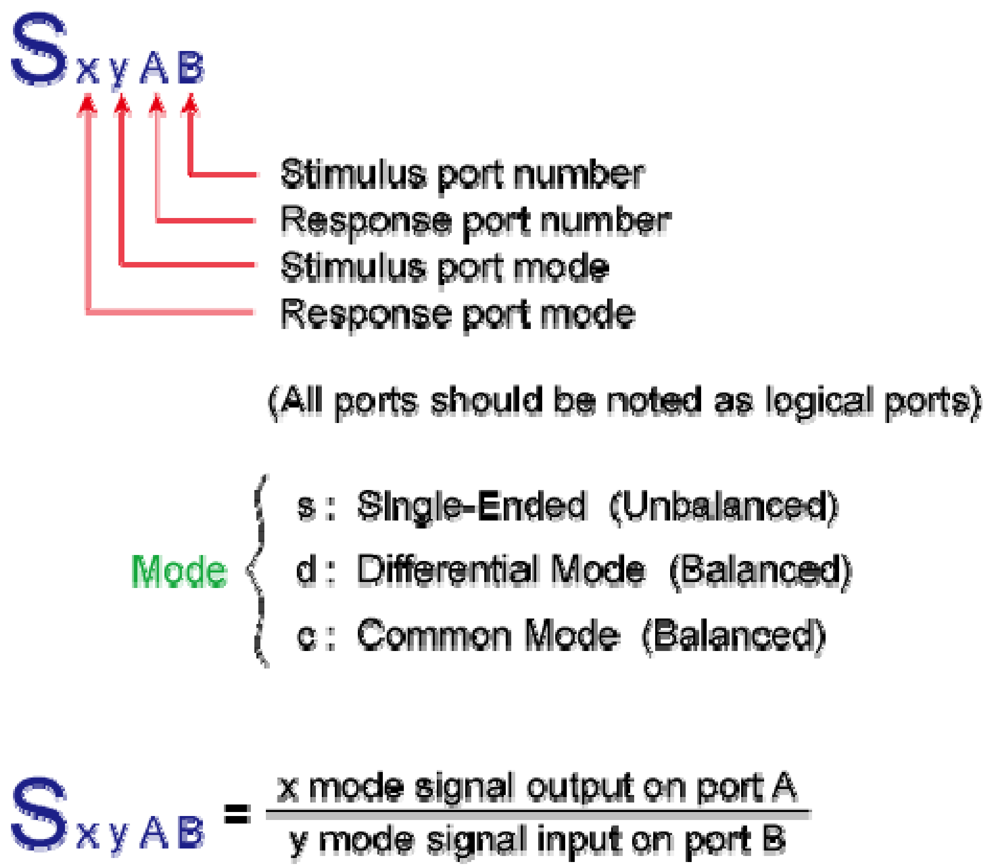
Turn on the balance-unbalance conversion function to measure the following parameters.

- Mixed mode S-parameter
- Imbalance parameter
- CMRR (Common Mode Rejection Ratio)

Mixed mode S-parameter

By turning on the balance-unbalance conversion function, you can obtain the S-parameter of the balanced port separately for two modes, the differential mode and the common mode. The following figure shows the notation of the S-parameter in balance measurement (mixed mode S-parameter).

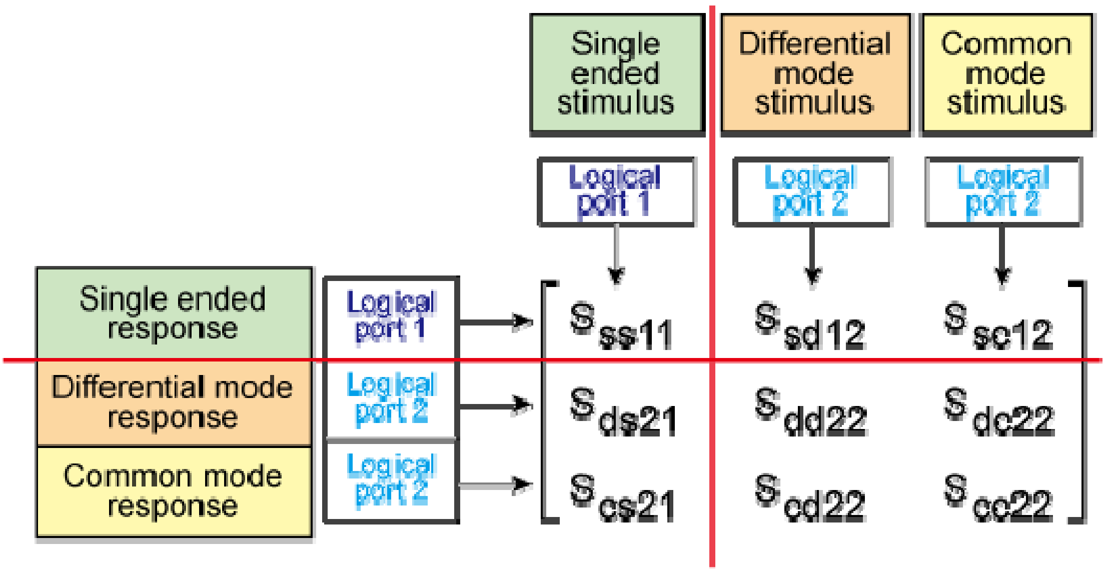
Notation of mixed mode S-parameter



e5071c381

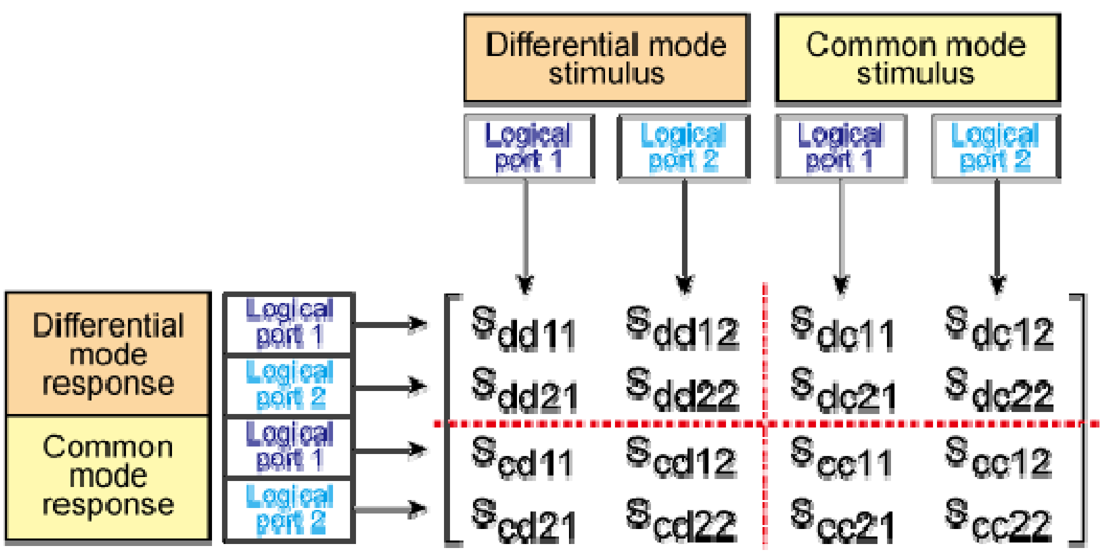
The following two figures show the mixed mode S-parameter when measuring each balanced device.

Mixed mode S-parameter when measuring a single-ended - balanced device



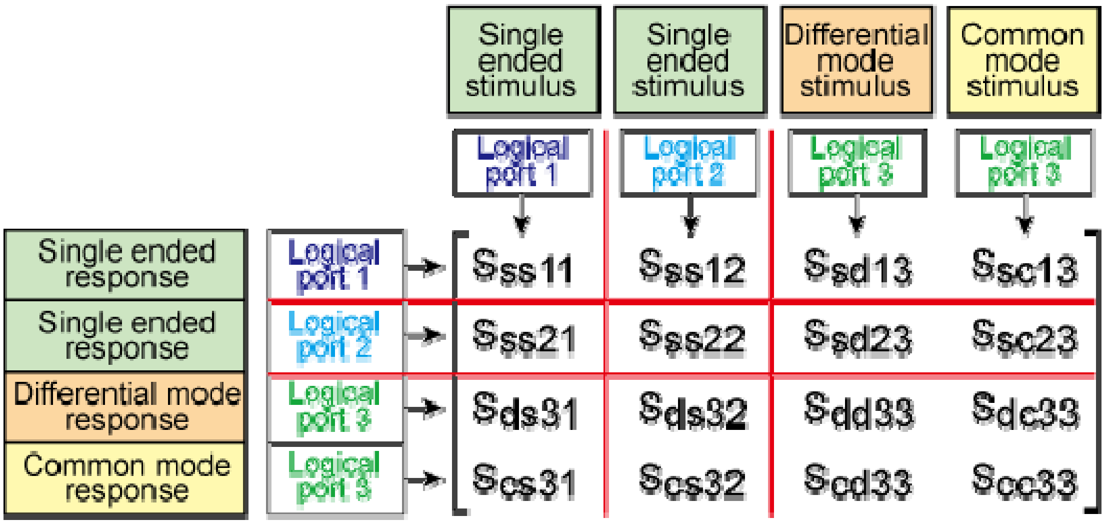
e5071c406

Mixed mode S-parameter when measuring a balanced - balanced device



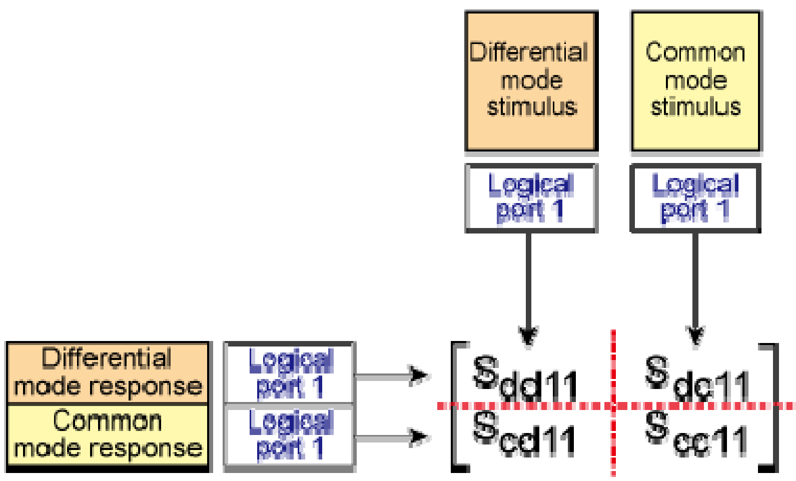
e5071c407

Mixed mode S-parameter when measuring a single-ended - single-ended - balanced device



e5071c409

Mixed mode S-parameter when measuring a balanced device

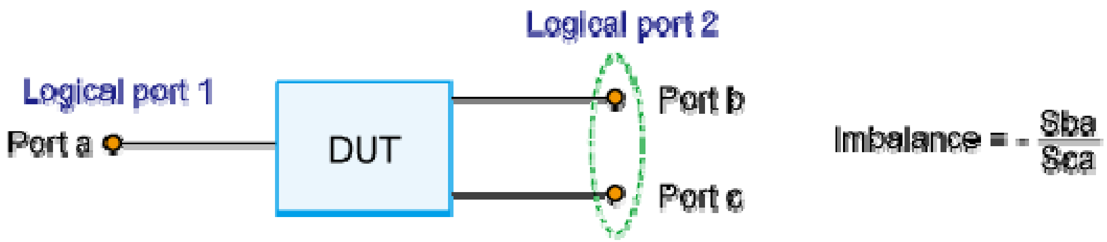


e5071c540

Imbalance Parameter

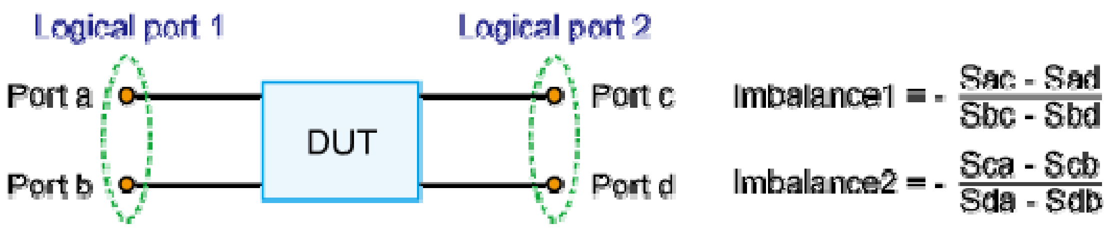
By turning on the balance-unbalance conversion function, you can select the imbalance parameter of the balanced port as the measurement parameter. The following three figures show the imbalance parameter you can select when measuring each balanced device.

Parameter when measuring a single-ended - balanced device (Imbalance)



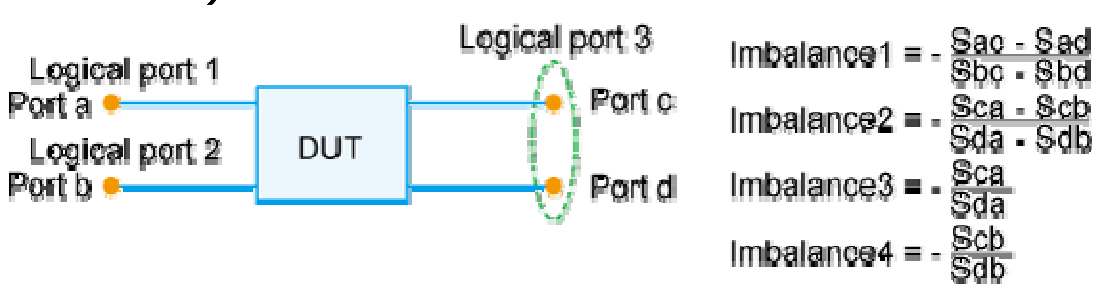
e5071c410

Parameter when measuring a balanced - balanced device (Imbalance1, Imbalance2)



e5071c411

Parameter when measuring a single-ended - single-ended - balanced device (Imbalance1, Imbalance2, Imbalance3, Imbalance4)



e5071c456

NOTE The balance matching circuit (Differential matching circuit embedding) is not applied to single-ended S-parameter results. For example, the balance matching circuit is not applied to the imbalance parameter as it is derived from single-ended S-parameters.

CMRR (Common Mode Rejection Ratio)

By turning on the balance-unbalance conversion function, you can select CMRR (ratio between the transmission characteristic in the differential mode and that in the common mode) of the balanced port as the measurement parameter. The table below shows the CMRR parameter you can select when measuring each balanced device.

Device Type	CMRR
Single-ended - balanced device	S_{ds21}/S_{cs21} and S_{sd12}/S_{sc12}
Balanced - balanced device	S_{dd21}/S_{cc21}
Single-ended - single-ended - balanced device	S_{ds31}/S_{cs31} and S_{ds32}/S_{cs32}

Procedure for Balance-Unbalance Conversion

- When using three test ports, perform a full three-port calibration on the test ports to be used. When using four test ports, perform a full four-port calibration.

NOTE Extend the calibration plane, if necessary, by using the port extension function or network de-embedding function. For more on the port extension and network de-embedding functions, see Extending the Calibration Plane Using Network De-embedding.

1. Press **Analysis** key.
2. Click **Fixture Simulator** > **Topology** > **Device**.
3. Select the balanced/unbalanced topology.

Softkey	Function
SE-Bal	Establishes port 1 on the DUT as an unbalanced port and port 2 as a balanced port
Bal-Bal	Establishes both port 1 and port 2 on the DUT as balanced ports
SE-SE-Bal	Establishes port 1 and port 2 on the DUT as unbalanced ports and port 3 as a balanced port
Bal	Establishes port 1 on the DUT as balanced ports

4. Select each port on the analyzer to which a port on the DUT is connected.

- When you have selected **SE-Bal**:

Softkey	Function
Port 1 (se)	Selects a port on the analyzer from among **1**, **2**, **3**, and **4** for connection to logical 1 (Port a)
Port 2 (bal)	Selects two ports on the analyzer from among **1-2**, **1-3**, **1-4**, **2-1**, **2-3**, **2-4**, **3-1**, **3-2**, **3-4**, **4-1**, **4-2**, and **4-3** for connection to logical port 2 (Port b and Port c)

- When you have selected **Bal-Bal**:

Softkey	Function
Port 1 (bal)	Selects two ports on the analyzer from among **1-2**, **1-3**, **1-4**, **2-1**, **2-3**, **2-4**, **3-1**, **3-2**, **3-4**, **4-1**, **4-2**, and **4-3** for connection to logical port 1 (Port a and Port b)
Port 2 (bal)	Selects two ports on the analyzer from among **1-2**, **1-3**, **1-4**, **2-1**, **2-3**, **2-4**, **3-1**, **3-2**, **3-4**, **4-1**, **4-2**, and **4-3** for connection to logical port 2 (Port c and Port d)

- When you have selected **SE-SE-Bal**:

Softkey	Function
Port 1 (se)	Selects a port on the analyzer from among **1**, **2**, **3**, and **4** for connection to logical 1 (Port a)
Port 2 (se)	Selects a port on the analyzer from among **1**, **2**, **3**, and **4** for connection to logical 2 (Port b)
Port 3 (bal)	Selects two ports on the analyzer from among **1-2**, **1-3**, **1-4**, **2-1**, **2-3**, **2-4**, **3-1**, **3-2**, **3-4**, **4-1**, **4-2**, and **4-3** for connection to logical port 3 (Port c and Port d)

- When you have selected **Bal**:

Softkey	Function
Port 1 (bal)	Selects two ports on the analyzer from among **1-2**, **1-3**, **1-4**, **2-1**, **2-3**, **2-4**, **3-1**, **3-2**, **3-4**, **4-1**, **4-2**, and **4-3** for connection to logical port 1 (Port a)

5. Click **Return**.
6. Click **BalUn** to turn on the balanced/unbalanced state conversion function.
7. Click **Fixture Simulator** to turn on the fixture simulator function.

Measurement Parameter Setup

Performing balance-unbalance conversion enables you to make measurements with mixed mode S-parameters, imbalance parameters and CMRR. Parameters that can be used differ depending on the balance-unbalance topology specified in Balance-Unbalance Conversion.

1. Press **Meas** key (or **Analysis** > **Fixture Simulator** > **Measurement**).
2. Select the measurement parameter.

Checking Device Type and Port Assignment

You can check the device type and the port assignment for the balance-unbalance conversion by displaying the balance measurement topology property as shown below.

Procedure to Turn On/Off Balance Measurement Topology Property Display

Follow these steps to turn on/off the balance measurement topology property display.

1. Press **Analysis** key.
2. Click **Fixture Simulator** > **Topology**
3. Click **Property**. Each press toggles between on/off.

Converting Reference Impedance of Balanced Port

- Overview
- Converting Port Reference Impedance in Differential Mode
- Converting Port Reference Impedance in Common Mode

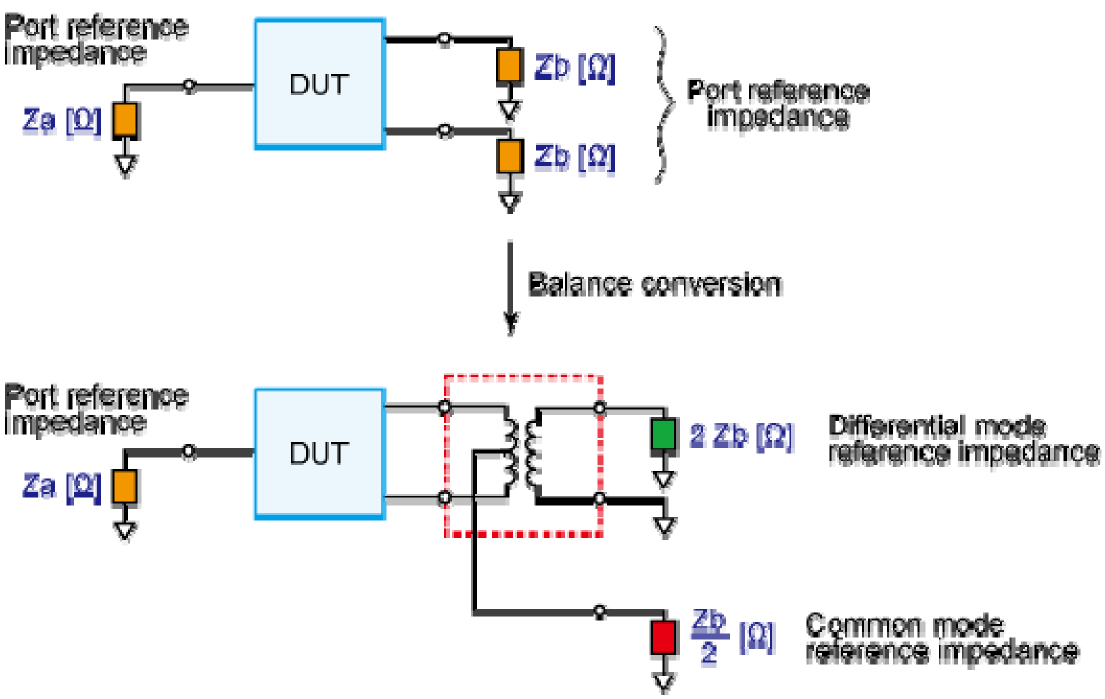
Other topics about Fixture Simulator

Overview

By using the port impedance conversion function, you can specify the impedance of each test port. As a result of this conversion, the impedance of the balanced port in differential mode is set to a value twice as large as the impedance of the two unbalanced ports before conversion, and in common mode to a value one-half as large.

NOTE Be sure to set the impedances of the two unbalanced ports equal to each other. For more details on setting up port impedance for unbalanced ports, refer to Converting the Port Impedance of the Measurement Result.

Port impedance after a balance-unbalance conversion



e5071c403

As described above, the impedance of the balanced port is automatically specified as the result of specifying the impedance of the two unbalanced ports prior to balance-unbalance conversion. However, the port impedance can be changed to an arbitrary value by using the differential port impedance conversion function and the common port impedance conversion function.

Converting Port Reference Impedance in Differential Mode

If you turn on the differential port impedance conversion function, the port reference impedance in the differential mode is converted to an arbitrary value specified with this function instead of the value in the above figure.

Procedure to Turn On/Off Differential Port Reference Impedance Conversion Function

1. Press **Analysis** key, then click **Fixture Simulator** > **Diff Z Conversion.**
2. Click **Diff Z Conversion** to set the differential impedance conversion function to **ON**.

You can only turn on or off Differential Port Impedance Conversion for all of the balanced ports together, not for each port individually. If you want to turn off only a specific port, set the reference impedance of the port to the value in the above figure.

Procedure to Set Differential Port Reference Impedance

1. Press **Analysis** key, then click **Fixture Simulator** > **Diff Z Conversion.**
2. Specify the port reference impedance in differential mode.
3. Select the balanced port from **Port 1 (bal) Real**, **Port 2 (bal) Real**, or **Port 3 (bal) Real** to set the differential port reference impedance in "Real" format.
4. Select **Port 1 (bal) Real** and **Port 1 (bal) Imag**, **Port 2 (bal) Real** and **Port2 (bal) Imag**, or **Port 3 (bal) Real** and **Port 3 (bal) Imag** to set the differential port reference impedance in "Complex" format.
    5. Ports 1, 2, and 3 refer to the logical ports 1, 2, and 3.

Converting Port Reference Impedance in Common Mode

If you turn on the common port impedance conversion function, the port reference impedance in the common mode is converted to an arbitrary value specified with this function instead of the value the above figure.

Procedure to Turn On/Off Common Port Reference Impedance Conversion

1. Press **Analysis** key, then click **Fixture Simulator** > **Cmn Z Conversion.**
2. Click **Cmn Z Conversion** to set the common port impedance conversion function to **ON.**
3. You can only turn on or off Common Port Impedance Conversion for all of the ports together, not for each port individually. If you want to turn off only a specific port, set the impedance of the port to the value in the above figurerocedure to set common port reference impedance
1. Press **Analysis** key, then click **Fixture Simulator** > **Cmn Z Conversion.**

2. Specify the port reference impedance in common mode.
3. Select the balanced port from **Port 1 (bal) Real**, **Port 2 (bal) Real**, or **Port 3 (bal) Real** to set the common port reference impedance in "Real" format.
4. Select **Port 1 (bal) Real** and **Port 1 (bal) Imag**, **Port 2 (bal) Real** and **Port2 (bal) Imag**, or **Port 3 (bal) Real** and **Port 3 (bal) Imag** to set the common port reference impedance in "Complex" format.

   NOTE Ports 1, 2, and 3 refer to the logical ports 1, 2, and 3.

Determining the Characteristics that Result from Adding a Matching Circuit to a Differential Port

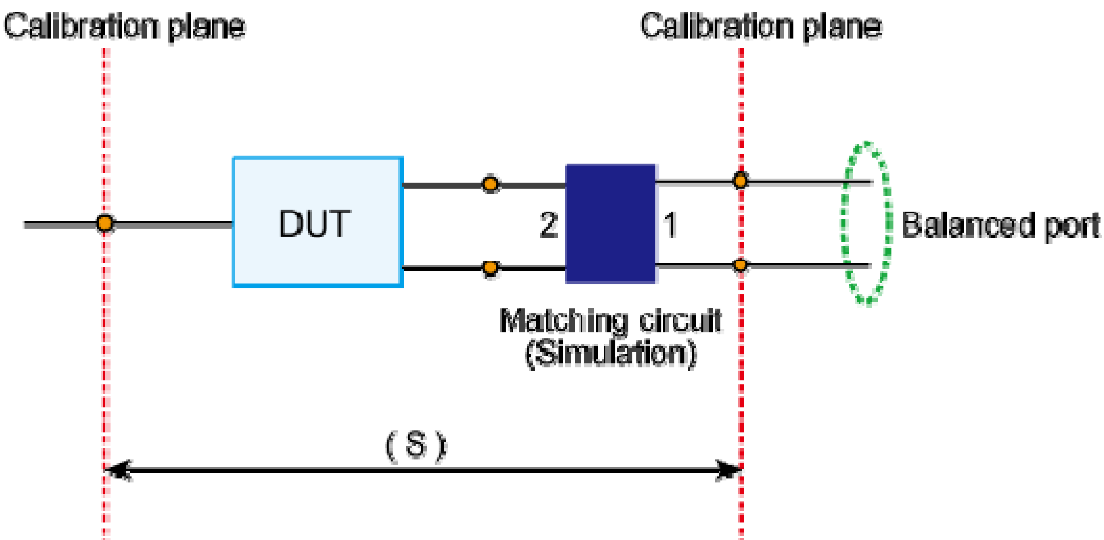
- Overview
- Procedure

Other topics about Fixture Simulator

Overview

You can obtain the characteristics resulting from the pseudo addition of a balance matching circuit to a balanced port created by balance-unbalance conversion. By using the matching circuit function, you can obtain the characteristics resulting from the addition of an arbitrary matching circuit for each test port.

Balance matching circuit function



e5071c376

NOTE The balance matching circuit (Differential matching circuit embedding) is not applied to single-ended S-parameter results. For example, the balance matching circuit is not applied to the imbalance parameter as it is derived from single-ended S-parameters.

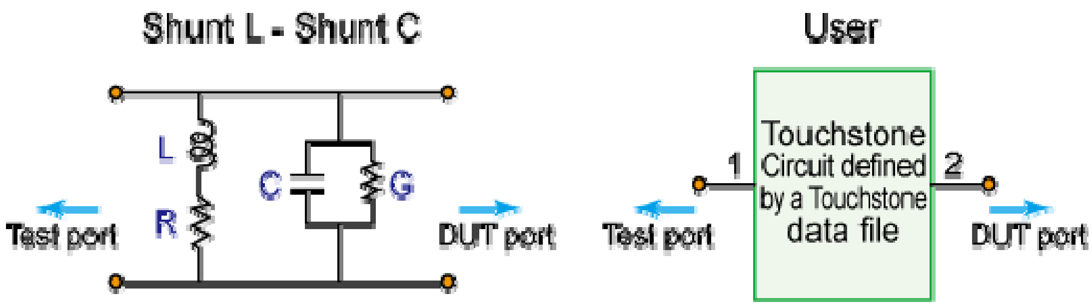
Procedure

Define the matching circuit to be added by one of the following methods:

- Use a predetermined circuit model and specify the values for the elements in the circuit model.
- Use a user file (in two-port Touchstone format) to define the matching circuit to be added.

The following figure shows the circuit models used in defining a balance matching circuit.

Circuit models used to define balance matching circuit



e5071c379

NOTE For a network defined in a user file, it is assumed that port 1 is connected to the test port and port 2 is connected to the DUT.

The setup steps are shown below.

1. Press **Analysis** key, then click **Fixture Simulator** > **Diff. Matching.**
2. Click **Select Port.**
3. Click **1**, **2**, or **3** to select the port on the DUT to which a differential matching circuit will be added.
4. To add a matching circuit defined in a user file, perform the following operations:
    a. Press **User File.**
    b. Using the dialog box that appears, select the 2-port Touchstone data file (.s2p format) for the matching circuit to be added.
    c. Once you have specified the user file, the selection of **Select Circuit** automatically changes to **User**. In this case, you do not have to execute Step 5 and Step 6.
5. Click **Select Circuit.**
6. Select a differential matching circuit model.

Softkey	Function
None	The matching circuit is not added.
Shunt L - Shunt C	Selects a circuit model consisting of a shunt inductor and a shunt capacitor
User	Selects the circuit model defined in the user file imported in Step4.

7. Specify the values for the elements in the circuit model selected.

Softkey	Function
C	Specifies the capacitance [F]
G	Specifies the conductance [S]
L	Specifies the inductance [H]
R	Specifies the resistance [Ω]

8. NOTE When either of the two components of Shunt-L, "L" and "R", is set to 0, the component is defined as "Short"; when both components are set to 0, they are defined as "Open."
9. Repeat the procedure to set up the differential matching circuit to be added to the selected ports on the DUT.
10. Click **Diff. Matching** to turn the differential matching circuit **ON.**
11. Click **Return**.
12. If **Fixture Simulator** is **OFF**, press the key again to turn it **ON.**

Example of Using Fixture Simulator

Other topics about Fixture Simulator

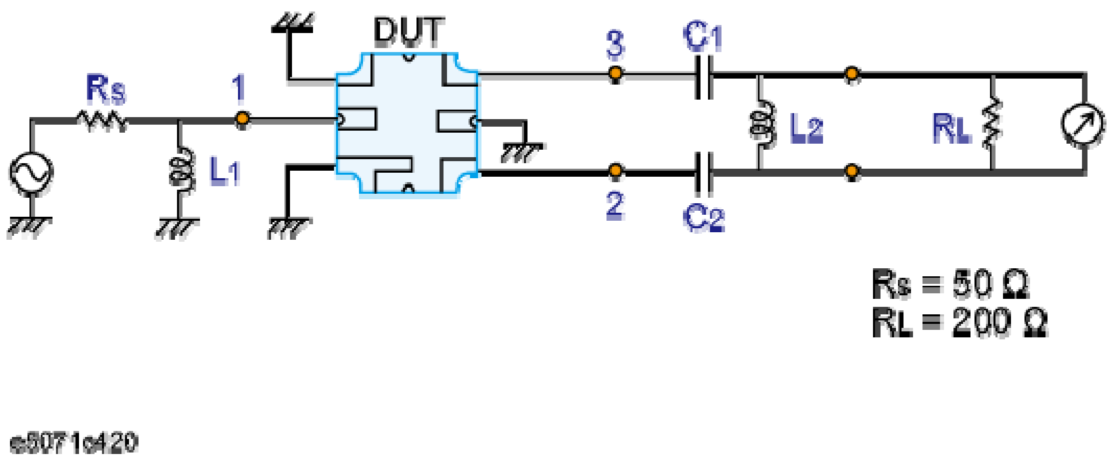
Overview

In this section, the fixture simulation function is explained based on an evaluation example for a DUT (balanced SAW filter) with a balanced port.

Measurement Circuit Example: DUT with Balanced Port

The following figure shows an example of a measurement circuit used to evaluate a balanced SAW filter. DUT port 1 is an unbalanced port connected to source impedance R_s and input matching circuit L_1. DUT port 2 is a balanced port connected to an output matching circuit (C_1, C_2, and L_2) and load resistance R_L.

Measurement circuit of balance SAW filter



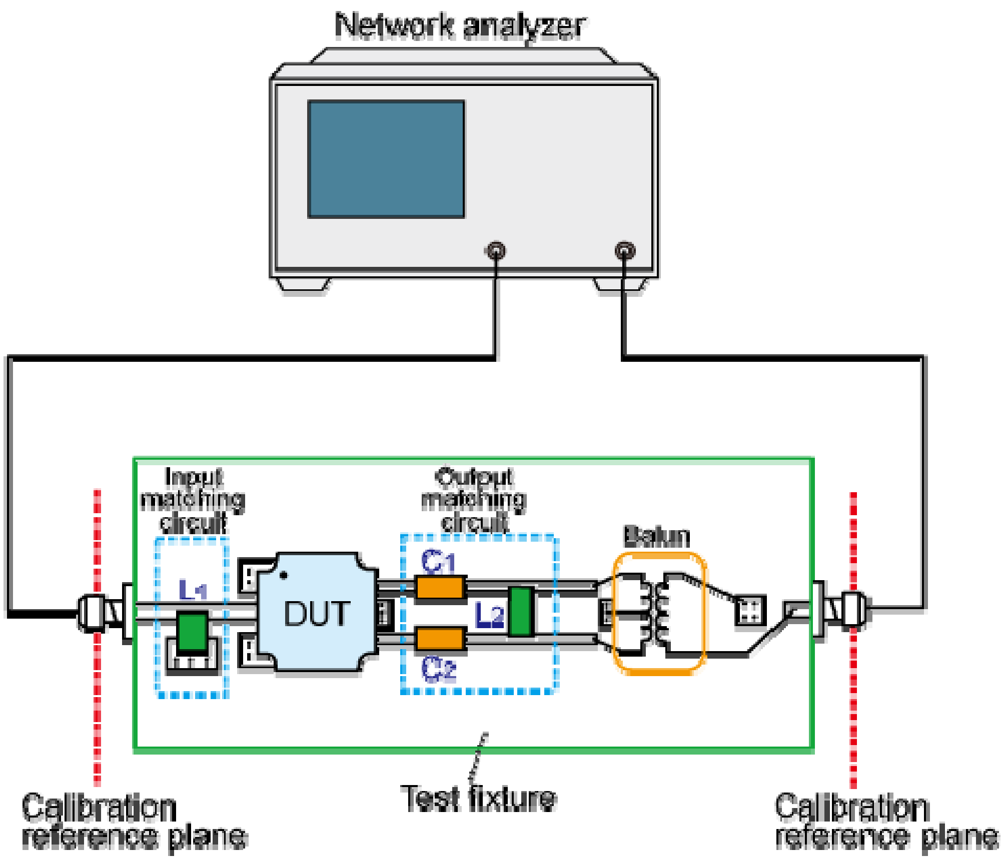
Evaluation Using Actual Test Fixture

Generally, a test fixture as shown in the following figure is fabricated for evaluating the characteristics of a DUT in a measurement circuit by using a network analyzer.

In typical use, a network analyzer performs measurements at a 50 ohm port reference impedance and in a single-ended (unbalanced) state. Therefore, DUT port 1 can be connected directly to the test port of the network analyzer.

On the other hand, DUT port 2 is a balanced port that cannot be connected directly to the test port of the network analyzer. Usually, a balun (BALance-UNbalance transformer) is used to convert the DUT's balanced port to an unbalanced port and to connect the converted port to the test port of the network analyzer. Matching circuits are mounted in the test fixture as shown in the following figure.

DUT evaluation using an actual test fixture



e5071c404

Problems in Measurement with Actual Test Fixture

Evaluating a balanced device with an actual test fixture involves the following problems:

- Calibration cannot be performed at the DUT's terminals. (A DUT's terminals are in the test fixture and calibration standards cannot be connected to them. In addition, it is very difficult to obtain calibration standards that can be used to calibrate a balanced port.) As a result, calibration is performed at appropriate connectors connected to the test fixture, and the network between the calibration reference plane and the DUT's terminals induces measurement errors.

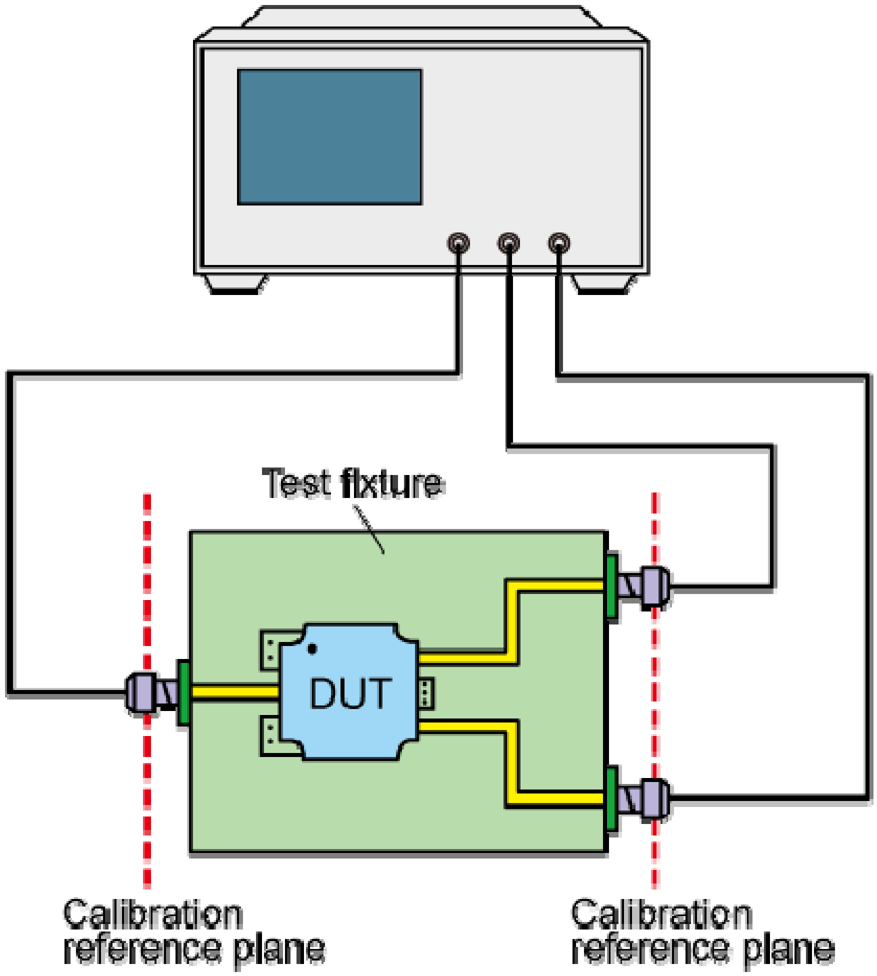
- Different test fixtures must be fabricated for evaluating different types of DUTs because they require different characteristic impedances and matching circuits.
- An actual balun does not have ideal characteristics, so measurement error cannot be avoided. Furthermore, a common mode signal evaluation cannot be performed when an actual balun is used.

DUT Evaluation using E5071C's Fixture Simulator

The E5071C's fixture simulator function simulates a test fixture by using internal software instead of using an actual test fixture for evaluating DUTs.

The following figure shows an example connection for evaluating a DUT with the E5071C's fixture simulator function. The unbalanced port of the DUT should be directly connected to a test port of the E5071C, and the balanced port of the DUT should be connected to two other test ports of the E5071C. The actual measurement by the E5071C is performed at single-ended ports with a 50 ohm port reference impedance.

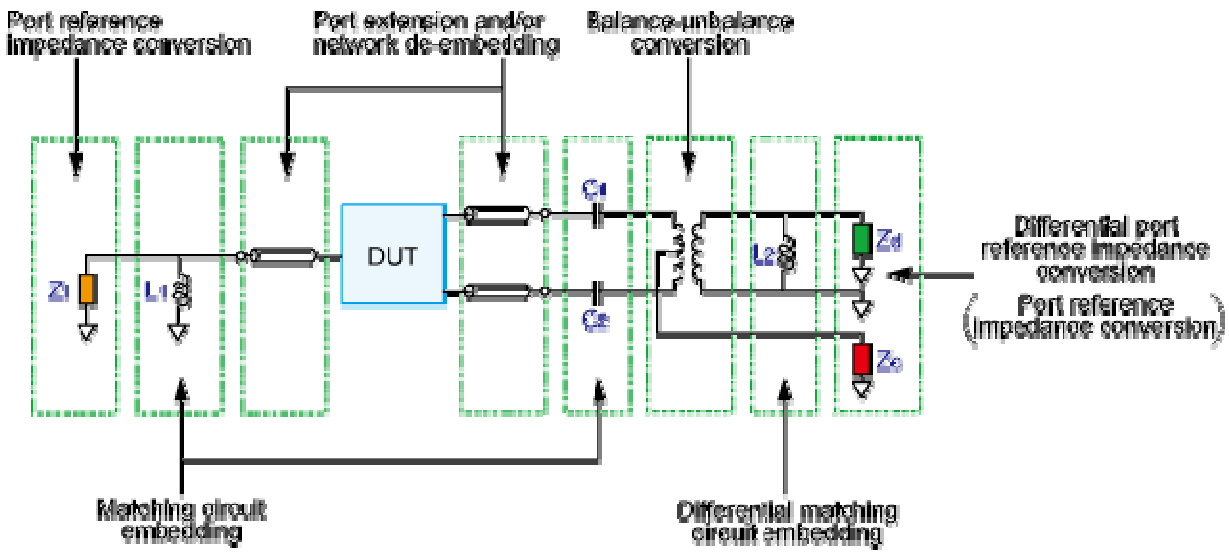
DUT connection when fixture simulator is used



e5071c362

The following figure shows the measurement circuit simulated by the fixture simulator based on actual measurement with the test fixture shown in figure above.

Measurement circuit simulated by fixture simulator



e5071c405

First, the effect of an undesired network can be eliminated by port extension and/or network de-embedding. In the above figure (DUT connection when fixture simulator is used), since calibration standards cannot be connected to the DUT terminals to perform calibration, calibration should be performed at the connectors to the test fixture. Using port extension and/or network de-embedding enables you to remove an undesired network by using data processing and moving the calibration reference plane to the DUT's side equivalently. This function is performed for a single-ended port even if balance-unbalance conversion is applied to the port.

Port reference impedance conversion converts measured S-parameters to those at arbitrary port reference impedance. In the above figure (Measurement circuit simulated by fixture simulator), since the single-ended port of the DUT is connected to the E5071C's test port (50 ohm , single-ended), port reference impedance conversion is not required. This function is performed for a single-ended port even if balance-unbalance conversion is applied to the port.

Matching circuit embedding converts measured S-parameters to those when a matching circuit is added to the DUT's terminal. This function is performed for a single-ended port even if balance-unbalance conversion is applied to the port.

Balance-unbalance conversion converts S-parameters measured at an unbalanced state to mixed-mode S-parameters measured at a balanced state. The balanced port signal can be evaluated by using differential mode and common mode signals.

Differential matching circuit embedding converts measured S-parameters to those when a matching circuit is added to the DUT's differential mode port.(L2 in there figure Measurement circuit simulated by fixture simulator)

Differential port reference impedance conversion converts a differential port reference impedance to an arbitrary impedance. Port reference impedance Z [ohm] at the two single-ended ports before balance conversion is automatically converted to 2Z [ohm] for differential mode port and Z/2 [ohm] for common mode port after balance conversion. Accordingly, if port reference impedance conversion is not performed for the two single-ended ports before balance conversion, differential mode port reference impedance Zd becomes 50 ohm * 2 = 100 ohm , and common mode port reference impedance Zc becomes 50 ohm / 2 = 25 ohm . Since the differential port is terminated with 200 ohm in the figure,Measurement circuit of balance SAW filter, differential port reference impedance Zd should be set to 200 ohm.

Advantages of Balanced DUT Evaluation using Fixture Simulator

Balanced device evaluation using the fixture simulator offers the following advantages:

- Calibration reference plane can be easily moved to the DUT's terminal after calibration is performed at the connectors where calibration standards can be connected. Undesired network can be removed to eliminate measurement errors (port extension, network de-embedding).
- Characteristics of a DUT, including desired matching circuits, can be obtained easily (matching circuit embedding, differential matching circuit embedding). Port reference impedance can be set freely (port reference impedance conversion, differential port reference impedance conversion).
- Differential mode and common mode signal evaluation (mixed-mode S-parameter evaluation) can be performed easily (balance-unbalance conversion).