Agilent E5063A Network Analyzer Help

The E5063A help provides an easy access to the information related to the use of Agilent E5063A Network Analyzer. Pressing **Help** key on the front panel displays the topic related to the selected softkey.

You can navigate this help system through the navigation tools provided in the left or you can directly access the chapters:

Quick Start

Measurement

Test Wizard

Using this Help **(Read me first)**

Precautions

Setting Control Functions

Using Windows

Programming

Product Information

|	
E5063A Firmware Revision	A.01.0x
Help Edition No.	1.00
Help Revision Date	November 14, 2013

Agilent Technologies

© Copyright 2014 Agilent Technologies

Using this Help

This help provides the user and programming documentation in an searchable electronic format for the E5063A Network Analyzer. This section describes the usage of this help system.

- Opening E5063A Help
- Context Sensitive Help
- Closing E5063A Help
- Viewing E5063A Help on PC
- Switching between E5063A Help and Measurement View
- Navigating E5063A Help
- Conventions used in E5063A Help

E5063A Online Help

e5063a113
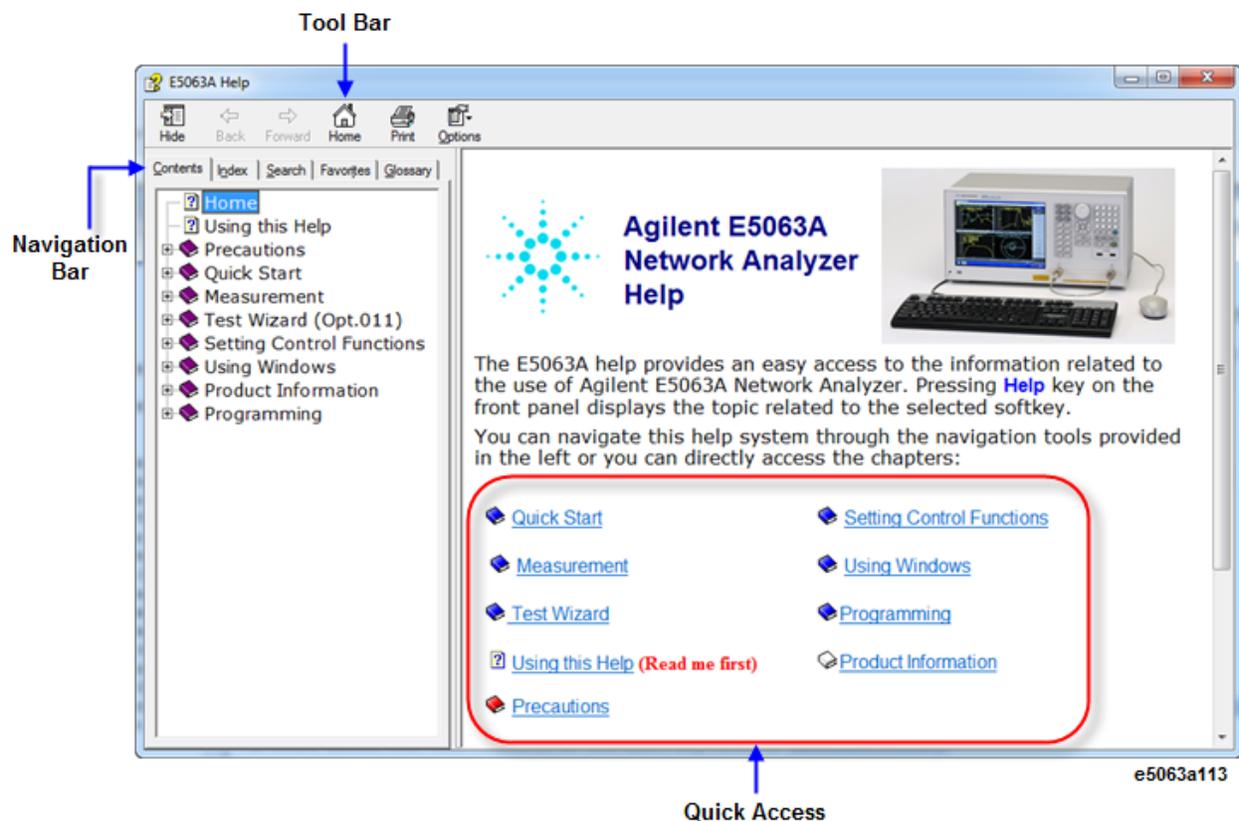
Opening E5063A Help

This help system is provided in Microsoft Compressed HTML Help format (.chm). This help can be also be viewed on a normal PC.

To open E5063A help, use either of the following methods:

- By pressing the **Help** key located in the **ENTRY** Block.
- By pressing the **F1** key on a keyboard attached with the E5063A.
- By double-clicking **E5063A_Help.chm** (located in **C:\Program Files (x86)\Agilent\E5063\Help**).

Context Sensitive Help

Context sensitive help is a great feature of the E5063A help. It allows you to get information about the selected softkey by pressing the **Help** key in the E5063A or by pressing **F1** in a keyboard attached to the E5063A or by clicking the help button in a dialog box. It provides information relevant to the task that needs to be accomplished and reduces the time to search relevant information required to complete a task.

Closing E5063A Help

To close the E5063A help, click , located on the top right of the E5063A help viewer.

Viewing E5063A Help on your PC

The E5063A help can be opened and viewed on a normal PC. The help file (**E5063A_Help.chm**) is located in the E5063A Network Analyzer storage at **C:\Program Files (x86)\Agilent\E5063\Help**. Copy it to the local storage drive on your PC, then double-click it to view.

A web help is also available at http://www.agilent.com/find/e5063a-manual.

> **NOTE** Do not locate E5063A_Help.chm in a network drive because the topics is not displayed.

Switching between E5063A Help and Measurement View

The **Foc** Key, located in the **Entry** Block, can be used to switch between the E5063A Measurement View and Help View.

Navigating E5063A Help

The E5063A help system provides several ways to navigate through the information related to the use of E5063A Network Analyzer. These sections describe the navigation system of the E5063A help which consists of:

- [Navigation bar](#)
- [Quick Access](#)
- [Toolbar](#)

Navigation bar

Navigation bar comprises of tabs related to **Contents**, **Index**, **Search** and **Glossary**. The **Contents** tab contains the main navigational structure of the E5063A help. The **Glossary** tab contains explanation of the terms significant to the E5063A Network Analyzer. The **Index** tab is an additional tool that can be used to navigate different topics according to their alphabetical listing. The **Search** tab can be used to search any term/phrase used in the E5063A online Help.

Toolbar

The toolbar can be used to navigate through the help. The **Home** option can be used to return to the home page of the E5063A help which contains Quick Access to chapters in the help. The **Back** and **Forward** options can be used to toggle between visited topics. The **Print** option can be used to print the selected, or all the topics of the E5063A help.

Quick Access

Quick access, located on the **Home Page** of the E5063A help, is a convenient and quick way to access the contents of the E5063A help.

Conventions used in E5063A Help

Naming Conventions Used in this Help

Convention	Description	Example
File Name/Path	File names and path associated with them as displayed as **Bold** in Arial with 12 pt size.	**C:\Documents and Settings\test.txt** **example2.xls**
Hard keys	Hard Keys (Keys located on the Front panel of E5063A) are displayed in Blue color, **Bold** in Verdana with 12 pt size.	**Trace Max** **Foc** **Marker**

Softkeys	Also known as menu keys, are the names of menu that appear in the Firmware (Software) of E5063A and are displayed as **Bold** in Arial with 12 pt size.	**Auto Scale All** **S22**
BLOCK Names	E5063A Front Panel is divided into 7 blocks. These blocks are displayed in ALL CAPS, **Green Color**, **Bold** in Arial with 12 pt size, followed by the word 'Block'.	**RESPONSE** Block **ENTRY** Block **STIMULUS** Block
Note	This Note sign denotes important information. It calls attention to a condition that is essential for the user to understand.	NOTE This is a Note.
Caution	This Caution sign denotes a hazard. It calls attention to a procedure, practice, or condition that, if not correctly performed or adhered to, could result in damage to or destruction of part or all of the instrument.	CAUTION This is a Caution.
Warning	This warning sign denotes a hazard. It calls attention to a procedure, practice, or condition that, if not correctly performed or adhered to, could result in injury or death to personnel.	WARNING This is a Warning.

Precautions

Precautions

- [Safety](#)
- [Notices](#)
- [Installing Software](#)
- [Protecting the E5063A](#)
- [Before Contacting us](#)

Installing Software

Updating Pre-Installed Software

Do not update pre-installed software in except when Agilent recommends to do so. Before updating or installing software refer to Windows Support Information in http://www.agilent.com/find/ena_support for more information.

Installing User Application Software

Users can install commercial application software for Windows on the E5063A at their own risk. Some application software may affect the measurement performance, especially measurement speed.

Protecting the E5063A

To protect your E5063A, follow the instructions below:

Read the warning labels and specifications

Do not exceed the values provided in the specifications guide or as indicated by the yellow warning labels on the front panel of the E5063A. Refer to the specifications for the conditions required to meet the listed specifications. There will be information regarding E5063A settings, and calibration requirements.

Do NOT Plug off Power Cable during Shutdown Process

Do Not Plug off Power cable until completing the shutdown process.

If you directly interrupt the power supply to the power cable receptacle when the power supply is on, or while turning off the Line Switch (Always ON), the shutdown process will not work. This could damage the HDD of the E5063A.

Do NOT Modify or Reconfigure the Operating System

The Microsoft Windows operating system has been modified and optimized by Agilent to improve the performance of the E5061.

- Do NOT install standard version of the Windows operating system on the E5063A.
- Do NOT change advanced performance settings or group policies.
- Do NOT add or delete any storage drive partitions on the E5063A.
- Do NOT delete the Agilent user account.
- Do NOT modify any of the Agilent software registry entries.
- Do NOT change the settings of Standards and Formats in Regional Options and Languages from default setting (English).
- DO NOT change the display property.

Install Antivirus Protection

The E5063A does NOT have antivirus protection when shipped. Use of an antivirus program is strongly recommended if you connect the E5063A to the LAN (Internet).

In addition, the use of a firewall could help to protect the E5063A from viruses. However, some firewalls could limit DCOM connectivity of the E5063A.

Install Windows Critical Updates

The E5063A is always shipped with the latest service packs and critical updates that were available at the time when firmware is updated. Agilent recommends you to maintain the latest available protection for your E5063A by automatically accepting and installing the latest critical security patches from the Microsoft Windows Update website:
http://windowsupdate.microsoft.com

Run Error Check and Disk Defragmenter

When the E5063A is shutdown unexpectedly or power is removed without properly shutting down, large amounts of storage Drive space is rendered unusable. If shutdown is done in this manner a number of times, the E5063A could become unstable and no longer work.

This HDD space can be recovered by first running Windows Error-checking to find and correct errors on the disk, and then the Disk Defragmenter to recover storage Drive space. See the Windows Help file for more information about Disk Defragmenter.

To run error check and defragmenter follow the procedures below:

1. Click **Windows Start Menu** > **My Computer**.
2. Select the drive you want to perform defragmentation or Error-checking.
3. Click **File**, then **Properties**.
4. Click the **Tools** tab.

Error-checking

1. Click **Check Now**.
2. Check **Automatically fix file system errors**.
3. Click **Start**.
4. Click **Yes** to run disk check on next restart.
5. Manually restart the E5063A. The disk check will run before Windows restarts.

Periodically, check the second box in addition to the first box. The error-checking process takes longer time, but performs a complete check.

Defragmentation

1. Click **Defragment Now...**
2. Click **Defragment** to begin the defragment process.
3. Click **Close** when defragmentation is complete.

Precaution for Use of Storage

Do NOT modify or delete any Files and Folders in the drives other than D drive. Doing so will result in malfunctioning of the device. It is required to execute the System recovery if there is any trouble with the above operations.

Precaution for input connector and cable

- Do not apply excessive DC voltage or current to the test port. Applying excessive DC voltage or current may lead to device failure. In particular, the capacitor might remain charged. Connect the measurement sample (DUT) to the test port (or the test fixture, cables, etc. connected to the test port) after the analyzer has been completely discharged. The damage level is described in the data sheet.

- Do NOT bend, bump or flex any device under test (DUT) connected to the input of the E5063A (such as filters, couplers etc). This will reduce the amount of strain placed on the input connector and the mounting hardware. Make sure externally connected items are properly supported (not freely suspended) from the input.

- Do NOT bend cables repeatedly, as this may damage the cables instantly. Limit the number of connections and disconnections to reduce wear and tear. Inspect connectors prior to use; look for dirt, nicks, and other signs of damage or wear. A bad connector can ruin a good connector instantly. Clean dirty connectors to prevent poor electrical connections and damage to the connector. For more information on cable and connector care, refer to http://www.agilent.com/find/cable_care

Precautions for Electrostatic Discharge (ESD)

ESD can damage or destroy electronic components. Whenever possible, conduct testing at a static-safe workstation. Keep static-generating materials at least one meter away from all components. Before connecting any coaxial cable to an analyzer, momentarily short the center and outer conductors of the cable together.

Maintain working environment condition

Control your environment. Maintain temperature & humidity with a satisfactory range within the instruments specification and prevent large fluctuations.

Precautions for transportation

- Do NOT lift the instrument with your hands at the front panel. If the instrument slips, damage may occur to the keypad, knob, or input connectors. Lift the Instrument by the handles when transporting.
- Do NOT use styrene pellets as packaging materials as these may cause damage to E5063A by generating static electricity.

Check for Proper Ventilation

Periodically check and clean the cooling vents of the E5063A. Inadequate airflow can result in excessive operating temperatures which can lead to instrument failures. When installing the product in a cabinet, the convection in and out of the instrument must not be restricted.

See the installation guide for details.

Precautions for Proper Grounding

- Proper grounding prevents building-up of static charge which may be harmful to the E5063A.
- Do NOT defeat the earth-grounding protection by using an extension cable, power cable, or autotransformer without a protective ground conductor.

Before Contacting us

If you encounter the following problems during startup or operation of the E5063A, in which initial registration of the Windows Operating System has been properly performed, execute system recovery and update the Firmware revision.

The system starts up, but the normal measurement screen does not appear

- The system automatically shuts down immediately after the startup, or the startup process stops.
- The measurement screen appears, but "Power on test fail" or "Calibration data lost" is displayed in the instrument message/warning area against a red background in the lower-left part of the screen. The system enters the service mode. (The instrument status bar in the lower-right displays SVC in red).

Unstable Operation

- The system hangs while the instrument is controlled from an external PCs.
- The blue screen appears and the system hangs.
- The response is much slower than usual.

When execution of system recovery does not result in normal operation, it indicates that failure may have occurred. Contact Agilent customer contacts.

For other problems, refer to Troubleshooting.

Safety

Safety Summary

The following general safety precautions must be observed during all phases of operation, service, and repair of this instrument. Failure to comply with these precautions or with specific WARNINGS elsewhere in this manual may impair the protection provided by the equipment. Such noncompliance would also violate safety standards of design, manufacture, and intended use of the instrument. Agilent Technologies assumes no liability for the customer's failure to comply with these precautions.

- The E5063A complies with INSTALLATION CATEGORY II as well as POLLUTION DEGREE 2 in IEC61010-1. The E5063A is an INDOOR USE product.

- The LEDs in the E5063A are Class 1 in accordance with IEC60825-1, CLASS 1 LED PRODUCT

- Ground the Instrument

  To avoid electric shock, the instrument chassis and cabinet must be grounded with the supplied power cable's grounding prong.

- DO NOT Operate in an Explosive Atmosphere

  Do not operate the instrument in the presence of inflammable gasses or fumes. Operation of any electrical instrument in such an environment clearly constitutes a safety hazard.

- Keep Away from Live Circuits

  Operators must not remove instrument covers. Component replacement and internal adjustments must be made by qualified maintenance personnel. Do not replace components with the power cable connected. Under certain conditions, dangerous voltage levels may remain even after the power cable has been disconnected. To avoid injuries, always disconnect the power and discharge circuits before touching them.

- DO NOT Service or Adjust the Instrument Alone

  Do not attempt internal service or adjustment unless another person, capable of rendering first aid and resuscitation, is present.

- DO NOT Substitute Parts or Modify the Instrument

  To avoid the danger of introducing additional hazards, do not install

substitute parts or perform unauthorized modifications to the instrument. Return the instrument to an Agilent Technologies Sales and Service Office for service and repair to ensure that safety features are maintained in operational condition.

- Dangerous Procedure Warnings

  Warnings, such as the example below, precede potentially dangerous procedures throughout this manual. Instructions contained in the warnings must be followed.

- Dangerous voltage levels, capable of causing death, are present in this instrument. Use extreme caution when handling, testing, and adjusting this instrument.

Safety Symbols

Instruction Manual symbol: the product is marked with this symbol when it is necessary for the user to refer to the instrument manual.

Alternating current.

Direct current.

On (Supply).

Off (Supply).

A chassis terminal; a connection to the instrument's chassis, which includes all exposed metal structure.

Stand-by.

Notices

The information contained in this document is subject to change without notice.

This document contains proprietary information that is protected by copyright. All rights are reserved. No part of this document may be photocopied, reproduced, or translated to another language without the prior written consent of Agilent Technologies.

Microsoft®,MS-DOS®,Windows®,Visual C++®,Visual Basic®,VBA® and Excel® are registered trademarks of Microsoft Corporation.

Java® is registered trademark of Sun Microsystems Corporation.

© Copyright 2014 Agilent Technologies. All rights reserved.

Certification

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

Documentation Warranty

The material contained in this document is provided "as is," and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement will control.

Exclusive Remedies

The remedies provided herein are Buyer's sole and exclusive remedies. Agilent Technologies shall not be liable for any direct, indirect, special,

incidental, or consequential damages, whether based on contract, tort, or any other legal theory.

Assistance

Product maintenance agreements and other customer assistance agreements are available for Agilent Technologies products.

For any assistance, contact your nearest [Agilent Technologies Sales and Service Office](#).

Sample Program

The customer shall have the personal, non-transferable rights to use, copy, or modify SAMPLE PROGRAMS in this manual for the customer's internal operations. The customer shall use the SAMPLE PROGRAMS solely and exclusively for their own purposes and shall not license, lease, market, or distribute the SAMPLE PROGRAMS or modification of any part thereof.

Agilent Technologies shall not be liable for the quality, performance, or behavior of the SAMPLE PROGRAMS. Agilent Technologies especially disclaims any responsibility for the operation of the SAMPLE PROGRAMS to be uninterrupted or error-free. The SAMPLE PROGRAMS are provided AS IS.

AGILENT TECHNOLOGIES DISCLAIMS ANY IMPLIED WARRANTY OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Agilent Technologies shall not be liable for any infringement of any patent, trademark, copyright, or other proprietary right by the SAMPLE PROGRAMS or their use. Agilent Technologies does not warrant that the SAMPLE PROGRAMS are free from infringements of such rights of third parties. However, Agilent Technologies will not knowingly infringe or deliver software that infringes the patent, trademark, copyright, or other proprietary right of a third party.

Useful sample program library for ENA Series Network Analyzers are available at http://www.agilent.com/find/ena_support.

Quick Start

Quick Start

Quick Start helps you to understand the E5063A operation quickly.

- Overview
  - [Front Panel](#)
  - [Rear Panel](#)
  - [Screen Area](#)
- [Three Methods of Operation](#)
  - [Basic Measurement Procedures](#)
  - [Measurement Example of a Bandpass Filter](#)

Overview

Overview

- [Front Panel](#)
- [Rear Panel](#)
- [Screen Area](#)

Front Panel: Names and Functions of Parts

Ground Terminal

Ground terminal is provided with the E5063A and is connected to the chassis of the E5063A. You can connect a banana-type plug to this terminal for grounding.

Hardkeys

ACTIVE CH/TRACE Block

A group of keys for selecting active channels and traces. For more on the concepts of channels and traces, see Setting Channels and Traces.

Hardkey Name	Description
Channel Next	Selects the next channel as the active channel. (Each time the key is pressed, the active channel steps up from the current channel to the channel with one number larger than the current channel number). A channel must be active before you can define such parameters as the sweep range. To change the settings of a channel, use this key to first make the channel active.
Channel Prev	Selects the previous channel as the active channel. (Each time the key is pressed,the active channel steps down from the current channel to the channel with one number smaller than the current channel number).
Trace Next	Selects the next trace as the active trace. (Each time the key is pressed, the active trace steps up from the current trace to the trace with one number larger than the current trace number). A trace must be active before you can define measurement parameters and other settings. To change the settings of a trace, use this key to first make the trace active.
Trace Prev	Selects the previous trace as the active trace. (Each time the key is pressed, active trace steps down from the current trace to the trace with one number smaller than the current trace number).

ENTRY Block

A group of keys used for entering numeric data is provided on the front panel of the E5063A.

Hardkey Name	Description

19

0, 1, 2, 3 ..... 9 , . Keys (numeric keys)	Type numeric characters or a decimal point at the position of the cursor in the data entry area.
+/-	**+/-** alternately changes the sign (+, -) of a numeric value in the data entry area.
G/n, M/u, k/m, x1	Adds a prefix to the numeric data typed by using the numeric key and **+/-** key. One of the two prefixes written on the surface of the key is automatically selected depending on the parameter to be entered. **x1** is entered without a prefix.
Softkey On/Off , Entry Off	Turns off the data entry bar if it is displayed. If the dialog box is displayed, it cancels the entry and closes the dialog box. If the data entry bar and dialog box are not displayed, it turns the softkey menu display on/off.
Bk Sp	Backspace key.
Foc	Changes the selection (focus) among the objects to be manipulated by the **NAVIGATION** Block keys and **ENTRY** Block keys. The objects to be manipulated by the **NAVIGATION** Block keys and **ENTRY** Block keys include softkey menus, data entry areas, tables (e.g., segment tables, limit tables, and marker tables), and dialog boxes. When two or more of these objects are displayed on the screen and need selecting, use this key to change the selection (focus) among the objects to be manipulated. When a softkey menu is selected, the menu name area at the top of the menu is displayed in blue. When a data entry area is selected, the data entry bar is displayed in blue. When a table is selected, the frame of the table window is displayed in light gray. While a dialog box is displayed, the focus is fixed on the dialog box and cannot be changed.
Help	Displays help for E5063A.

INSTR STATE Block

A group of keys related to the macro function, store and call function, control/management function, and presetting of the E5063A (returning it to the preset state).

Hardkey Name	**Description**

Macro Setup	Displays the Macro Setup Menu in Softkey Menu Bar. Manipulating the Macro Setup Menu enables you to start up the Macro Setup dialog.
Macro Menu	Displays the Macro Menu Softkey Menu Bar. Executes a registered macro. Manipulating the Macro Menu enables you to select and execute the defined executable program.
Macro Run	Executes the selected executable program such as VB script.
Save/Recall	Displays the Save/Recall Menu in Softkey Menu Bar. Manipulating the Save/Recall Menu enables you to store the setup conditions to or read from the storage devices, calibration data, and trace data of the analyzer.
Capture/System	First, temporarily saves the data for the image displayed on the LCD screen the moment this key is pressed to the internal memory (clipboard). Immediately after that, displays the System Menu in Softkey Menu Bar. Manipulating the System Menu enables you to define the setup for the limit test and then execute it, or to define the setup for the control and management of the analyzer. Using the **Dump Screen Image** option enables you to store the image data in the clipboard to a file on the storage devices. Also, using the **Print** option in the System menu enables you to print the image data in the clipboard to a printer.
Preset	Displays the Preset Menu in Softkey Menu Bar. Clicking **OK** in the Preset Menu enables you to return the analyzer to the initial setup state, called the preset setup.

MKR/ANALYSIS Block

A group of keys used for analyzing the measurement results by using the markers and etc.

Hardkey Name	Description
Marker	Displays the Marker Menu in Softkey Menu Bar. Manipulating the Marker Menu enables you to turn the markers on/off and move them by entering stimulus values. You can place up to 10 markers on each trace.
Marker Search	Displays the Marker Search Menu in Softkey Menu Bar. Manipulating the Marker Search Menu enables you to move a marker to a specific point (maximum, minimum, peak, and a point with a target value) on a trace. You can also find the

	bandwidth parameters (up to six) and display them.
Marker Fctn	Displays the Marker Function Menu in <u>Softkey Menu Bar</u>. Manipulating the Marker Function Menu enables you to not only specify the marker sweep range and the coupling of markers on a channel but also to display statistics data on traces.
Analysis	Displays the Analysis Menu in <u>Softkey Menu Bar</u>. Manipulating the Analysis Menu enables you to use the analytical function of the fixture simulator, time domain Option 011), Test Wizard (Option 011), and each limit test.

NAVIGATION Block (No Label on Front Panel)

The keys and Rotary knob in the **NAVIGATION** Block are used to navigate between softkey menus, tables (limit table, segment table, etc.), or selected (highlighted) areas in a dialog box as well as to change a numeric value in the data entry area by stepping up or down. When selecting one of two or more objects (softkey menus, data entry areas, etc.) to manipulate with the **NAVIGATION** Block keys displayed on the screen, first press the **Foc** (Focus) key in the <u>ENTRY Block</u> to select the object to be manipulated (placing focus on the object) and then manipulate the **NAVIGATION** Block keys (knob) to move among selected (highlighted) objects or change numeric values.

The following descriptions show how the **NAVIGATION** Block keys work both when the focus is on a softkey menu and when the focus is on the data entry area. For more on tables and dialog boxes manipulation, refer to the manipulation procedure for each of these functions.

- **When the focus is on a softkey menu (softkey menu is selected)**
- **When the focus is on the data entry area (data entry area is selected)**

RESPONSE Block

A group of keys used mainly for setting up response measurements on the E5063A.

Hardkey Name	**Description**
Channel Max	Changes between normal and maximum display of the active channel window. In normal display, all of the defined channel windows (both active and non-active) are displayed in split views on the screen. In maximum display, only the active channel window is displayed over the entire area, with non-active windows not displayed. To maximize the active channel, double-click the channel window frame.

	Measurements are also carried out on the non-active channels that are not displayed.
Trace Max	Changes between normal and maximum display of the active trace window. In normal display, all of the defined trace windows (both active and non-active) are displayed in split views on the screen. In maximum display, only the active trace is displayed over the entire channel window, with non-active traces not displayed. To maximize the trace, double-click anywhere in the channel window. Measurements are also carried out on the non-active traces that are not displayed.
Meas	Displays the Measurement Menu in Softkey Menu Bar. Manipulating the Measurement Menu enables you to specify the measurement parameters (types of S-parameters) for each trace.
Format	Displays the Format Menu in Softkey Menu Bar. Manipulating the Format Menu enables you to specify the data format (data transformation and graph formats) for each trace.
Scale	Displays the Scale Menu in Softkey Menu Bar. Manipulating the Scale Menu enables you to specify the scale for displaying a trace (magnitude per division, value of the reference line, etc.) for each trace. You can also specify the electrical delay and phase offset for each trace.
Display	Displays the Display Menu in Softkey Menu Bar. Manipulating the Display Menu enables you to specify the number of channels and the channel window array, the number and arrangement of traces, the setup for data math, etc.
Avg	Displays the Average Menu in Softkey Menu Bar. Manipulating the Average Menu enables you to define the averaging, smoothing, and IF bandwidth.
Cal	Displays the Calibration Menu in Softkey Menu Bar. Manipulating the Calibration Menu enables you to turn the calibration and error correction on/off and change definitions for calibration kits.

STIMULUS Block

A group of keys for defining the stimulus values (signal sources and triggers).

Hardkey Name	**Description**

Start	Displays the data entry bar for specifying the start value of the sweep range in the upper part of the screen. (It also displays the Stimulus Menu for specifying the sweep range in Softkey Menu Bar.)
Stop	Displays the data entry bar for specifying the stop value of the sweep range in the upper part of the screen. (It also displays the Stimulus Menu in the same way as Start.)
Center	Displays the data entry bar for specifying the center value of the sweep range in the upper part of the screen. (It also displays the Stimulus Menu in the same way as Start.)
Span	Displays the data entry bar for specifying the span value of the sweep range in the upper part of the screen. (It also displays the Stimulus Menu in the same way as Start.)
Sweep Setup	Displays the Sweep Setup Menu in Softkey Menu Bar. Manipulating the Sweep Setup Menu enables you to specify the signal source power level, sweep time, number of points, sweep type, etc.
Trigger	Displays the Trigger Menu in Softkey Menu Bar. Manipulating the Trigger Menu enables you to specify the trigger mode and trigger source. You can specify the trigger mode for each channel.

LCD Screen

The E5063A is equipped with a 10.4-inch TFT color, touch-sensitive LCD screen for displaying traces, scales, settings, softkeys and other measurement related information. The touch screen LCD allows you to manipulate softkeys by touching the LCD screen directly with a finger. For more on the LCD screen, see Screen Area: Names and Functions of Parts.

> **CAUTION** Do not press the surface of the LCD screen with a sharp object (e.g., nail, pen, or screwdriver). Pressing the surface with a sharp-pointed object will damage the LCD screen surface or cause the screen to fail.

> **NOTE** Valid pixels are 99.998 % and more. Below 0.002 % of fixed points of black, blue, green or red are not regarded as failure.

Standby Switch

This switch can turn on/off the E5063A. The color on the button shows the status as shown below:

Indicator Color	Description
Green	Normal power on status.

Orange	Standby status.
Red	Illegal power on status.

To turn off the power of the E5063A, be sure to follow the steps described below:

1. First, press this standby switch or send a shutdown command from the external controller to activate the shutdown process (the processing of software and hardware necessary to turn off the power supply). This will put the E5063A into the standby state.

2. Next, if necessary, turn off power supply to the Power Cable Receptacle (to LINE) on the rear panel.

> **CAUTION** Under normal use, never directly interrupt the power supply to the power cable receptacle on the rear panel when the power supply is on. Always keep the Line Switch (Always ON) at (|). Never turn it off (O).
>
> If you directly interrupt the power supply to the power cable receptacle when the power supply is on, or turn off the Line Switch (Always ON), the shutdown process will not work. This could damage the software and hardware of the E5063A and lead to device failure.
>
> Turning on the power supply after a faulty shutdown may cause the system to start up in a condition called "safe mode." If this occurs, first shut down the system to set it to the standby state and then turn on the power supply again to start up the system in normal mode.

⚠ Test Ports

Port 1 and Port 2

Port 1 and Port 2 are for S-parameter measurement. While signals are being output from a test port, the yellow LED above the test port is lit.

The test ports comply with Installation Category I of IEC 61010-1.

USB Ports

Two USB (Universal Serial Bus) Ports are provided that can be used for connecting to ECal (Electronic Calibration) module, USB keyboard, USB mouse, USB memory or a printer. Connecting a designated ECal module to this port enables ECal measurements to be taken. Connecting a compatible printer to this port enables screen information on the E5063A to be printed. See Using USB for more detail.

Rear Panel: Names and Functions of Parts

24 Bit (Handler) I/O Port

The terminal to which an automatic machine (handler) used on a production line is connected. See 24 Bit (Handler) I/O Port.

Connector type: 36-pin Ribbon (Centronics) connector

Certificate of Authenticity Label

The label shows the licence information of the Windows Operating System

Ethernet Port

A terminal for connecting the E5063A to a LAN (Local Area Network). Connecting this instrument to a LAN enables you to access the storage of this instrument from an external PC or to control this instrument by using SICL-LAN or telnet.

Specification	Value
Connector type	8-pin RJ-45 connector
Base standard	10Base-T/100Base-TX/1000Base-T

External Monitor Output Port (VIDEO)

A terminal to which an external color monitor (display device) can be connected. By connecting a color monitor to this terminal, the same information shown on the LCD screen of the main body can be displayed on an external color monitor.

Connector type: 15-pin VGA connector, female

External Trigger (EXT TRIG)

Input

A connector to which external trigger signals are input. This connector detects the downward transition from the HIGH state in TTL signals as the trigger signal. To use this connector to generate a trigger, you must set the trigger source to the "external" side. The connector type is BNC, female.

Output

The External Trigger Output Port can output the pulse with the specified polarity and position either before or after the measurement of each point. The connector type is BNC, female.

Parameter	Typical Value	Condition

HIGH Level Output Voltage	5 V	$I_{out}=-50\mu A$
LOW Level Output Voltage	0 V	$I_{out}=50\mu A$

CAUTION   The External Trigger Output Port can safely handle a maximum output current of 50 mA.

GPIB Port

The connection of an external controller through General Purpose Interface Bus (GPIB) connector allows you to configure an automatic measurement system.

This GPIB port is used only for controlling the E5063A from an external controller. Use USB/GPIB interface to control other devices from the E5063A. You cannot control other devices from the E5063A through this GPIB port.

Solid State Drive (SSD)

Built in Solid State Drive

Reference Signal Ports (10MHz)

The specification for the signals is specified in the data sheet.

Input (REF IN)

The reference signal input connector is used for phase-locking the measurement signal from the E5063A to the external frequency reference signal. Inputting reference signal to this connector improves the accuracy and frequency stability of the measurement signal from the E5063A. When the frequency reference signal is input to this connector, the measurement signal from the E5063A is automatically phase-locked to the reference signal. When an input signal is not present, the frequency reference signal inside the E5063A is automatically used. The ExtRef on the instrument status bar is displayed in blue when the system is phase-locked to the external reference signal and in gray when not phase-locked.

Output (REF OUT)

A connector for outputting the internal frequency reference signal from the E5063A. By connecting this output connector to the external reference signal input connector of another device, the device can be phase-locked to the internal reference signal of the E5063A and used under this condition.

Line Switch (Always ON)

Always keep this switch on (|).

CAUTION   Do not use this switch to turn off (O) the mains. Doing so may cause the analyzer to fail. For more information, see the description of the Standby Switch.

Power Cable Receptacle (to LINE)

The receptacle (outlet) to which the power cable is connected.

> **NOTE** To connect the device to a power source (outlet), use the supplied three-prong power cable with a ground conductor. The plug attached to the power cable (on the power outlet side or device side of the cable) serves as the disconnecting device (device that cuts off power supply) of the E5063A. When the power supply must be cut off to avoid such danger as electric shock, pull out the power cable plug (on the power outlet side or device side of the cable). For the procedure for turning off the main in normal use, see the description in Standby Switch.

For more on the power supply, see the Installation Guide.

Serial Number Plate

The label showing the product number, serial number and the installed option number. The accessary and system rack options are not listed on this label. (CFGxxx or ATOxxx in the first line is Agilent Use Only.)

USB Interface Port (USBTMC)

Through this port, you can control the E5063A from external controllers. For more information on the measurement system using the USB port, see the USB Remote Control System.

Specification	Value
Connector type	Universal serial bus (USB), type B (4 contact positions), Female (jack)
Compliance Standards	USBTMC-USB488 and USB2.0

USB Ports

Four USB (Universal Serial Bus) ports are provided that can be used for connecting to ECal (Electronic Calibration) module, USB keyboard, USB mouse, USB memory or a printer. Connecting a designated ECal module to this port enables ECal measurements to be obtained. Connecting a compatible printer to this port enables screen information on the E5063A to be printed. See Using USB for more detail.

Screen Area

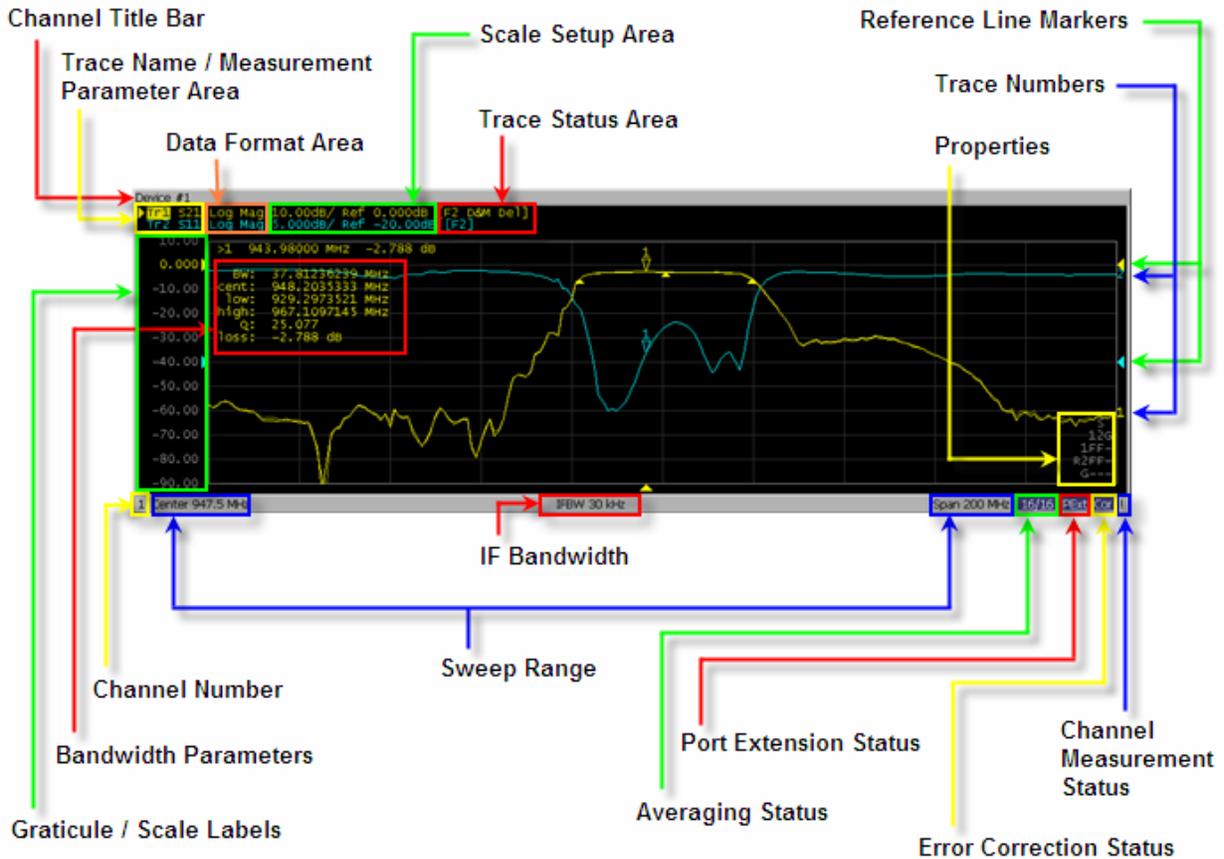### Screen Area: Names and Functions of Parts

Click on the name or area for details of the topic.

- [Menu Bar](#)
- [Data Entry Bar](#)
- [Softkey Menu Bar](#)
- [Instrument Status Bar](#)
- [Channel Window](#)

E5063A Help Operation Section
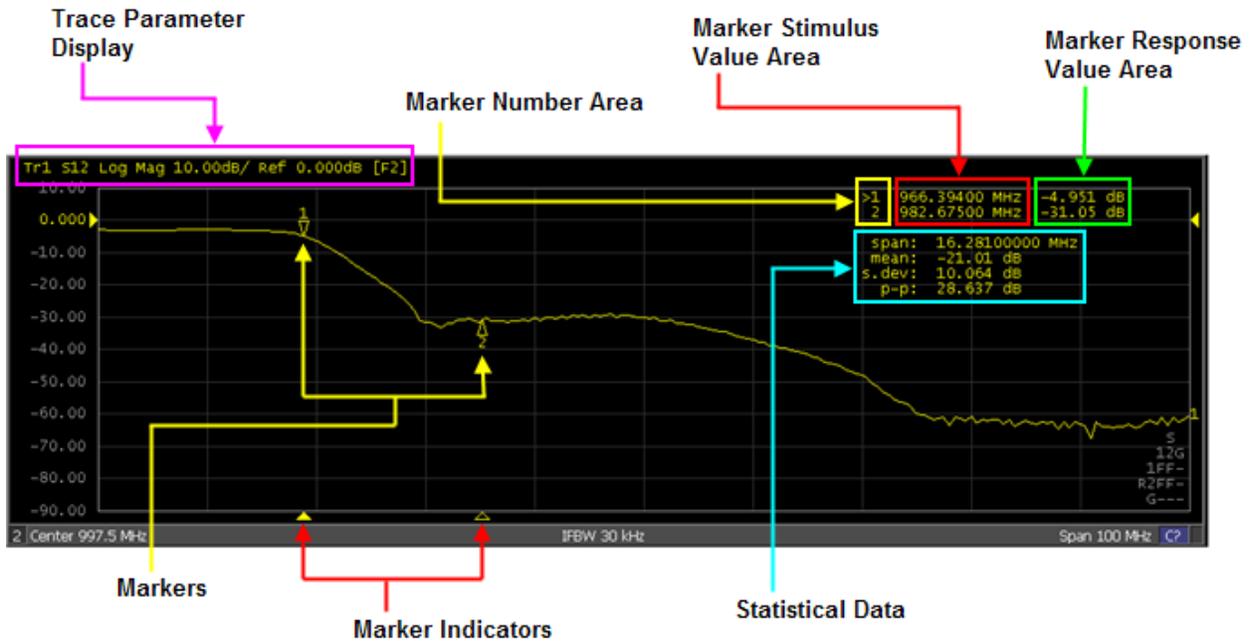
Screen Area: Names and Functions of Parts

Click on the name or area for details of the topic.

- Menu Bar
- Data Entry Bar
- Softkey Menu Bar
- Instrument Status Bar
- Channel Window

Channel Window

Window for displaying the traces. Because a channel corresponds to a window, it is called a channel window. When the outer frame of a channel window is displayed in light gray, it shows that the channel is an active channel (the channel for which setup is being performed). In the following figure, Channel 1 (the upper window) is the active channel. To make a channel active, use **Channel Next** or **Channel Prev**. Clicking inside a channel window will also make the channel active.

Channel 1 Window and Channel 2 Window describes different measurement parameters available in the channel measurement window. The measurement parameters described in the Channel 1 and 2 Window correspond to the same channel measurement window and are displayed in separate windows for ease of read.

Channel 1 Window

Channel 2 Window

Trace Parameter Display

Marker Number Area

Marker Stimulus Value Area

Marker Response Value Area

Markers

Marker Indicators

Statistical Data

e5061b080

Fixture Simulator

e5063a117

Fixture Simulator Status

Bandwidth Parameters
Turning on the bandwidth search function displays the bandwidth parameters here.

Channel Number
Indicates the channel number.

Channel Status Bar
The status of each channel is displayed here.

Averaging Status

Displays the averaging factor and averaging count when averaging is turned on.

n/m (displayed in blue)	Averaging: ON (m: averaging factor; n: averaging count)
(not displayed)	Averaging: OFF

Channel Measurement Status

Displays the update status of traces on the channel.

!	Measurement in progress. When the sweep time exceeds 1.5 seconds, ↑ is displayed at the point on the trace.
#	Invalid traces. The measurement conditions have changed, but the traces on the channel currently displayed have not been updated to match the new conditions.
(No display)	The measurement has not been executed.

Error Correction Status

Displays the execution status of error correction on the channel.

IF Bandwidth

Indicates the IF bandwidth when the sweep type is linear/log frequency.

Port Extension Status

Shows whether the port extension is turned ON or OFF.

PExt (displayed in blue)	Port extension: ON
(not displayed)	Port extension: OFF

Sweep Range

Indicates the sweep range by using the start/stop or center/span.

Channel Title Bar
You can assign a title to each channel and have the title displayed on the bar.

Data Format
The data format of each trace is displayed here.

Graticule Labels
Y-axis divisions in the rectangular display format. When traces in the rectangular display format are overlaid, the Y-axis divisions for the active trace are displayed. The value of the reference line (the division line between ▶ and ◀) is entered numerically by opening the data entry bar using the keys: **Scale** > **Reference Value**. You can change values of the reference line at one-division intervals by placing the mouse pointer in the area of the graticule label (the pointer changes from ↖ to ↕), moving the pointer vertically with the left mouse button pressed, and then releasing the button at the desired location.

34

Fixture Simulator

Fixture Simulator Status indicates that the Fixture Simulator function has been turned ON. To turn ON/OFF Fixture Simulator :

Analysis > **Fixture Simulator** > **ON | OFF**

Marker Indicators

Indicates the positions of markers on the stimulus axis.

▲	Active marker indicator
△	Non-active marker indicator

You can also move a marker to the desired position by placing the mouse pointer on the marker indicator or position of the marker itself (the pointer changes from ⟍ to ↔), moving the indicator vertically with the left mouse button pressed, and then releasing the button at the desired location.

Marker Numbers

Displayed marker number is listed. For the active marker (the one for which setup and analysis are being performed), > is displayed at the left of the marker number. For the reference marker, △ is displayed instead of the marker number.

Marker Response Values

The marker response value for each marker (the measurement value at the marker point) is displayed here. Two (or three) response values are displayed for data in Smith chart or polar display format.

Markers

The markers used for reading values on a trace. Up to 10 markers can be displayed for each trace.

n ▽	Active marker (the one for which setup and analysis are being performed)
△ n	Non-active marker

Here, "n" denotes a marker number. For the reference marker, however, nothing is displayed at the location of n. Clicking the marker or one of the Marker Indicators makes the marker active.

Marker Stimulus Values

The marker stimulus value for each marker (the frequency/power level at the marker point) is displayed here.

Properties

Displays the status of the obtained calibration coefficients on the channel.

Reference Line Indicators

The indicators that indicate the position of the reference line for the Y-axis scale in the rectangular display format. One indicator is to the right and the other is to the left of the scale (▶and ◀). To enter a numeric value for the position of the reference line, open the data entry bar using the keys: **Scale** > **Reference Position**. You can also move the position of the reference line by placing the mouse pointer on either of the two reference line indicators (the pointer changes from ⇖ to ↕.), moving the indicator vertically with the left mouse button kept pressed, and then releasing the button at the desired location (i.e., a drag-and-drop operation).

Scale Settings

The scale setting for each trace is displayed here. This example shows that "10.00dB/" corresponds to 10 dB per division. "Ref 0.000dB" shows that the value of the reference line is 0 dB.

Statistics Data

Turning on the statistics data function displays statistics data here.

Trace Name/Measurement Parameter

The names of the traces, such as Tr1, on the channel and their measurement parameters are displayed here. ▶ to the right of the trace name indicates the active trace (the trace for which setup is being performed). To make a trace active, use **Trace Next** or **Trace Prev**. Clicking the line where the trace name is placed (the mouse pointer changes from ⇖ to 🖑) also makes a trace active.

Trace Number

In the rectangular display format, the trace number is displayed in the same color as the trace at the right end of each trace.

Trace Status Area

The setup for each trace is displayed here.

Trace status display

Classification	Contents inside [ ]	Meaning
Error correction	**RO**	Error correction: ON (OPEN (n) response calibration)

	RS	Error correction: ON (SHORT (n) response calibration)
	RT	Error correction: ON (THRU (n) response calibration)
	ER	Error correction: ON (Enhanced response calibration)
	F1	Error correction: ON (1-port calibration)
	F2	Error correction: ON (Full 2-port calibration)
Turning on/off traces	Nothing	Data trace: ON, Memory trace: OFF
	M	Data trace: OFF, Memory trace: ON
	D&M	Data trace: ON, Memory trace: ON
	off	Data trace: OFF, Memory trace: OFF
Performing data math	**D+M** (**D+M&M**)	Execution of Data+Mem math
When a	**D- M** (**D-**	Execution of

memory trace is ON, see the contents inside ()	**M&M**)	Data- Mem math
	D*M (D*M&M)	Execution of Data*Mem math
	D/M (D/M&M)	Execution of Data/Mem math
Electrical delay	**Del**	A numeric value other than 0 (zero) is specified as the electrical delay or phase offset.
Smoothing	**Smo**	Smoothing: ON
Gating	**Gat**	Gating: ON
Time Domain	**FL**	Fault location: ON
Parameter conversion	**Zr**	Conversion: ON (Impedance: Reflection measurement)
	Zt	Conversion: ON (Impedance: Transmission measurement)
	Ztsh	Conversion: ON (Impedance: Transmission-Shunt measurement)

	Yr	Conversion: ON (Admittance: Reflection measurement)
	Yt	Conversion: ON (Admittance: Transmission measurement)
	Ytsh	Conversion: ON (Admittance: Transmission-Shunt measurement)
	1/S	Conversion: ON (Inverse S-parameter)
	Conj	Conversion: ON (Conjugation)
Equation Editor	Equ	Equation Editor: ON

Data Entry Bar

Used to enter numeric data into the E5063A. Press a hardkey or softkey to enter data, and the data entry bar will appear at the top of the screen. To assign a title to a channel window, an entry bar that allows you to enter letters and symbols by using the front panel keys or mouse is displayed.

Data entry bar

- To manipulate the data entry bar by using the front panel keys, the data entry bar must be selected as the object to manipulate (with the focus placed on it). When the focus is placed on the data entry bar, the entire bar is displayed in blue. Pressing or clicking **Foc** Key in the ENTRY Block enables you to move the focus to the desired object.

Close Button

Closes the data entry area (turns off the display). Use mouse to manipulate this button.

Data Entry Area

When the data entry bar is displayed for the first time, the current settings are displayed on it. You can change numeric values by typing from the keyboard or in the ENTRY block on the front panel.

You can hide the frequency information in order to ensure its confidentiality or for other reasons. For detailed information, see Hiding Softkey's Frequency Information.

Parameter Name

Displays the name of the parameter for which data will be entered.

Step Button (Small)

Increases or decreases the numeric value in the data entry area in small steps. Use the mouse to manipulate this button.

Step Button (Large)

Increases or decreases the numeric value in the data entry area in large steps. Use the mouse to manipulate this button.

Instrument Status Bar

The instrument status bar displays the status of the entire instrument.

e5063a093
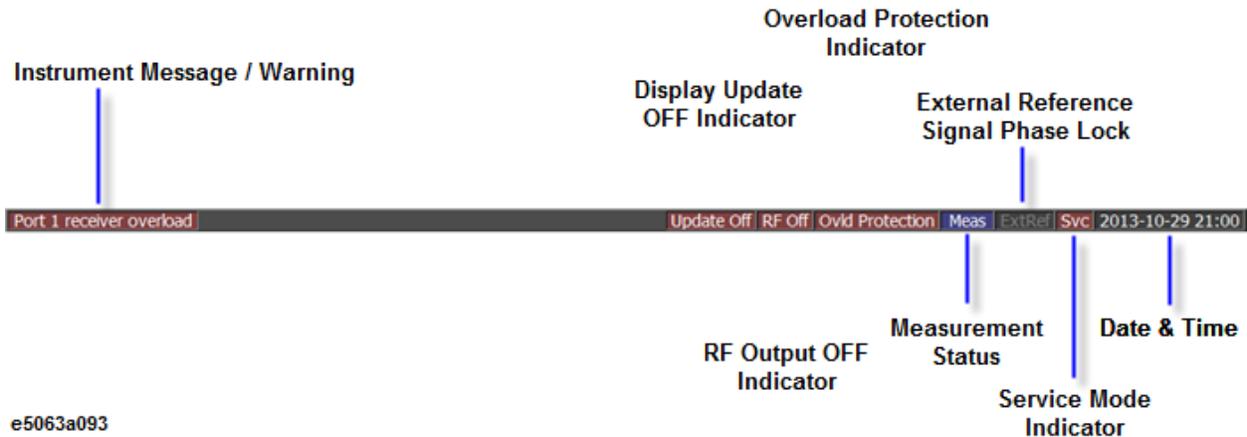
Date and Time

Displays the date and time generated by the internal clock. The display format is as follows:

YYYY-MM-DD HH:MM

Where:

YYYY: Year (AD)
MM: Month
DD: Day
HH: MM: Time (0:00 to 23:59)

- You can turn the date and time display on/off by: **System** > **Misc Setup** > **Clock Setup** > **Show Clock**.

Display Update OFF Indicator

When information update display on the LCD screen is turned off, this indicator is displayed.

External Reference Signal Phase Lock

When the frequency reference signal is input to the Reference Signal Input (REF IN) on the rear panel and the measurement signal of the E5063A is phase-locked to the reference signal, ExtRef is displayed in blue.

Value	Description
ExtRef (displayed in blue)	Measurement signal is phase-locked to the external reference signal.
ExtRef (displayed in gray)	Measurement signal is not phase-locked to the external reference signal.

Instrument Message/Warning

Displays instrument messages and warnings. Instrument messages are displayed in gray and warnings in red.

41

Measurement Status

Displays the measurement status of the E5063A.

Value	Description
Setup	Setup for measurement in progress
Hold	Measurement on hold (idling)
Init	Measurement being initialized
Man	The trigger source is set to "Manual" and waiting for trigger.
Ext	The trigger source is set to "External" and waiting for trigger.
Bus	The trigger source is set to "Bus" and waiting for trigger.
Meas	A measurement is in progress.

Overload Protection Indicator

Displays when the overload is detected.

RF Output OFF Indicator

When RF signal output is turned off, this indicator is displayed.

Service Mode Indicator

Indicates the service mode status. The service mode indicator is displayed when E5063A enters the following state.

Value	Description
SVC (displayed in blue)	The E5063A is in service mode, which is used for self-diagnosis and repair of the E5063A or 8 term calibration mode. Therefore, measurement performance will not be guaranteed according to the specifications. If, under normal use, the system remains in the service mode and does not return to normal operating mode, there is a possibility that the instrument is out of order.
SVC (displayed in red)	An abnormal condition has been detected inside the E5063A. The unit may be damaged. Notify the Customer Contact listed at the end of this manual or the distributor from whom the unit was purchased.

Menu Bar

By using the mouse and keyboard to manipulate the menu bar, you can perform interface operations that are equivalent to those of the keys in the **ACTIVE CH/TRACE** Block, **RESPONSE** Block, **STIMULUS** Block, **MKR/ANALYSIS** Block, and **INSTR STATE** Block on the front panel of the E5063A. The menus on the menu bar correspond to the key blocks, and their submenus to the hardkeys inside the key blocks.

Softkey Menu Bar

A group of keys on the screen called by the softkeys and menu bars. You can manipulate these keys by using the **NAVIGATION** Block keys on the front panel, the mouse, or the keyboard. You can perform manipulations by directly touching the screen with your finger instead of using a mouse.

Softkey Menu Bar
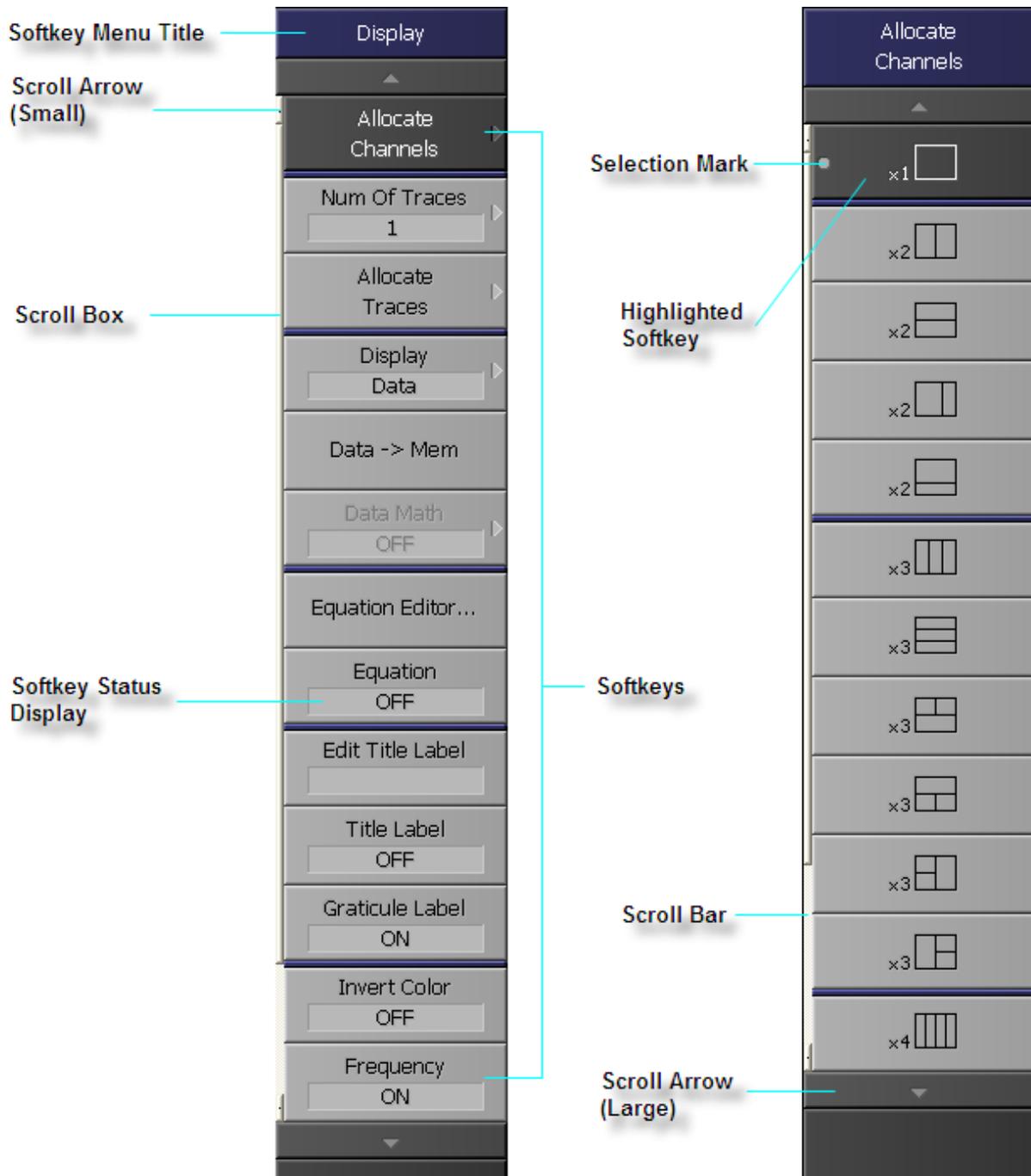
NOTE   To manipulate a menu bar, it has to be selected as the object to manipulate (with the focus placed on it). When the focus is placed on a menu bar, the menu title area at the top is displayed in blue. Pressing or clicking on **Foc** Key in the ENTRY Block enables you to move the focus to the desired object.

Highlighted Softkey

Pressing ⊙ and **Enter** key on the front panel or pressing **Enter** key on the keyboard causes the highlighted (selected) softkey to be executed. You can change which softkey in the menu is highlighted by turning ⊙ or pressing ⊙⊙ on the front panel or by pressing ⬆⬇ on the keyboard. Pressing the ⬅ key on the front panel or the ⬅ key on the keyboard brings up the upper level softkey menu, and pressing the ➡ key on the front panel or the ➡ key on the keyboard brings up the lower level softkey menu.

Scroll Arrow (Large)

When the softkeys in a menu overflow the screen, use this key to enable you to scroll the menu page by page. Both upward and downward scroll arrows are available. Use the mouse to manipulate these buttons.

Scroll Arrow (Small)

Using this button, you can scroll the menu one softkey at a time. Both upward and downward scroll arrows are available. Use the mouse to manipulate these buttons.

Scroll Bar

When the softkeys in a menu overflow the screen, clicking on the blank part of the scroll bar enables you to scroll the softkey menu up or down.

Scroll Box

You can scroll the softkey menu up or down by using the mouse to select and drag the scroll box (pressing the button on the object to be moved and then releasing the button at the desired location). The length and position of the scroll box indicate the length and position of the currently displayed part of the softkey menu relative to the entire menu.

Selection Mark

Shows which softkey function is currently selected.

Softkeys

These are the actual keys you would use to perform setup. A ▶ displayed to the right of a softkey indicates that pressing that softkey displays the lower layer of softkeys.

Softkey Menu Title

The title of the softkey menu is displayed here. Double-clicking on this part of the menu bar displays the top layer of softkeys.

Softkey Status Display

Displays a softkey's setup status.

Some softkeys like Start/Stop/Center/Span include the frequency information. You can hide the frequency information in order to ensure its confidentiality or for other reasons. See [Hiding Softkey's Frequency Information](#).

Three Methods of Operation

You can operate the E5063A using one of three operating methods: using keys on the front panel, using a mouse and keyboard, and using the touch screen. This section illustrates these three operating methods through the example in which the channel window layout is set to four-channel display as shown in the following figure.

In the next section and those following it, a series of operations is expressed as follows:

Display > **Allocate Traces** >
Four-Trace Display

e5063a083
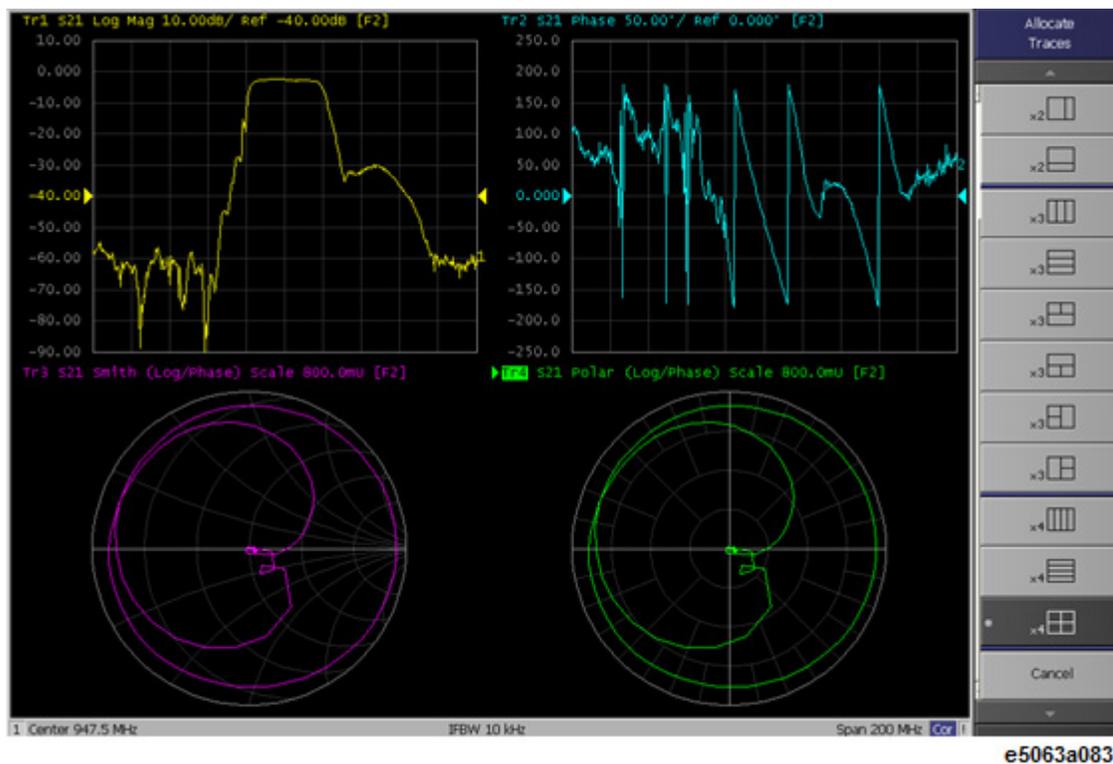
Operating Method Using Keys

    1. Press the **DISPLAY** key in the **RESPONSE** Block.
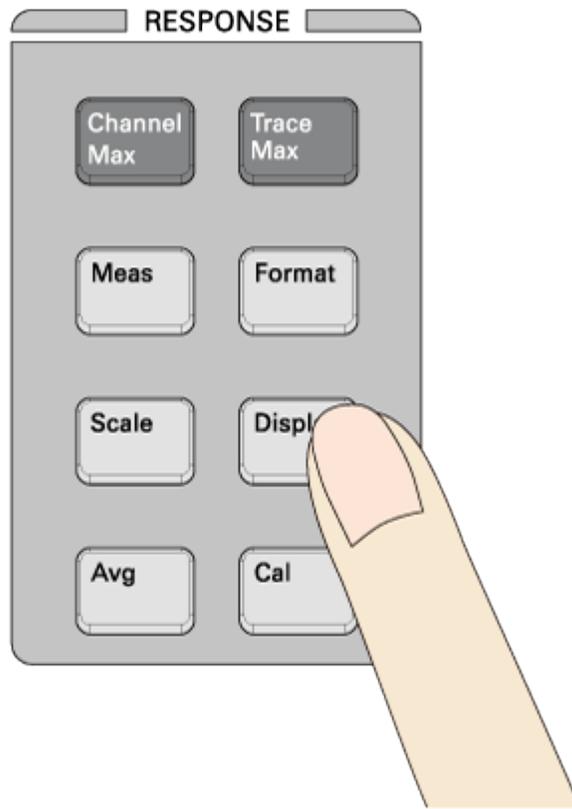
e5071c307

2. Press ⊕ or ⊕ key to move the cursor to the **Allocate Traces**, then press **Enter** or (⊕) key.

e5071c002

3. Press ⊕ or ⊕ key to move the cursor to [×4 ▦], then press **Enter** key.

Operation Method Using a Mouse

1. From the **Response** menu, press **Display** key.

e5071c051

2. Click **Allocate Traces**.

3. Click

Operation Method Using the Touch Screen

1. Press **Display** key after presetting the E5063A (Executing **Preset** key).
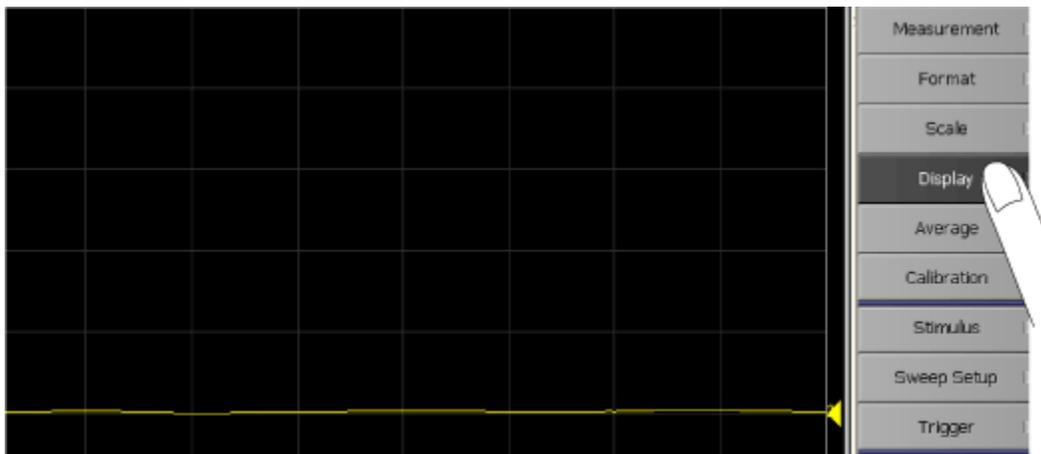
e5063a084

2. Click/Press **Allocate Traces.**

3. Click/Press any desired setting.

49

Basic Measurement Procedures

This section describes the basic measurement procedure using the E5063A and presents an example of the transmission measurement of a bandpass filter, for better understanding of how to use the E5063A.

Basic Measurement Flow

1. **Determining measurement conditions**
- Initializing Parameters
- Select:
    - Measurement Parameter
    - Data format
    - Sweep Type and range
    - Power level
    - IF bandwidth
2. **Calibration**
- Select calibration kit
- Make a calibration
3. **Connecting the Device Under Test (DUT)**
- Connect DUT
- Adjust the scale
4. **Analyzing measurement results**
- Analysis using markers
5. **Outputting measurement results**
- Store measurement result into a file

Measurement Example of a Bandpass Filter

This section describes how to measure the transmission characteristics of a 947.5 MHz bandpass filter. The measurement conditions for this measurement example are those suitable for a 947.5 MHz bandpass filter. To measure another device under test (DUT), change the measurement conditions to suit the particular DUT.

STEP 1. Determining Measurement Conditions

1. Preset the E5063A.

   Preset > **OK**

2. Set the S-parameter to S21.

   Meas > **S21**

3. When measuring the reverse transmission characteristics, set the S-parameter to S12.

3. Set the data format to the log magnitude format

   Format > **Log Mag**

4. Set the center frequency to the bandpass filter center frequency. Next, specify the span frequency, which is set to 200 MHz in this measurement example.

   Center > **9** > **4** > **7** > **.** > **5** > **M/m**
   Span > **2** > **0** > **0** > **M/m**

3. When entering the frequency unit using the keyboard, type "G" for GHz, "M" for MHz, and "k" for kHz.

5. Specify the number of measurement points per sweep. The number of measurement points in this measurement example is set to 401.

   Sweep Setup > **Points** > **4** > **0** > **1** > **x1**

6. Specify the power level of the signal source. The power level in this measurement example is set to -10 dBm.

   Sweep Setup > **Power** > **+/-** > **1** > **0** > **x1**

7. Specify the IF bandwidth of the receiver as necessary. In this measurement example, the IF bandwidth is set to 10 kHz because of the need to lower the noise floor.

Avg > **IF Bandwidth** > **1** > **0** > **k/m**

STEP 2. Calibration

To turn the error correction ON, set the calibration type to the full 2-port calibration and measure the calibration data.

For details about calibration, see Calibration.

1. Select the calibration kit suitable for the measurement cable. In this measurement example, Calibration Kit 85032F is selected.

   Cal > **Cal Kit** > **85032F**

2. Set the calibration type to the full 2-port calibration using the test port 1 and 2.

   Cal > **Calibrate** > **2-Port Cal**

3. Connect the OPEN standard (included in the calibration kit) to the other end of the measurement cable that is connected to the test port 1 as shown in the following figure, and measure the open calibration data at the test port 1. After measuring the open calibration data, a checkmark ☑ is displayed to the left of the **Port 1 Open** menu.

   Cal > **Calibrate** > **2-Port Cal** > **Reflection** > **Port1 Open**

   In the same way, measure the calibration data for the SHORT/LOAD standards at the test port 1.

   Connecting the OPEN/SHORT/LOAD standards
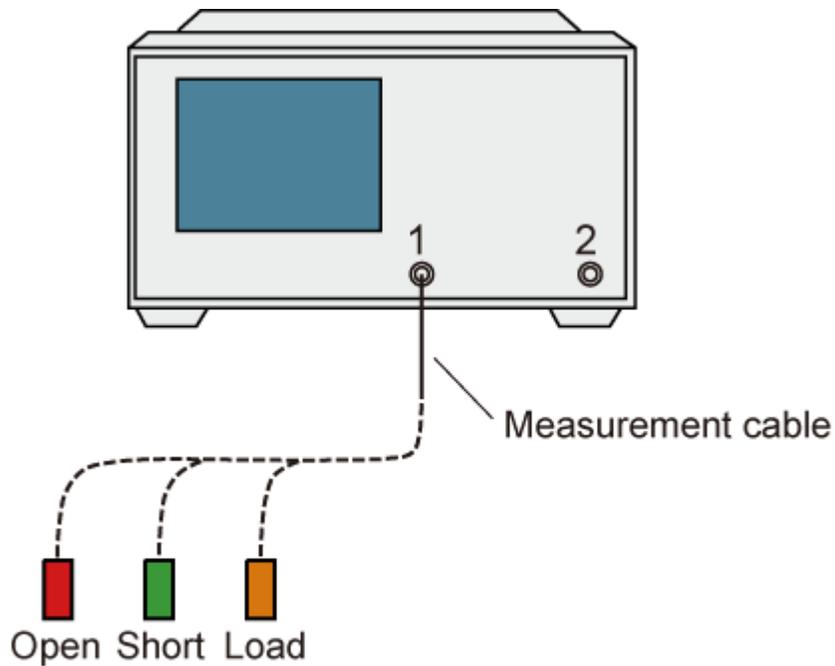
e5072a082.png

4. In the same way as described above, measure the calibration data for the OPEN/SHORT/LOAD standards at the test port 2.

5. Connect the measurement cables as shown in the figure below, and measure the thru calibration. After measuring the thru calibration data, a checkmark ☑ is displayed to the left of the **Port 1-2 Thru** button.

   Cal > **Calibrate** > **2-Port Cal** > **Transmission** > **Port 1-2 Thru**

   Making the through calibration
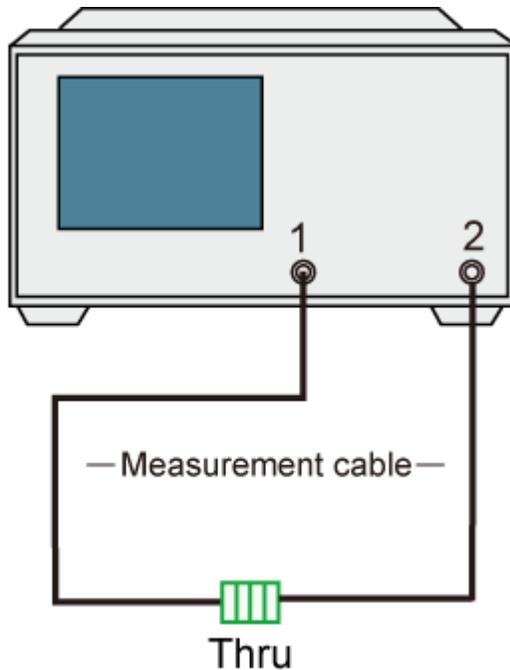
e5072a084.png

6. Set the full 2-port calibration measurement to DONE. The calibration factor is calculated based on the calibration data acquired, and the error correction is turned ON.

   Cal > **Calibrate** > **2-Port Cal** > **Done**

7. Select the type in which the data is to be saved before saving the calibration factor (calculated based on the calibration data).

   Save/Recall > **Save Type** > **State & Cal**

8. Store the calibration file to the disk of the E5063A. The symbol "X" appearing in the operations below represent the assigned numbers to be used when the file is saved.

   Save/Recall > **Save State** > **State 0X**

STEP 3. Connecting the Device Under Test (DUT)

1. Connect to the DUT to the E5063A. (See the below figure)

   Connecting the DUT
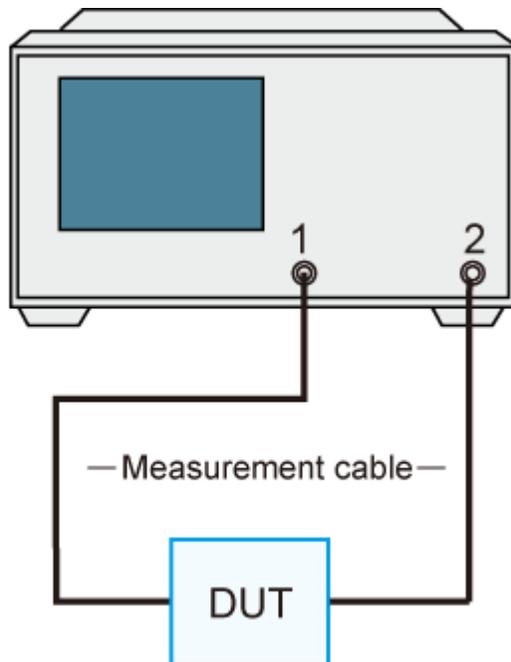
e5072a085.png

2. Set the appropriate scale by executing the auto scale. (See the below figure)

Scale > **Auto Scale**

You can also adjust the scale by entering arbitrary values in the **Scale/Div** button, **Reference Position** button, and **Reference Value**.

S21 trace after executing the auto scale

e5063a094
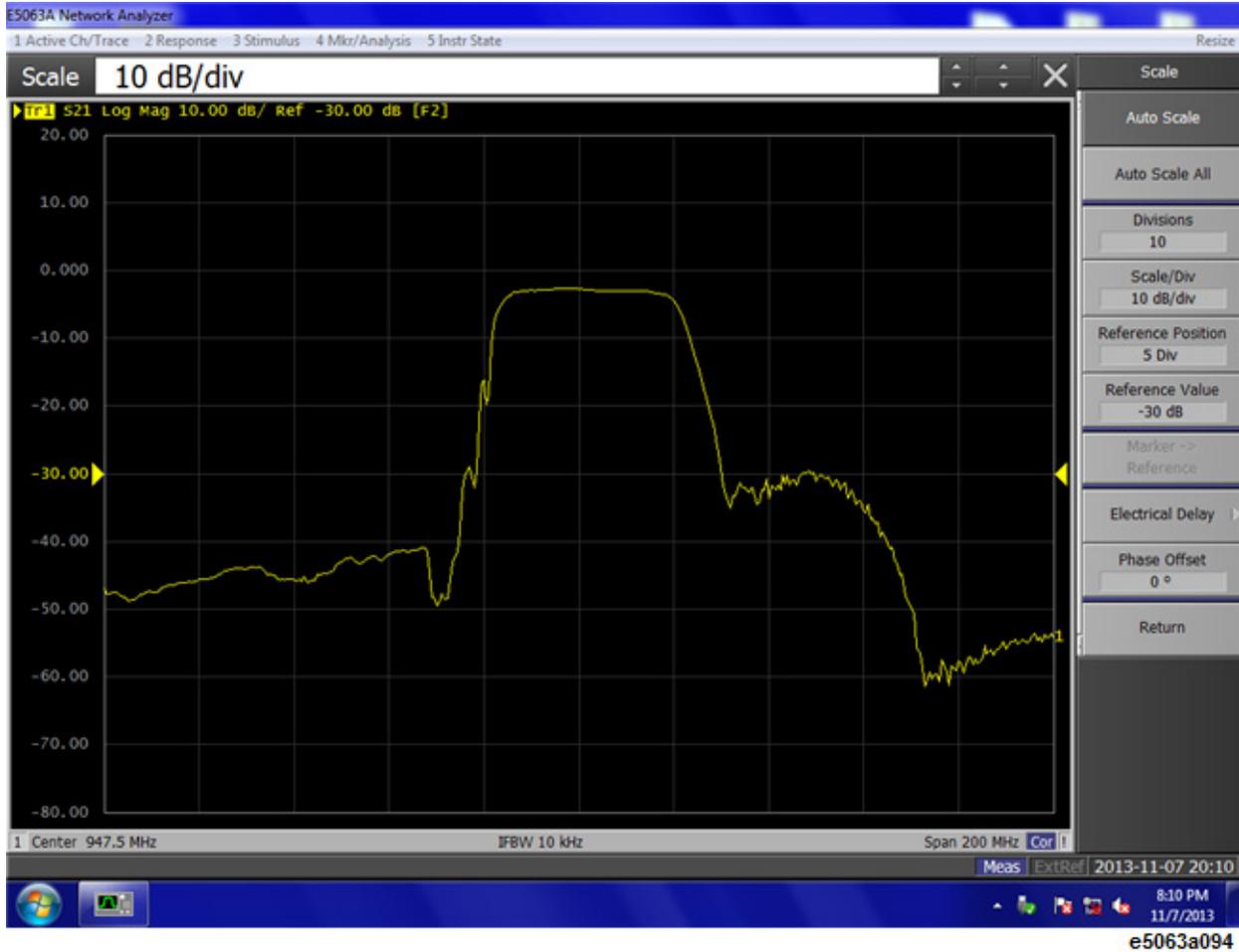
STEP 4. Analyzing Measurement Results

This section describes how to use the marker function to read out important parameters for the transmission measurement of the bandpass filter (insertion loss, -3 dB bandwidth).

Measuring the Insertion Loss

1. Display a marker.

   Marker > **Marker 1**

2. Using one of the following methods to move the marker to the center frequency of the bandpass filter.

   - On the entry bar, press **9**> **4** > **7** > **.** > **5** > **M/m**

   - Turn the rotary knob ⊙ on the front panel to set it to the center frequency (947.5 MHz).

3. Read the marker value displayed as shown in the figure below. In this example, the response value denotes the insertion loss.

Measuring an Insertion Loss

e5063a082
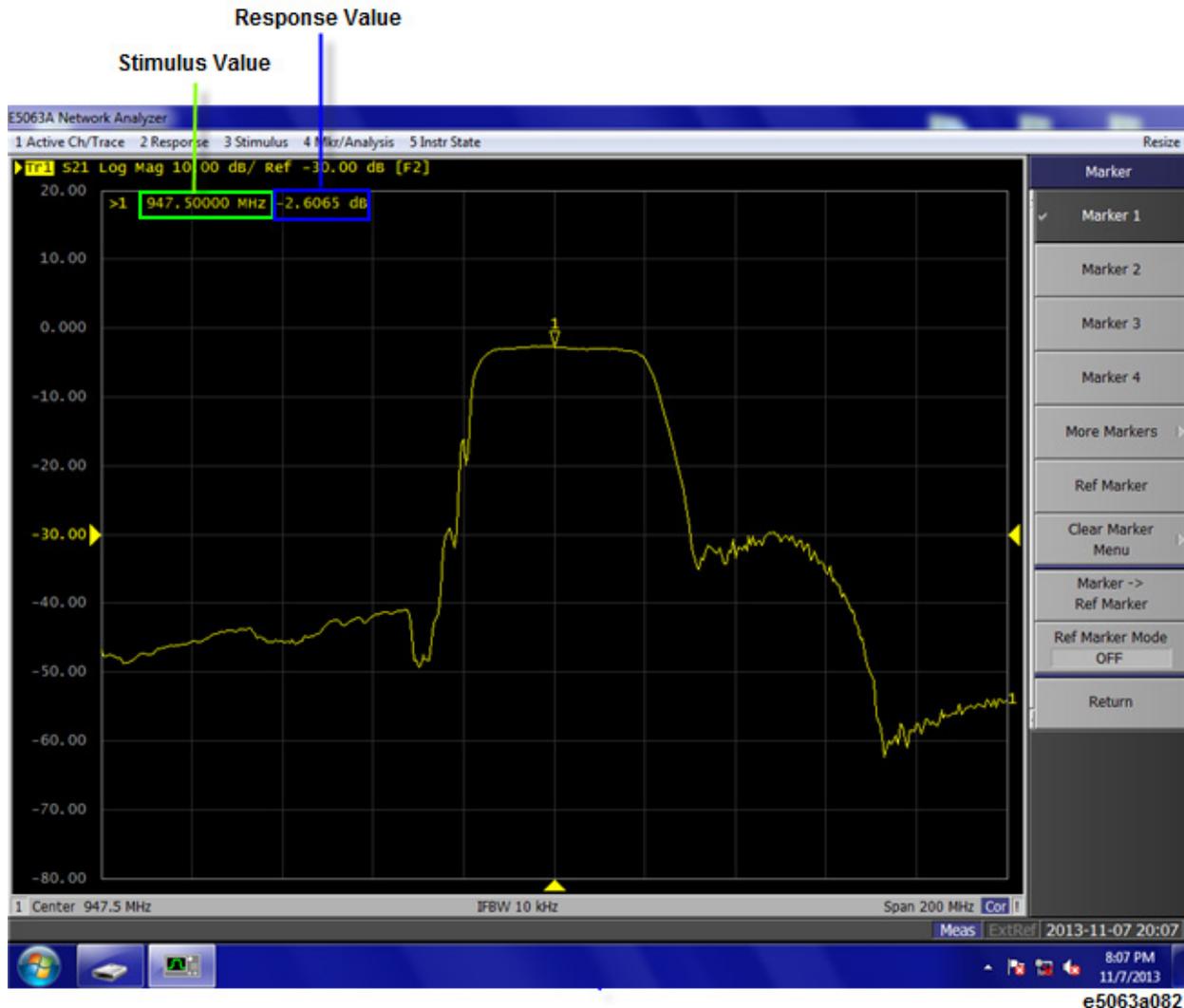
Measuring the -3 dB Bandwidth

Using the marker bandwidth search function, the bandwidth, center frequency between two cutoff frequency points, Q value, and insertion loss are all read out. These parameters are described in the following table.

- If the two cutoff frequency points are not found, all data items except the insertion loss revert to zero.

Parameter	Description
BW (Bandwidth)	Stimulus width between two cutoff frequency points (low and high)
cent (Center Frequency)	Center point between cutoff frequency points (low and high)
low (Left-side Cutoff	The lower frequency of the two cutoff

Frequency)	frequency points
high (Right-side Cutoff Frequency)	The higher frequency of the two cutoff frequency points
Q (Q Value)	Q = cent/BW
loss (Insertion Loss)	The measured value of the active marker.

1. Display a marker.

   Marker > **Marker 1**

2. Using one of the following methods to move the marker to the center frequency of the bandpass filter.

   - On the entry bar, press **9** > **4** > **7** > **.** > **5** > **M/m**

   - Turn the rotary knob ⊙ on the front panel to set it to the center frequency (947.5 MHz).

3. Specify the bandwidth definition value that defines the pass band of the filter. In this measurement example, it is set to -3 dB.

   Marker Search > **Bandwidth Value** > **+/-** > **3** > **x 1**

4. Set the bandwidth search function ON.

   Marker Search > **Bandwidth**

5. The bandwidth data items (BW, cent, low, high, Q, loss) is displayed. (See the following figure.)

Measuring the -3 dB Bandwidth

e5063a080
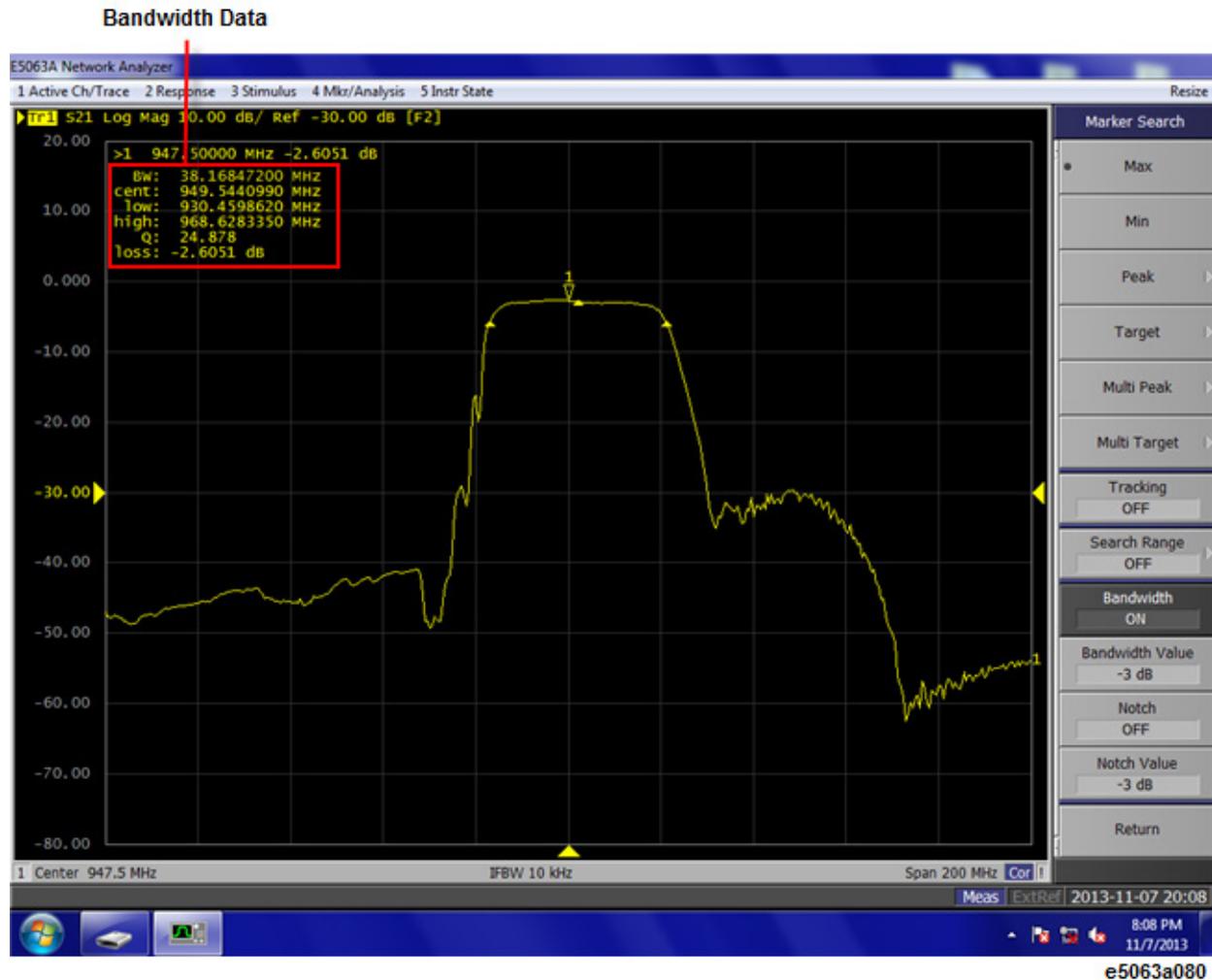
STEP 5. Outputting Measurement Results (Save)

You can save not only the internal data but also the measurement results such as trace data and display screens to the disk.

Saving the Trace Data(in CSV format)

You can save the trace data to the disk of the E5063A in CSV file format (extension: .csv). Since the CSV-formatted data to be saved is a text file, you can analyze the data using Microsoft Excel.

Follow the step below to save the trace data:

Save/Recall > **Save Trace Data**

Saving the Display Screen

You can save the screen displayed on the E5063A to the disk of the E5063A in Windows bitmap file format (extension: .bmp) or Portable Network Graphics format (extension: .png).

Follow the step below to save the display screen:

System > **Dump Screen Image**

- The image on the LCD display memorized in the volatile memory (clipboard) (the image on the LCD display when the **Capture/System** key is pressed) is saved.

Measurement

Measurement

This chapter provides information related to the measurements done through E5063A.

- Setting Measurement Conditions
- Calibration
- Making Measurements
- Analyzing Data
- Fixture Simulator
- Time Domain Analysis
- Outputting Data
- Optimizing Measurements

Setting Measurement Conditions

Setting Measurement Conditions

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

Initializing Parameters

The E5063A 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	E5063A 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 the System Z0

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

1. Press **Cal** key.

2. Click **Set Z0**, then input the system Z_0.

Other topics about Setting Measurement Conditions

Setting Channels and Traces

- [Overview of Channel and Trace](#)
- [Number of Channels Traces](#)
- [Setting Channel Display (Layout of Channels)](#)
- [Setting Trace Display](#)
- [Active Channel](#)

Other topics about Setting Measurement Conditions

Overview of Channel and Trace

The E5063A allows you to setup multiple channels to perform measurement under different stimulus conditions.

As multiple traces (measurement parameters) 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, you need to set the measurement conditions and execute calibration for each channel you use for 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 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		x		**Sweep Setup** > **Pow**
Sweep time/Sweep delay time		x		**Sweep Setup** > **Swe Time, Sweep Delay**
Number of points		x		**Sweep Setup** > **Poi**
Segment sweep		x		**Sweep Setup** > **Swe Type, Edit Segment**

65

				Table, Segment Dis
Trigger Settings				
Trigger mode		x		**Trigger** > **Hold/Sing** **Continuous**
	x			**Hold All Channels/Continuo Disp Channels**
Trigger source, Trigger Event, Trigger Scope	x			**Trigger** > **Trigger Source, Trig Event**, **Trigger Scop**
Trigger	x	_		**Trigger** > **Restart/Trigger**
Ext Trigger Input, Trigger Delay	x	_		**Trigger** > **Ext Trig I Trigger Delay**
Ext Trigger Output, Polarity, Position, Pulse Width	x	_		**Trigger** > **Ext Trig Output, Polarity, Position**, **Pulse Wid**
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/D** > **Mem/ Data Math**
Equation Editor		x		**Display** > **Equation Editor/Equation (ON/OFF)**
Window title		x		**Display** > **Edit Title Label/ Title Label (ON/OFF**
Graticule label in rectangular form		x		**Display** > **Graticule Label (ON/OFF)**
Color inversion	x			**Display** > **Invert Co**
Frequency display	x			**Display** > **Frequenc (ON/OFF)**

Display update	x			**Display** > **Update (ON/OFF)**
Averaging			x	**Avg** > **Averaging Restart/ Avg Factor/Averagi (ON/OFF)**
Averaging Trigger	x			**Avg** > **Avg Trigger (ON/OFF)**
Smoothing				x	**Avg** > **Smo Apertur Smoothing (ON/OF**
IF bandwidth, IF BW Auto, IF BW Auto Limit			x		**Avg** > **IF Bandwidth IFBW Auto**, **IFBW A Limit**
Calibration			x		**Cal**
System Impedance	x				**Cal** > **Set Z0**
Marker				x	**Marker**, **Marker Sea Maker Fctn**
Market Table	x				**Maker Fctn** > **Make Table**
Analysis					
Time domain				x	**Analysis** > **Gating**, Transform**
Parameter conversion				x	**Analysis** > **Convers**
Limit test, Ripple Test, Bandwidth Test, Point Limit				x	**Analysis** > **Limit Te Ripple Test**, **Bandw Limit**, **Point Limit**
Saving and recalling data	x				**Save/Recall**
Macro	x				**Macro Setup**, **Macr Menu**, **Macro Run**
System					
Printing/Saving display Screen/Beeper/GPIB settings/Network Settings/Date & Time/Key	x				**System**

Lock/Backlight/Firmware Revision/Service menu				
Preset	x			**Preset**

Number of Channels/Traces

The maximum number of channels are 12 and the number of traces are 4. The maximum number of points is 10001.

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 to 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.

- 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** > **Allocate Channels**.
2. Press the desired softkey to select the window layout.

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 ![×3 layout icon] as the graph layout and set the number of traces to 4, graph 1 (Gr1 in Graph layout) display traces 1 and 4, respectively, by superimposing, and graph 2 (Gr2 in Graph layout) and graph 3 (Gr3 in Graph layout) displays trace 2 and trace 3 as shown in the figure below.

e5063a111
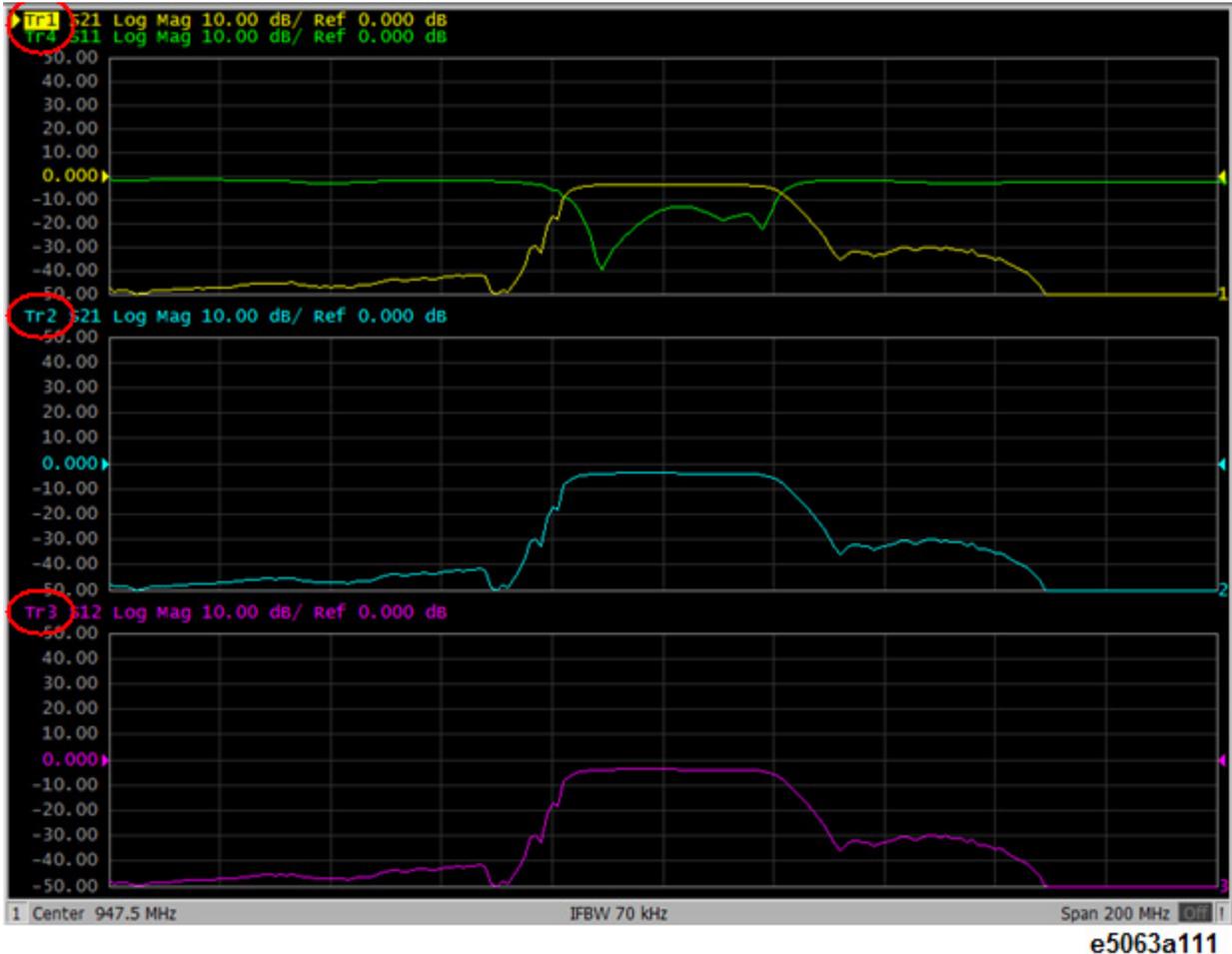
The procedure for setting the graph layout is as follows:

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
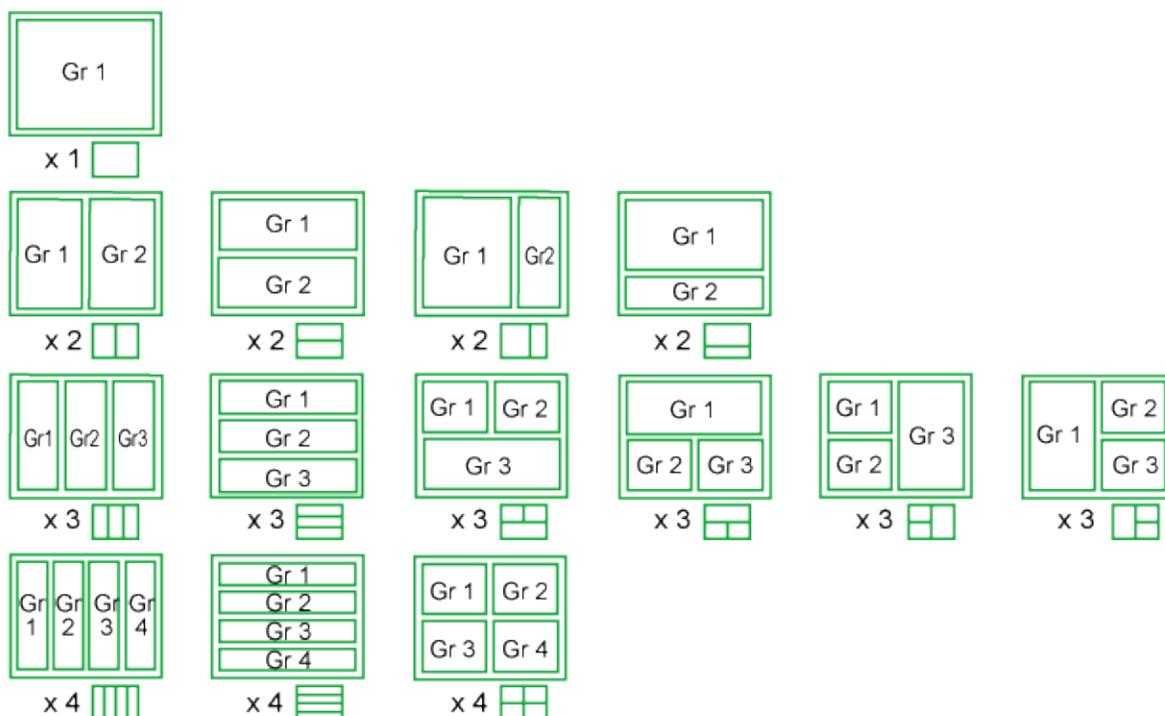
e5061b026

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, Tr2) 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

70

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.

Selecting Measurement Parameters

The E5063A allows users to evaluate the DUT (device under test) characteristics by using the following measurement parameters.

- S-parameters

Other topics about Setting Measurement Conditions

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.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace.

2. Select a softkey that corresponds to the desired measurement parameter. Press **Meas** > **S11 | S21 | S12 | S22**.

Setting Stimulus Conditions

You can set the stimulus condition for each channel independently.

- Setting Sweep Type
- Setting Sweep Range
- Enable Stimulus Signal Output
- Setting Power Level
- Setting Number of Measurement Points
- Setting Sweep Delay and Sweep Time
- Measuring in Time Series (Time Sweep)

Other topics about Setting Measurement Conditions

Setting Sweep Type

You can select the sweep type from the following 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).

The procedure for selecting the sweep type is as follows:

1. Press **Channel Next** (or **Channel Prev**) 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. The time sweep is shown in Measuring in Time Series. This allows you to display the measurement parameter versus time.

Setting 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.
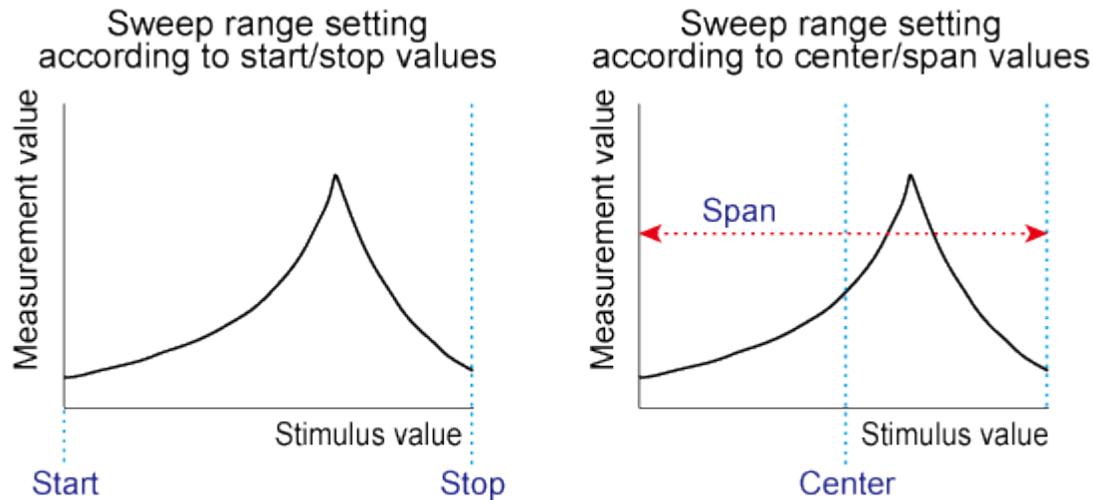
e5071c443

Setting the Sweep Range with the Lowest and Highest Values

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which 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** (or **Channel Prev**) to select the channel of which sweep range will be set.

2. Click **Center**, then input the center value.

3. Click **Span**, then input the span value.

Setting Sweep Range Using the Marker

1. In the channel window of which 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**.

3. Click the **softkey** that corresponds to each value.

- 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 range.

e5071c395
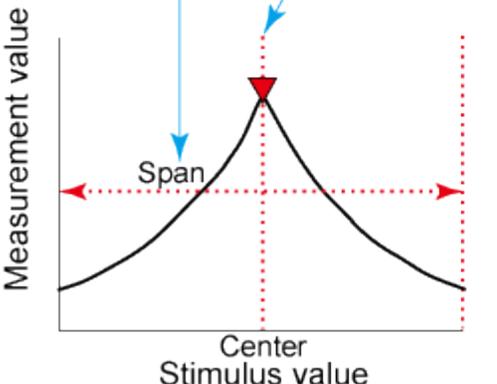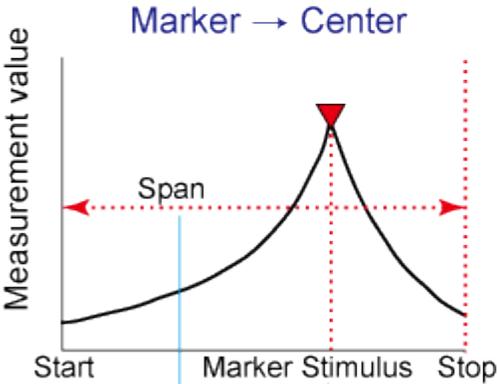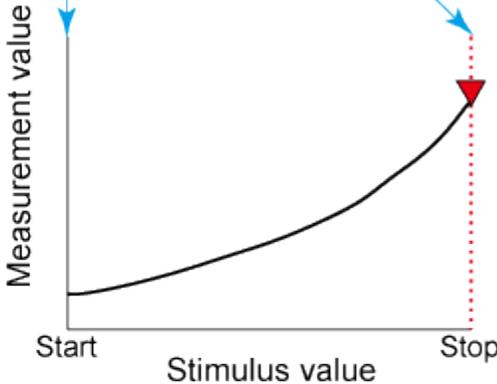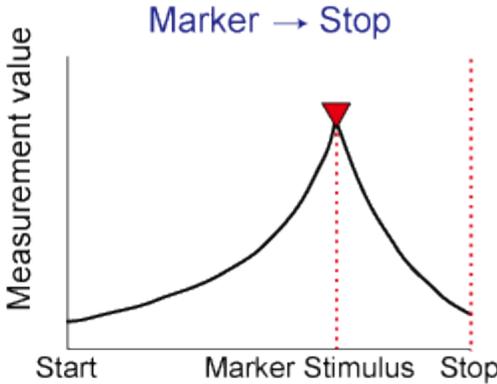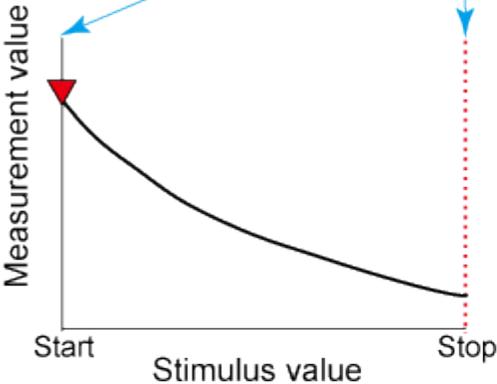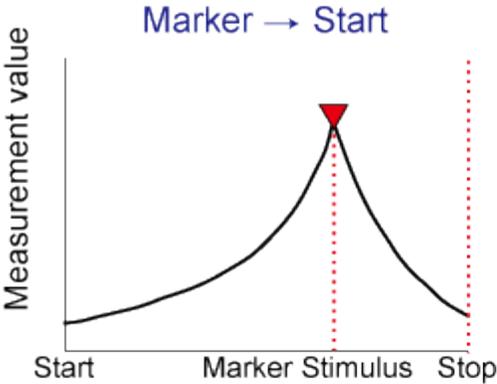
Enable Stimulus Signal Output

You can turn on/off the stimulus signal output, but this prevents you from performing the measurement. Therefore, normally this feature will not be used. This feature is mainly used to turn the output ON back after it has been turned OFF by the power trip feature (**System** > **Service Menu** > **Overload Detect**).

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

1. Press **Sweep Setup**.
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 of which 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** (or **Channel Prev**) 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 the ports.

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

5. 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. Follow the procedure below according to the 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)

. Press **Port Power**, then click the softkey corresponding to each port (**Port 1 Power**, **Port 2 Power**)

a. Enter the power level.

Setting Number of Measurement Points

> NOTE The number of points is the number of data items collected in one sweep. It can be set to any number from 2 to 10001 for each channel independently.

- To obtain a higher trace resolution against the stimulus value, choose a larger value for the 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.

1. Press **Channel Next** (or **Channel Prev**) to select the desired channel.

2. Press **Sweep Setup** key**.**

3. Click **Points**, then input the desired number of points.

Setting Sweep Delay and Sweep Time

Sweep time is the time taken 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.

Sweep Time Mode	Description
Manual	In this mode, the sweep time is set manually. Once the sweep time is set, changes in the measurement conditions do 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 is reset to the shortest time within the conditions. If the sweep time exceeds the analyzer's upper sweep time limit, the sweep time is reset to the longest time within the conditions.
Auto (Default)	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.

Sweep delay is time before starting a sweep for each stimulus (source) port. Sweep time does not include the sweep delay.
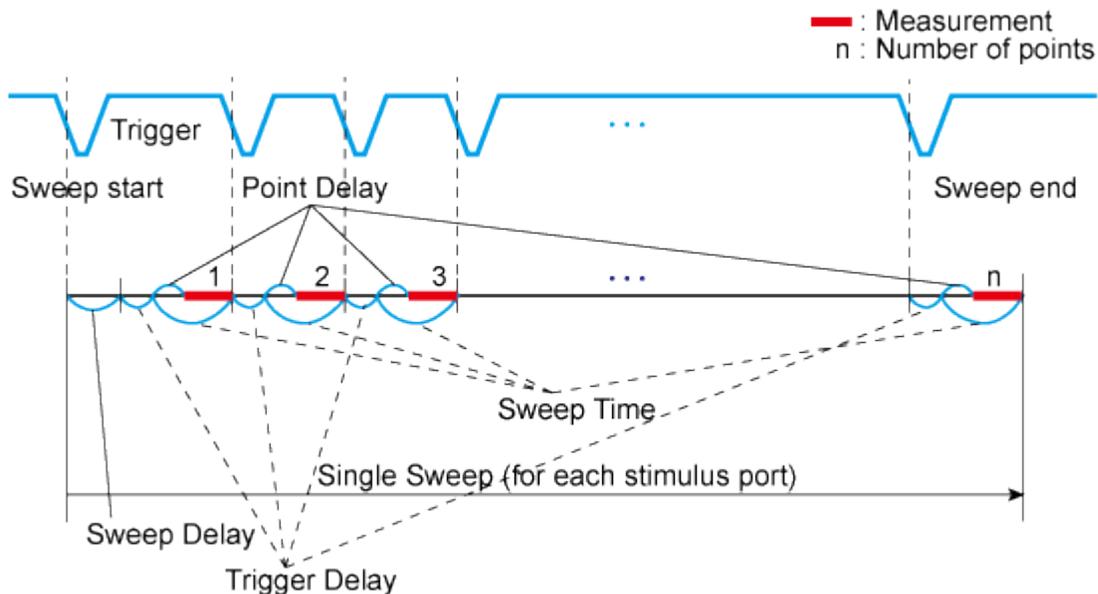
Timing Chart for Sweep

When the trigger mode is set at "On Sweep".
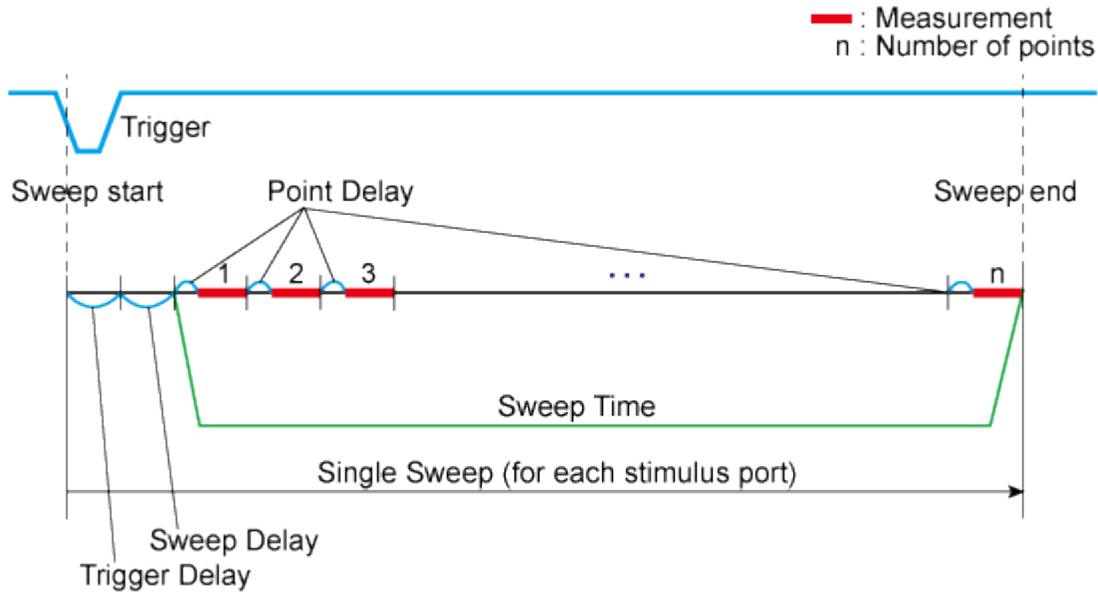
e5061b034

When the trigger mode is set at "On Point".

e5061b035

NOTE   Trigger delay is available when the trigger source is set at external.
Sweep time is the total of point delay and measurement time in all measurement points when the trigger mode set at "On Point".

Setting Sweep Delay Time

1. Press **Channel Next** (or **Channel Prev**) to select the desired channel.

2. Press **Sweep Setup** > **Sweep Delay**.

3. Using the **ENTRY** block keys on the front panel, input the desired sweep delay time (in seconds).

Setting Up Sweep Time (Manual Sweep Time Mode)

1. Press **Channel Next** (or **Channel Prev**) to select the desired channel.

2. Press **Sweep Setup** > **Sweep Time Auto** to turn OFF.

3. The softkey named **Sweep Time** is activated. Press **Sweep Time**.

4. Using the **ENTRY** block keys on the front panel, input the desired sweep time (in seconds).

Measuring in Time-Series (Time Sweep)

The following procedure allows you to time sweep and the measurement parameter is displayed versus time.

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which sweep range will be set.

2. Press **Span** > **0** > **x1** to set the span value to 0 (zero span).

3. Press **Center**, then input the desired value.

4. Press **Sweep Setup** > **Sweep Time Auto** to turn OFF.

5. Press **Sweep Time**, then input the duration of the sweep which is displayed on X-axis.

6. Press **Marker** to display the marker 1. The time at the marker shows as the marker position value at the upper left corner on the screen.

Selecting a Data Format

The E5063A allows you to display the 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. Several different formats are available depending on the selection of data for the Y-axis.

Rectangular display format
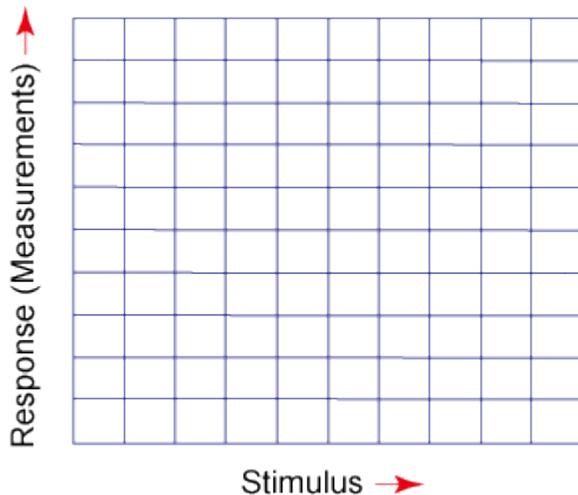
e5071c451

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°	Degrees (°)	Measurement of deviation from linear

	to +180°)		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	$\dfrac{1+\rho}{1-\rho}$ (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 the 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 to display the marker response values.

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

- Real and imaginary parts

Polar format
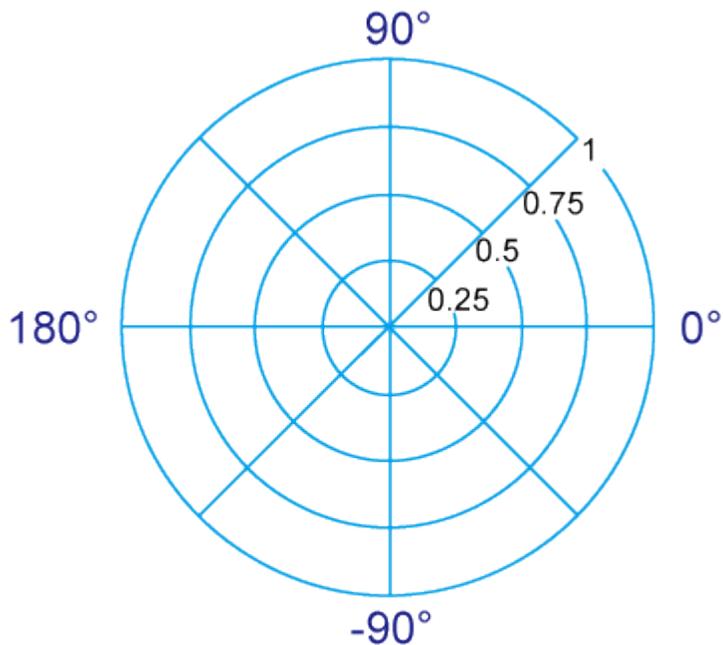
e5071c382

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 to display 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)

Smith chart format
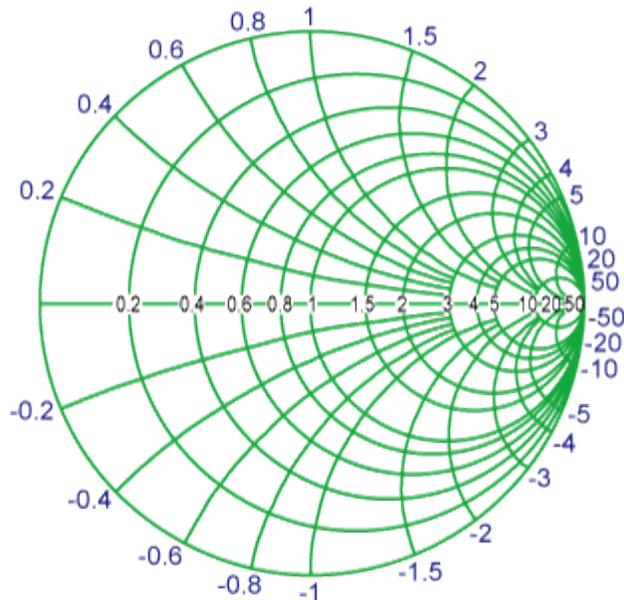
e5071c383

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 /**	Selects the Smith chart format (with the real and

Imag	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

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.

Single Trace Auto Scale

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

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace of 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** (or **Channel Prev**) to select the channel of which the auto scale function will be performed.

2. Press **Scale** > **Auto Scale All**.

Manual Scale Adjustment

Manual scale adjustment on the 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. The reference position is indicated by horizontal triangles at both edge of the reference position line.
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.

Rectangular display format
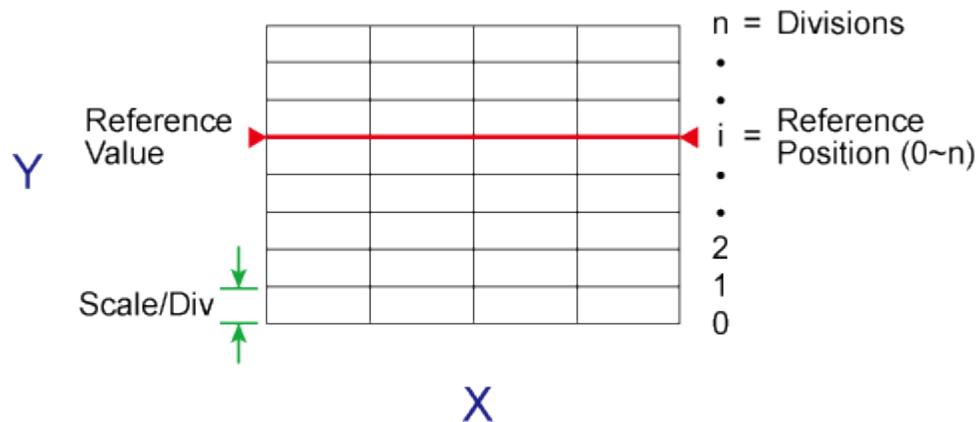
e5071c385

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace of which scale features will be adjusted.

2. Press **Scale**.

3. Select the **softkey** that corresponds to the particular feature that needs to be adjusted.

   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).

Polar format   Smith chart format
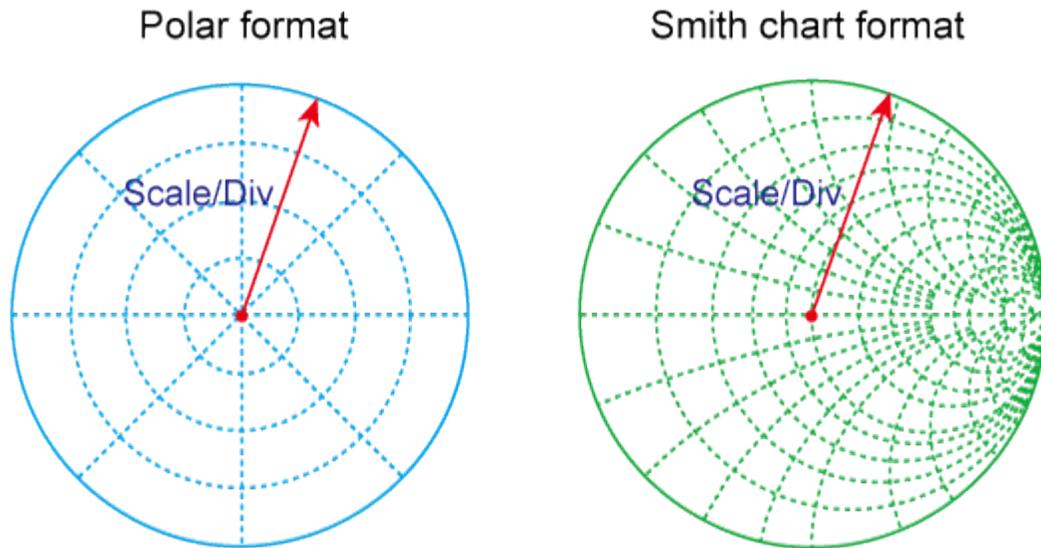
e5071c384

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace for which the scale will be adjusted.

2. Press **Scale**.

3. Click **Scale/Div**, then input the displacement of the outermost circle.

Setting Reference Line Value using 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.

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**.

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 is 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.

Maximizing a window

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which 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 of which 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 of 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.
	Save/Recall > **Save Trace Data** and **Save SnP** are inactive.
	This can be turned OFF by the Security Level menu.
High	Hides the frequency information with a series of asterisks.
	Save/Recall > **Save Trace Data** and **Save SnP** are inactive.
	This cannot be turned OFF by the Security Level menu.
	Resetting to OFF is only possible by executing Preset or Recall.

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 results for future reference.

Labeling a window

1. Press **Channel Next** (or **Channel Prev**) to select the channel to be labeled.

2. Press **Display** > **Edit Title Label**, and the title label input dialog box appears.

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 appears within a frame at the top of the channel window.

e5063a085
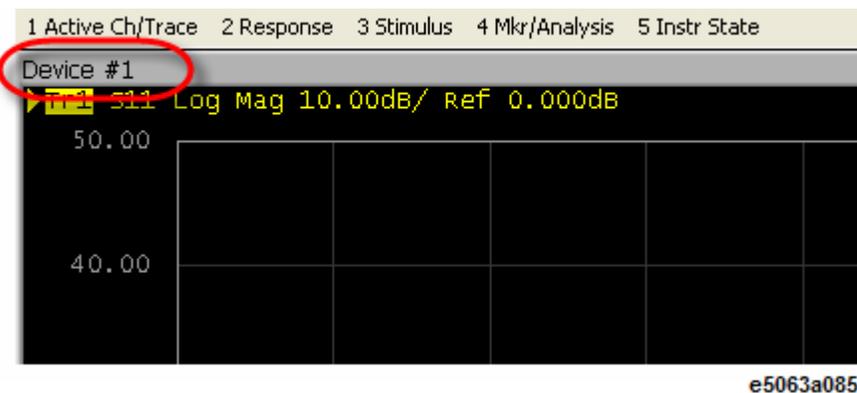
Setting display colors

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 inverting the default settings to 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 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

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, total of 216 colors 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 magenta	5	3	5	**Magenta**	5	0	5	**Dark Magenta**	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 of which you want to set the display color.
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 E5063A 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 the related softkey is disabled. For example, when the screen is displayed in full page size, **Maximize** is disabled.

Another option to minimize the E5063A screen is by using the Menu Bar and the procedure is as follows:

1. Press **Display**.
2. Click **Minimize E5063A**.

You can also hide and restore the title bar of the E5063A screen and the procedure is as follows:

1. Press **Display**.
2. Click **E5063A 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](#)

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](#)
- [Clear Calibration](#)

Calibration with ECal (Electronic Calibration)

- [ECal (Electronic Calibration)](#)
- [ECal Driver Installation](#)
- [Calibration Using ECal Module](#)

Advanced Calibration with ECal

- [Improving Calibration Accuracy along with ECal](#)
- [Confidence Check on Calibration Coefficients Using ECal](#)
- [Turning off ECal Auto-detect Function](#)

Advanced Calibration

- [Modifying Calibration Kit Definition](#)
- [Specifying Different Standard for Each Frequency](#)
- [Partial Overwrite Calibration](#)
- [Unknown Thru Calibration](#)
- [Adapter Removal-Insertion](#)
- [2-port TRL Calibration](#)
- [Sliding Offset Load Calibration](#)

Overview

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	Characterist
No calibration	None	None	All parameters	• Low ac • Calibra not re
Response Calibration	• OPEN or SHORT • LOAD	Following 2 error terms: • Reflection Tracking (Er) • Directivity (Ed)	S11(Reflection characteristics at 1 port)	• Mediu accura • Quick calibra • Isolati calibra improv accura reflect measu of a D high r loss
	• THRU • LOAD	Following 2 error terms: • Transmission Tracking (Et) • Isolation (Ex)	S21 (1 direction transmission characteristics at 2 ports)	• Mediu accura • Quick calibra • Isolati calibra improv accura transn measu of a d with h inserti
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 measu with t highes degre accura • Quick calibra with l chanc

				operat…
	• OPEN • SHORT • LOAD			• Highly accura port measu…
Enhanced Response Calibration	Ecal module (2-port/4-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 accura port measu… (highe respon calibra • Quick calibra with l… chanc… operat…
	• OPEN • SHORT • LOAD • THRU			• Highly accura port measu… (highe respon calibra
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 accura port measu… • Quick calibra with l… chanc… operat…
	• OPEN • SHORT • LOAD • THRU			• Highly accura port measu…
2-Port TRL Calibration	• Reflection (OPEN or SHORT) • THRU • LINE	Following 12 error terms: • Directivity (Ed1,Ed2) • Source Match (Es1,Es2)	S11,S21,S12,S22 (All S-parameters at 2 ports)	• Highly accura port measu… • Effecti non-c…

	• MATCH	• Load Match (El1,El2)		device measu
		• Transmission Tracking (Et21,Et12)		
		• Reflection Tracking (Er1,Er2)		

Other topics about Calibration

Measurement Errors and their Characteristics

- [Overview](#)
- [Drift Errors](#)
- [Random Errors](#)
- [Systematic Errors](#)

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 E5063A 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)

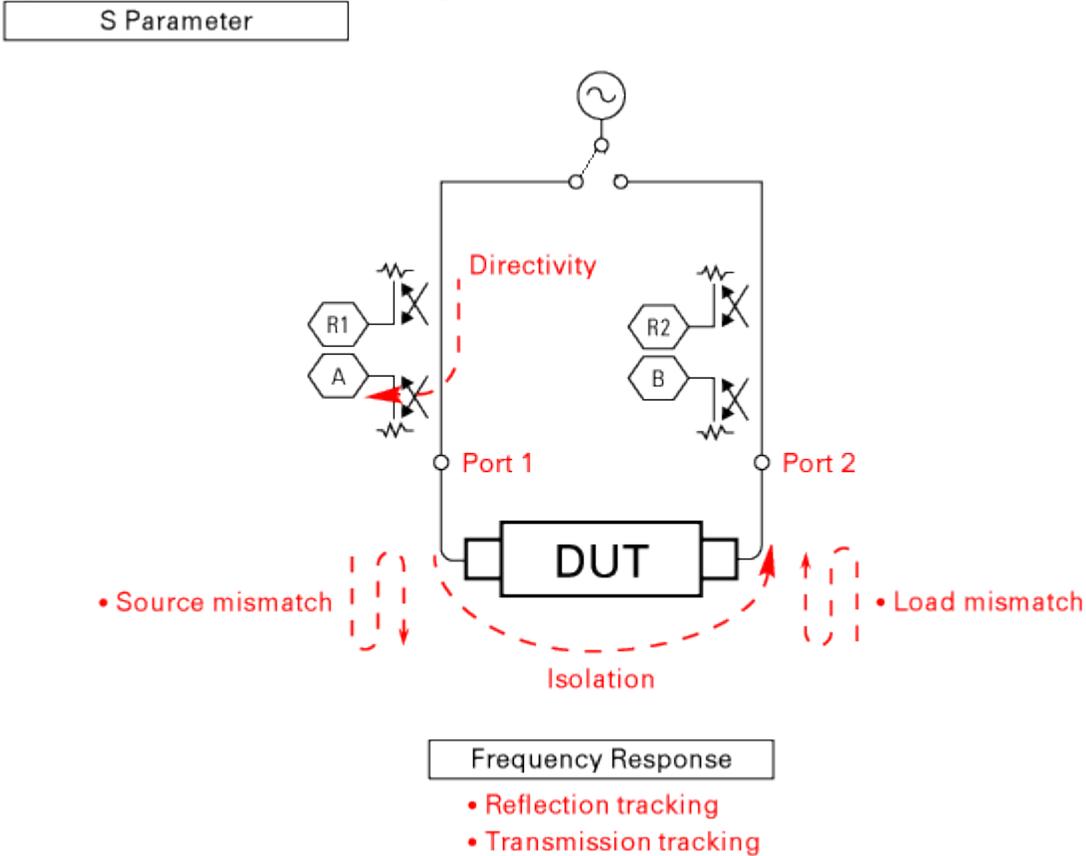Errors caused by reflections in the measuring system:

- Source match
- Load match

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

- Reflection tracking
- Transmission tracking

The E5063A 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.

E5063A port architecture and systematic

errors e5061b032

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 E5063A is the number of stimulus ports you use.

Ed1	Directivity error of port 1
Ed2	Directivity error of port 2

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 E5063A is the total number of combinations of stimulus ports and response ports.

Ex21	Isolation error when port 1 is a stimulus port
Ex12	Isolation error when port 2 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 E5063A is equivalent to the number of stimulus ports you use.

Es1	Source match error of port 1
Es2	Source match error of port 2

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 E5063A is the total number of combinations of stimulus ports and response ports.

El21	Load match error when port 1 is a stimulus port
El12	Load match error when port 2 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 E5063A is simply the number of stimulus ports you use.

Er1	Reflection tracking error of port 1
Er2	Reflection tracking error of port 2

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 E5063A is the total number of combinations of stimulus ports and response ports.

| Et21 | Transmission tracking error when port 1 is a stimulus port |
| Et12 | Transmission tracking error when port 2 is a stimulus port |

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, or sweep type is different from that when the calibration was executed.)
C! (displayed in blue)	Error correction: ON (Extrapolation is being executed).

An asterisk (*) at 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 actually executed error correction for each trace at the trace status area.

For a trace for which error correction is executed, the applied calibration type is indicated at the <u>trace status area</u> by the symbols shown 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

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 1-port calibration for test port 1 and response calibration for test port 2.

Example of calibration property display
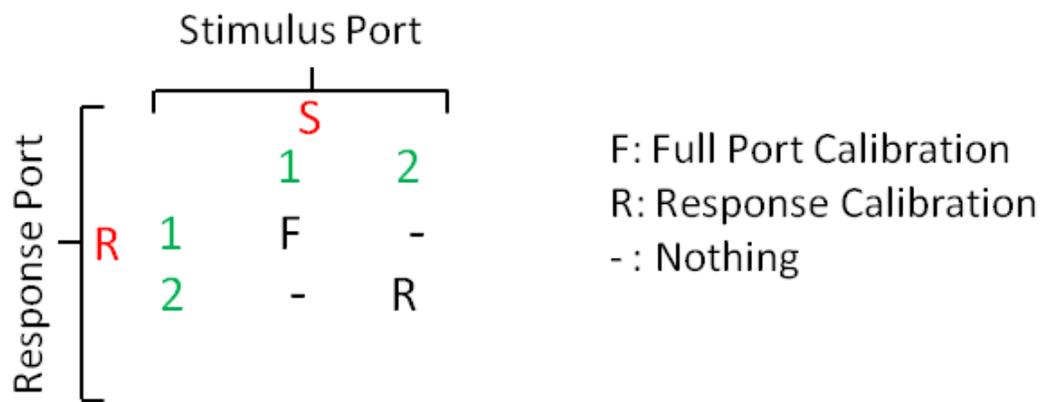
e5072a025

Procedure to turn ON/OFF calibration property display

Follow these steps to turn ON/OFF the calibration property display.

104

1. Press **Channel Next**/**Channel Prev** to select the channel for which you want to turn ON/OFF the calibration property display.
2. Press **Cal**.
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-Parameter is 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 of the overlapping S-Parameter. Take the Example of calibration property display as an example: If you acquire the calibration coefficient of the 1-port calibration for test port 2, neither calibration coefficient is cleared. On the other hand, if you acquire the calibration coefficient for the full 2-port calibration between test ports 1 and 2, the already acquired calibration coefficient of the full 2-port calibration between test ports 1 and 2 are cleared.

Basic Calibrations

Selecting Calibration Kit

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

Other topics about Basic Calibration

Overview

Before performing 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 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.

Procedure

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which you want to select the calibration kit.

2. Press **Cal** > **Cal Kit**, then select the calibration kit.

3. If the name (label) of the calibration kit has been changed, the label is displayed as the softkey.

4. 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 eliminates 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** (or **Channel Prev**) to select the channel for which you want to perform the calibration. Confirm if your required setup is set to the channel.

2. Press **Cal** > **Calibrate**.

3. Select **Response (Open) or Response (Short) calibration**.

4. Click **Select Port**.

5. Select the test port upon which you will perform OPEN/SHORT response calibration.

6. Connect a calibration standard (OPEN or SHORT) to the selected test port (connector to which the DUT is to be connected).

7. Click **Open** or **Short** to start the calibration measurement.

8. If an isolation calibration must be performed using a LOAD standard, follow the procedure below.

9. Connect a LOAD standard to the selected test port (connector to which the DUT is to be connected).

10. Click **Load (Optional)** to start the measurement on the LOAD standard.

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

Connecting standards in OPEN/SHORT Response calibration

e5061b047

108
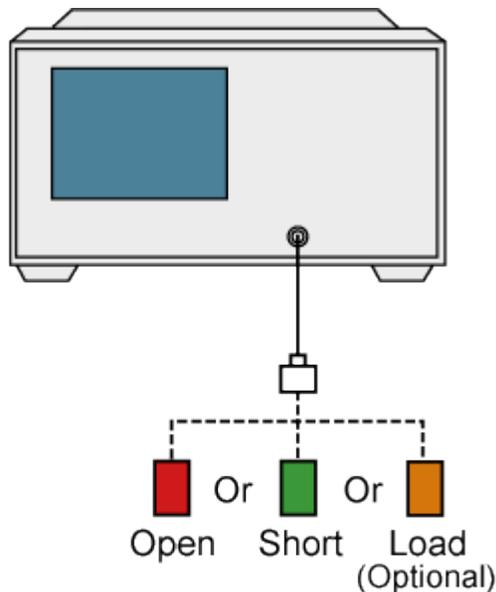
THRU Response Calibration (transmission test)

- [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)

E_t : Transmission tracking
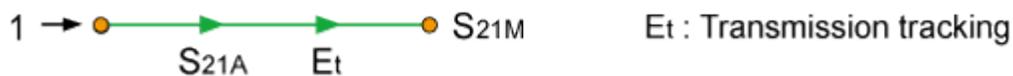
e5071c400

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

2-Port Error model (THRU response + isolation)

E_t : Transmission tracking
E_x : Isolation (Crosstalk)
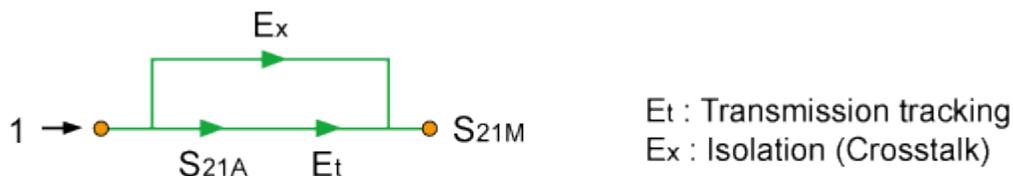
e5071c401

> **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** (or **Channel Prev**) to select the channel for which you want to perform the calibration. Confirm if your required setup is set at the channel.

2. Press **Cal** > **Calibrate** > **Response (Thru)** > **Select Ports**.

3. Select the test ports (and corresponding S parameters) upon which a THRU response calibration is performed.

4. Make a connection between the selected test ports (between the connectors to which the DUT will be connected).

Through connection
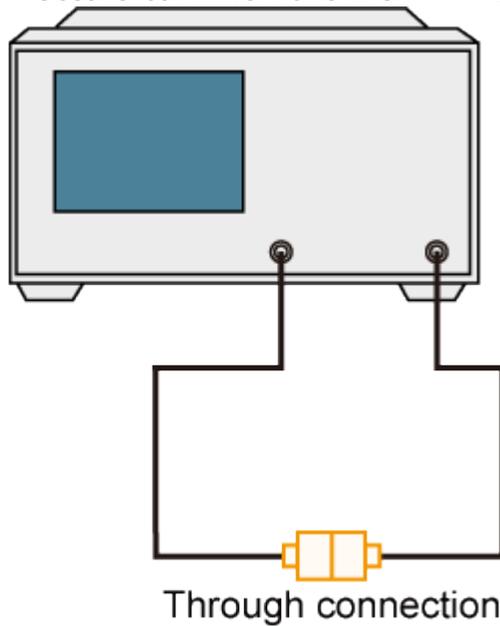
e5071c317

5. Click **Thru** to start the calibration measurement.

6. If an isolation calibration must be performed using a LOAD standard, follow the procedure below:

a.   Connect a LOAD standard to each of the two selected test ports (connectors to which the DUT is to be connected).
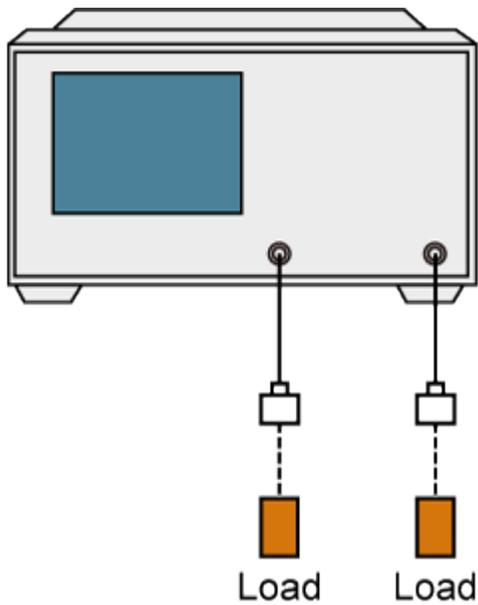
e5071c319

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 are calculated and saved. The error correction function is also enabled automatically.

Enhanced Response Calibration

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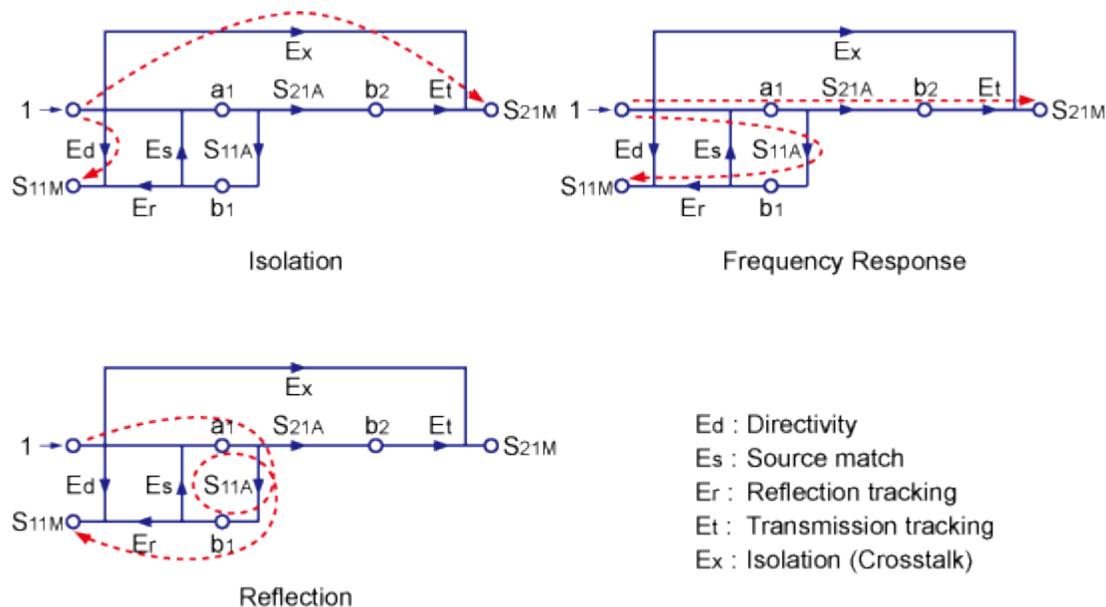
- [Procedure](#)

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 that uses those ports.

> **NOTE**   System Z0 should be changed to 1 ohm before calibration when using waveguide calibration kit and measuring waveguide devices.
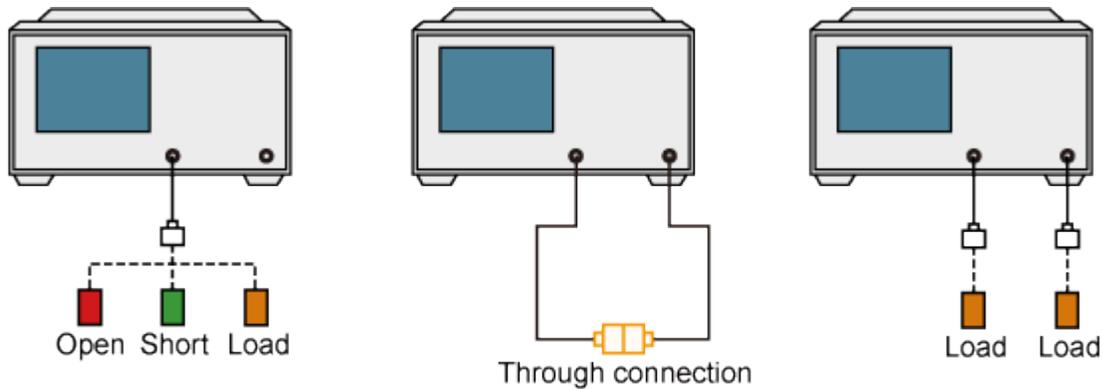
2-Port Error Model (Enhanced Response)

Isolation

Frequency Response

Reflection

Ed : Directivity
Es : Source match
Er : Reflection tracking
Et : Transmission tracking
Ex : Isolation (Crosstalk)

e5071c301

Procedure
Connecting the Standard at Enhanced Response Calibration

e5071c332

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which you want to perform the calibration. Confirm if your required setup is set to the channel.

2. Press **Cal** > **Calibrate** > **Enhanced Response** > **Ports** to select the test ports on which an enhanced response calibration is performed.

3. Connect an OPEN calibration standard to the output port.

4. Click **Open** to start the calibration measurement.

5. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.

6. Click **Short** to start the calibration measurement.

7. Disconnect the SHORT calibration standard and replace it with a LOAD standard.

8. Click **Load** to start the calibration measurement.

9. Make a THRU connection between the two ports.

10.   Click **Thru** to start the calibration measurement.

11.   If an isolation calibration must be performed using a LOAD standard, follow the procedure below:

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

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

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

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.

E_r : Reflection tracking
E_d : Directivity
E_s : Source match

e5071c402

### *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** (or **Channel Prev**) to select the channel of which you want to perform the calibration. Confirm if your required setup is set to the channel.

2. Press **Cal** > **Calibrate** > **1-Port Cal** > **Select Port**.

3. Select a test port (and corresponding S parameter) on which 1-port calibration will be performed.

4. Connect an OPEN calibration standard to the selected test port (connector to which the DUT is to be connected).

5. Click **Open** to start the calibration measurement.

6. Connect a SHORT calibration standard to the selected test port (connector to which the DUT is to be connected).

7. Click **Short** to start the calibration measurement.

8. Connect a LOAD calibration standard to the selected test port (connector to which the DUT is to be connected).

9. Click **Load** to start the calibration measurement.

10.   Click **Done** to terminate the 1-port calibration process. Upon pressing this key, calibration coefficients are calculated and saved. The error correction function is also enabled automatically.

Connecting the standard for 1-port calibration

Open   Short   Load
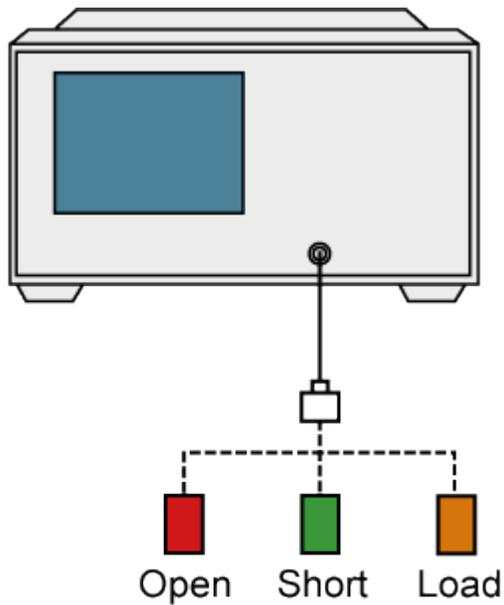
e5071c300

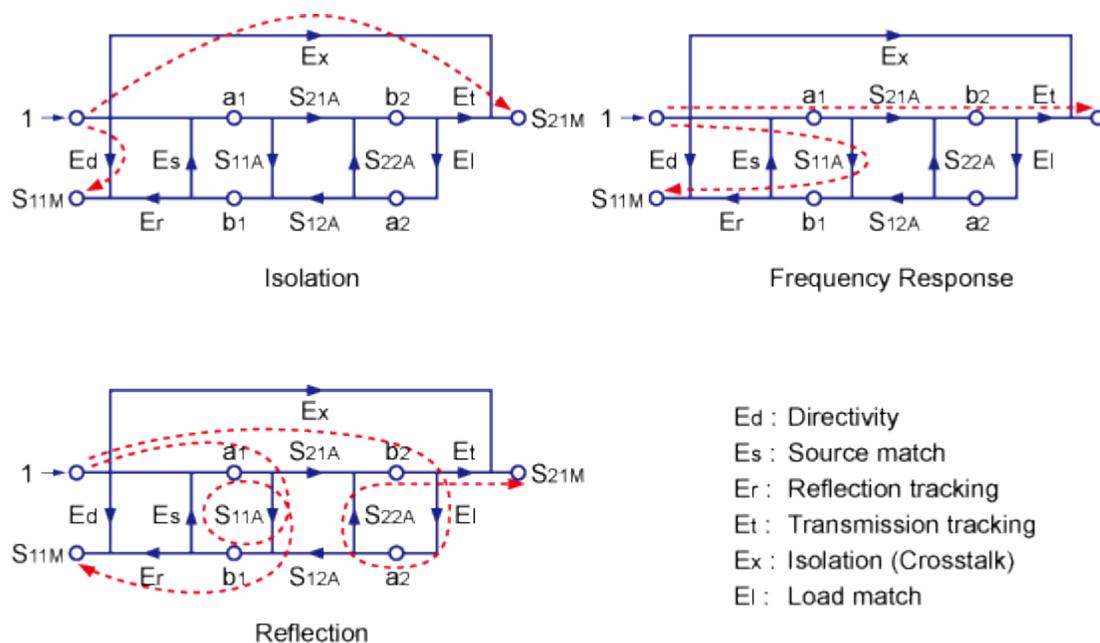## Full 2-Port Calibration

- 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 on ports 1 and 2 for S-parameter (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 tat uses 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 the calibration.

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

Isolation

Frequency Response

Reflection

Ed : Directivity
Es : Source match
Er : Reflection tracking
Et : Transmission tracking
Ex : Isolation (Crosstalk)
El : Load match

e5071c303

Procedure

1. Press **Channel Next** (or **Channel Prev**) to select the channel fof which you want to perform the calibration.

2. Press **Cal** > **Calibrate** > **2-Port Cal**.

3. Click **Reflection**.

4. Connect an OPEN calibration standard to test port x (the connector to which the DUT is to be connected).

5. Click **Port x Open** to start the calibration measurement (**x** denotes the test port to which the standard is connected).

6. Disconnect the OPEN calibration standard and replace it with a SHORT calibration standard.

7. Click **Port x Short** to start the calibration measurement (**x** denotes the test port to which the standard is connected).

8. Disconnect the SHORT calibration standard and replace it with a LOAD standard.

9. Click **Port x Load** to start the calibration measurement (**x** denotes the test port to which the standard is connected).

10.   Repeat the above procedure for port y.

11.   Click **Return**.

12.   Click **Transmission**.

13.   Make a THRU connection between ports x and y (between the connectors to which the DUT is to be connected).

14.   Click **Port 1-2 Thru** to start the calibration measurement.

15.   Click **Return**.

16.   If an isolation calibration must be performed using a LOAD standard, follow the procedure below.

17.   Click **Isolation (Optional)**.

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

19.   Click **Port 1-2 Isol** to start the calibration measurement.

20.   Click **Return**.

21.   Click **Done** to terminate the full 2-port calibration process. Upon pressing this key, calibration coefficients are calculated and saved. The error correction function is also enabled automatically.

Connecting standards in full 2-port calibration

Open Short Load Open Short Load   Through connection   Load   Load
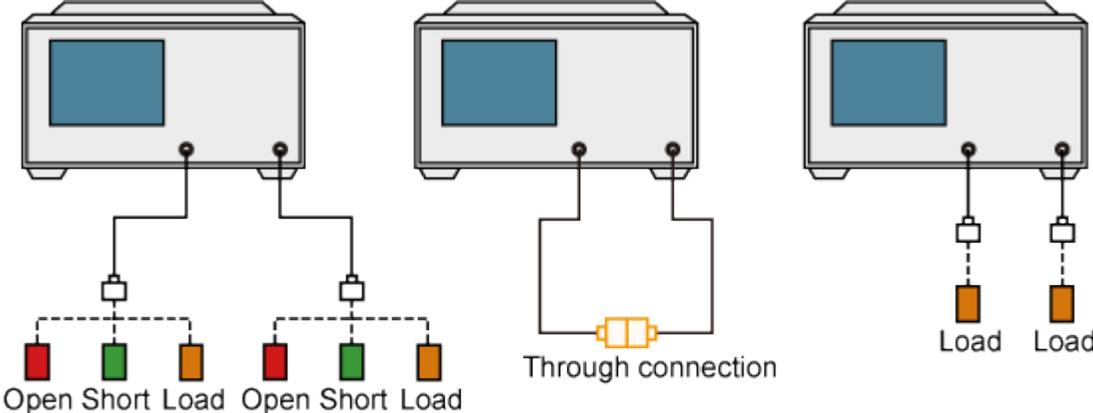
e5071c331

Clear Calibration

This softkey clears the user calibration data. When Calibration is done for a particular DUT, the data get stored in the E5063A. To clear this data, Clear Calibration function (**Cal** > **Clear** > **OK**) can be used which removes the calibration data from the E5063A.

Other topics about 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 ad tear 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 is used for the exceeding frequency range and extrapolation is executed.

Refer the following section for ECal calibration.

- ECal Driver Installation

- Calibration Using Ecal Module

- Improving Calibration Accuracy along with ECal

- Confidence Check on Calibration Coefficients Using ECal

- Turning off ECal auto-detect function

Other topics about Calibration with ECal

ECal Driver Installation

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

1. Connect an ECal to the USB port of the E5063A.

2. "Agilent USB ECal Module : Device driver software installed successfully" message box prompts at the right end of the task bar.

3. ECal is now ready to be used.

Other topics about Calibration with ECal

Calibration Using Ecal Module

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

Other topics about Calibration with ECal

### Overview

The E5063A allows you to perform calibration using the 2 or 4-port ECal module in S-parameter measurement.

Procedure

1. Connect the USB cable between the USB port of the 2 or 4-port ECal module and the USB port of the E5063A. You can make this connection while the E5063A's power is on.

2. Allows the ECal module to warm up for up to 20 minutes until the module indicator changes from WAIT to READY. Warm up time depends on the ECal module.

3. Connect the ports of the 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** (or **Channel Prev**) to select the channel of which you want to perform the calibration.

6. Press **Cal** > **Ecal**.

7. Select the **calibration type**.

8. 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, skip this step.

9. The E5063A detects the test ports connected to the ECal and then measurement starts. If the test ports to be calibrated are not connected to the ECal module, error occurs.

Connecting 4-port ECal module (for full 2-port calibration)

USB

USB Cable

4 Port ECal
Module

▌ : termination
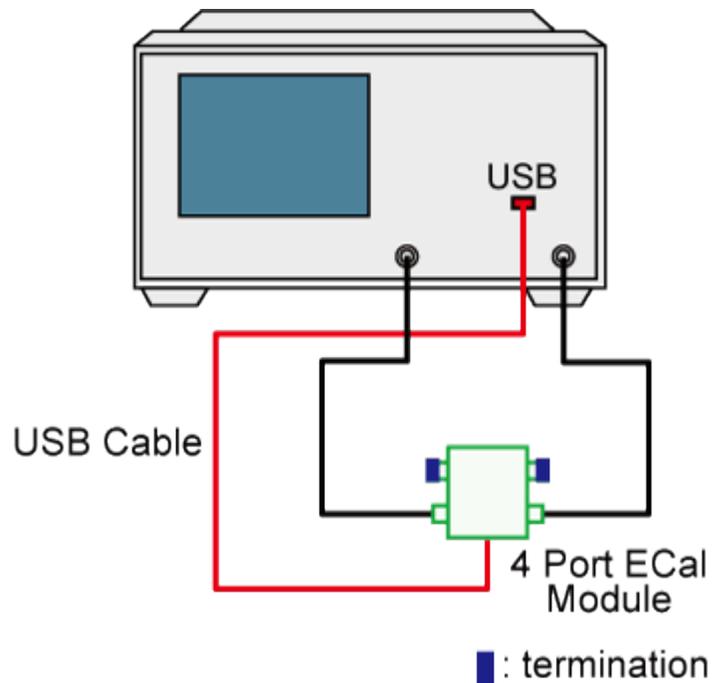
e5061b013

- You can connect the ports of the ECal and the test ports of the E5063A arbitrarily. Connected ports can be manually specified although they are automatically detected before the data measurement. For more information, see Turning off ECal auto-detect function.

Improving Calibration Accuracy along with ECal

Inaccuracy caused by Thru calibration in the ECal can be reduced by 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

E5063A 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. E5063A performs the Thru calibration as an unknown Thru calibration.

1. Press **Cal**,then click **ECal** > **Unknown Thru** to turn it ON.

2. Perform your desired ECal calibration.

Confidence Check on Calibration Coefficients Using ECal

- Overview
- Procedure

Other topics about Calibration with ECal

Overview

By using the ECal module, the E5063A allows you to verify the obtained calibration coefficients to determine whether correct measurement is possible with them.

The E5063A is able to 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 E5063A 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 E5063A. You can make this connection while the E5063A's power is ON.

2. Allows the ECal module to warm up for up to 20 minutes until the module indicator changes from WAIT to READY.

3. Press **Channel Next** (or **Channel Prev**) to select the channel of 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 E5063A 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 do not use all of the ECal module's ports, terminate connections to the unused ports.

8. Press **Cal** > **ECal**.

9. When using an adapter to the ECal, click **Characterization** and then press the softkey corresponding to the characterization of the adapter you are using.

10. Click **Confidence Check**.

11. Compare the data trace and memory trace and verify whether measurement is correct.

12. The following is the procedure for comparison when simultaneously displaying the data trace and the memory trace.

   a. Press **Display** > **Display** > **Data & Mem**.

   b. Press **Scale** > **Auto Scale**.

   c. Determine whether the differences between the traces are acceptable. The differences should be read in terms of linear values instead of dB error. If you compare the magnitude of the linear error with 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.

13. For all of the parameters you want to verify, repeat the procedure.

Connecting ECal module (for verification of S21)
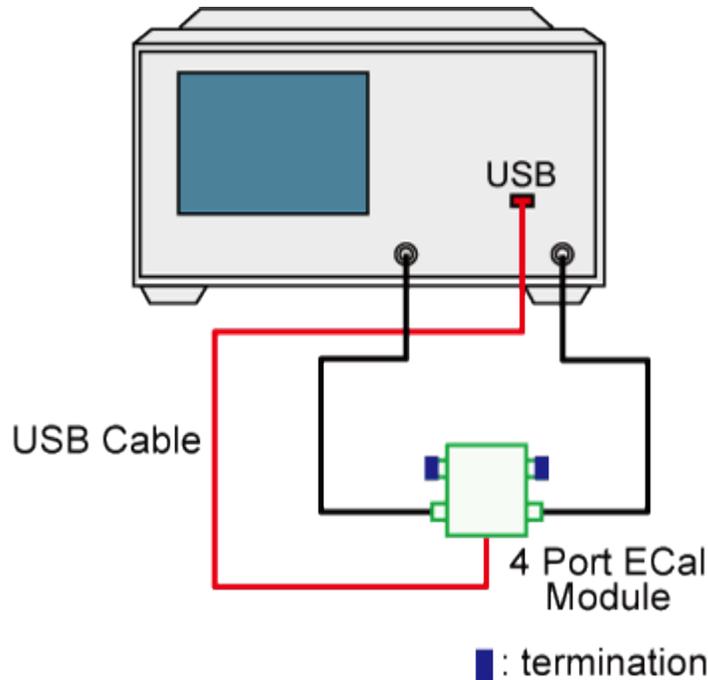
e5061b013

Turning off ECal auto-detect function

The ECal module automatically detects the connection between E5063A's test ports and ECal module's ports. You can turn off this function to set ports manually.

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which you want to turn off the auto-detect function.

2. Press **Cal** > **ECal** > **Orientation** > **Manual**.

3. Specify a test port of the E5063A.

4. Specify an Ecal port for the port of the E5063A you specified.

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

Other topics about Calibration with ECal

User-characterized ECal

- [Overview](#)
- [Executing User characterized ECal](#)

Overview

The E5063A 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.

The E5063A cannot define the User-characterized ECal and cannot save them to the built-in flash memory of the ECal module as the user characteristics. To do that, other ENA network analyzers are required.

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** > **ECal** > **Characterization**.

3. Select a user characteristic.

4. To check the information on the user characteristics you have selected, click **Characterization Info**. The following dialog box appears.
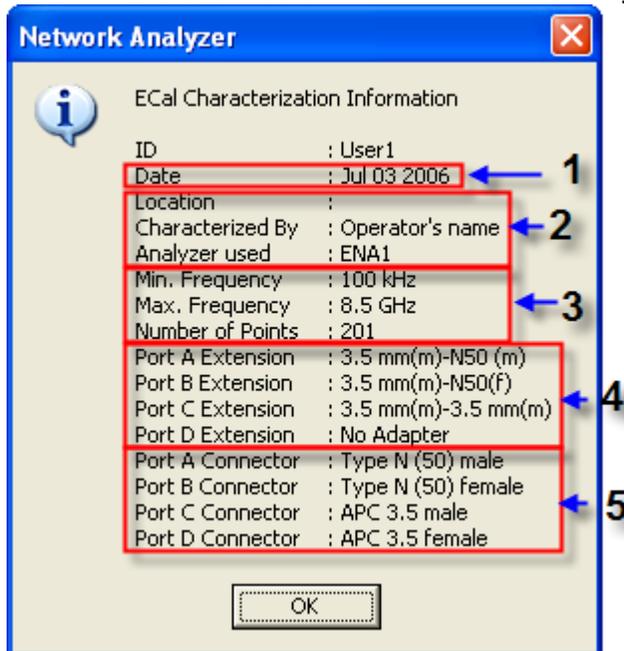
1:	The date when the user characteristics were measured

2:	The information you entered in Characterization
3:	The stimulus conditions when the user characteristics were measured
4:	The information you entered in Adapter Description
5:	The information you entered in Connectors

Advanced Calibrations

Advanced Calibrations

- [Modifying Calibration Kit Definition](#)
- [Specifying Different Standard for Each Frequency](#)
- [Partial Overwrite Calibration](#)
- [Unknown Thru Calibration](#)
- [Adapter Removal-Insertion](#)
- [2-port TRL Calibration](#)
- [Sliding Offset Load Calibration](#)
- [Performing 8 term Calibration using External PC](#)

Modifying Calibration Kit Definition

- [Definition of Terms](#)
- [Defining Parameters for Standards](#)
- [Redefining a Calibration Kit](#)
- [Setting Options for TRL Calibration](#)
- [Defining Calibration Kit Gender](#)
- [Saving Definition File of Calibration Kit](#)
- [Loading Definition File of Calibration Kit](#)
- [Restoring Definition File of Calibration Kit](#)

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 E5063A (e.g., a SMA connector).

- When the user wants to use different standards in place of one or more standards pre-defined in the E5063A. 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 Ω instead of 50.0 Ω.

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 E5063A, the user may define up to 30 standards per calibration kit. Each standard is numbered from 1 through 30.

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 30 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)

Calibration plane

offset loss

C_{0-3}

L_{0-3}

offset Z_o

offset delay

R_L
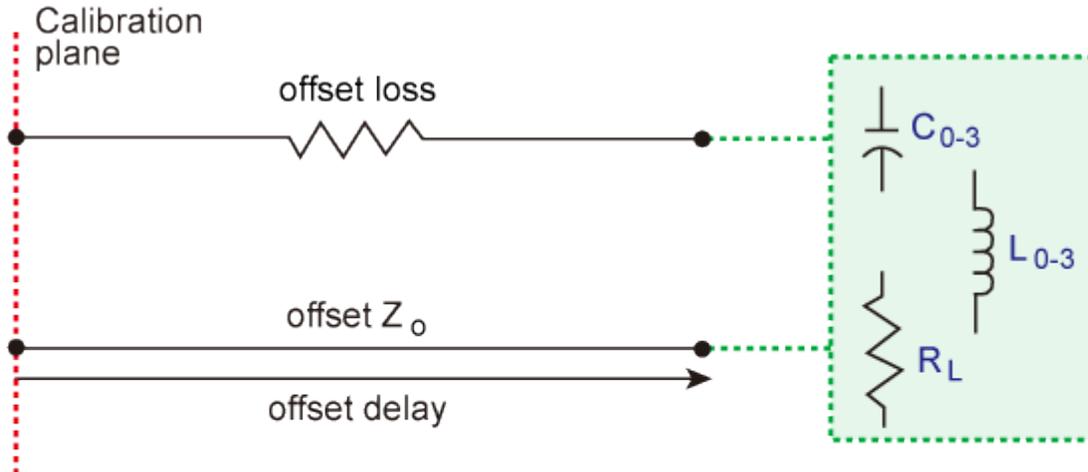
- Frequency range
- Coax or Waveguide
- Fixed, Sliding, or Offset (load only)

Model for Reflection Standard (short, open, load or arbitrary Impedance)

e5071c494

Transmission Standard Model (THRU)

Calibration plane

Calibration plane

offset loss

offset Z_o

offset delay
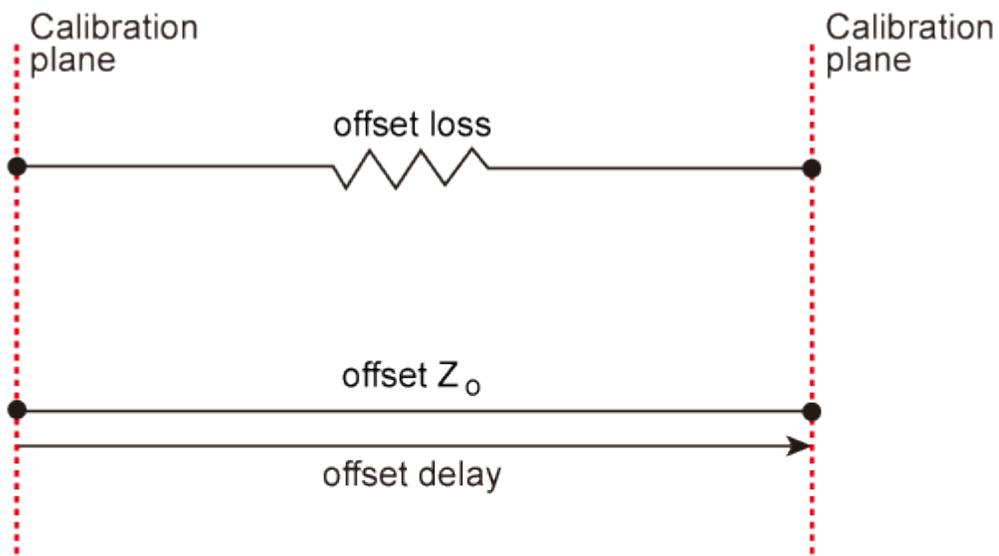
- Frequency range
- Coax or Waveguide

Model for Transmission Standard (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 (components in the calibration kit) 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

NOTE The calibration kit definition are stored in the D:\Agilent\CalKit. Redefined calibration kit data are stored into the files.

Procedure to select and define a calibration kit

1. Press **Cal**.

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 Cal Kit**. To change the pre-defined connector type (e.g. OPEN(f) to OPEN (m)) skip to the [specify 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**.

2. Click **Modify Cal 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**.

5. Set the **standard coefficient**.

6. 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**.

2. Click **Modify Cal Kit**.

3. Click **Specify CLSs**, then select the **class**.

4. Select the test port. Select **Set All** to use the same standards for all the test ports.

5. Select the standards to be registered in the class from among standards numbered 1 through 30. 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](#).

6. Repeat the procedure until classes are defined for all the test ports that need to be redefined, then click **Return**.

7. Repeat the procedure to redefine all classes that need to be modified, then click **Return**.

8. 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 GΩ /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**.
2. Click **Cal Kit**, then select a User kit that has not been registered.
3. Click **Modify Cal Kit** > **Label Kit [User]**, then enter your desired name.

Procedure to define Thru and Reflect

1. Click **Define STDs**.
2. Click **1:No Name** > **Label** > **THRU**.
3. Select **STD Type** > **Delay/Thru**.
4. Set **Offset Loss** to 1.3Gohm/s, and **Offset Delay** to 0.
5. Click **Return**.
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** twice.

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

1. Click **Specify CLSs** > **Sub Class.**

2. Select **Sub Class1**.

3. Click **TRL Thru** > **Port 1-2** > **THRU** > **Return**.

4. Click **TRL Reflect** > **REFLECT**.

5. Click **TRL Line/Match** > **Port 1-2** > **MATCH <2G** > **Return**.

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

1. Click **Sub Class2**.

2. Click **TRL Line/Match** > **Port 1-2** > **LINE <7G** > **Return**.

3. Click **Sub Class3**.

4. Click **TRL Line/Match** > **Port 1-2** > **LINE >7G** > **Return**.

5. Press **Cal** 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**.

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

3. Click **Modify Cal Kit** > **TRL Option**.

4. Click **Impedance** to select the **reference impedance**.

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

Setting a media type for the calibration kit

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

1. Press **Cal**.

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**.

6. If you select waveguide as the media type, set the system impedance and the characteristic impedance to 1ohm.

7. 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 <u>Saving Definition File of Calibration Kit</u>.

Defining Calibration Kit Gender (m/f)

The gender of a calibration can be set using the subclass function in the E5063A. 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 Cal 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 E5063A 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**.

2. Click **Cal Kit** and select **85032F**.

3. Select **Cal** > **Modify Cal Kit** > **Specify CLSs** > **Sub Class**.

The subclass gender is set from the E5063A side and not from the Calibration kit side.

> **NOTE**   The gender setting option using subclass is defined from the E5063A 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**.

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 Do not modify any files and folders in drives other than drive D:. Modifying the contents of drives other than drive D may give serous damage to the functions and performance of the analyzer. Even in drive D, do not change the files and folders other than you created.

Loading Definition File of Calibration Kit

1. Press **Cal** > **Cal Kit**, then select a calibration kit.

2. Click **Modify Cal Kit** > **Import Cal Kit...** to open the dialog box.

3. Specify a folder, enter a file name, and click **Open**.

Restoring Definition File of Calibration Kit

1. Press **Cal** > **Cal Kit**, then select a calibration kit.

2. Click **Modify Cal Kit** > **Restore Cal Kit...**.

3. Click **OK** to restore the definition of the calibration kit selected by **Cal Kit** to factory default settings.

Specifying Different Standards for Each Frequency

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

2. Click **Cal Kit**, then select a calibration kit you want to use.

3. Click **Modify Cal Kit** > **Specify CLSs**.

4. Click **Sub Class** > **Sub Class 1**.

5. Click **Open** > **Set All** to set OPEN standard type for all the ports.

6. Select #1 (**Open 3G**).

7. Click **Return**.

8. Click **Sub Class** > **Sub Class 2**.

9. Click **Open** > **Set All** to set OPEN standard type for all the ports.

10.   Select #2 (**Open 6G**).

11.   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**.

2. Click **Cal Kit**, then select a calibration kit you want to use.

3. Click **Modify Cal Kit** > **Specify CLSs**.

4. Click **Sub Class** > **Sub Class 2**.

5. Click **Open** > **Set All** to set OPEN standard type for all the ports.

6. Click **None**> **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 about the 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	Possible

[range](#)

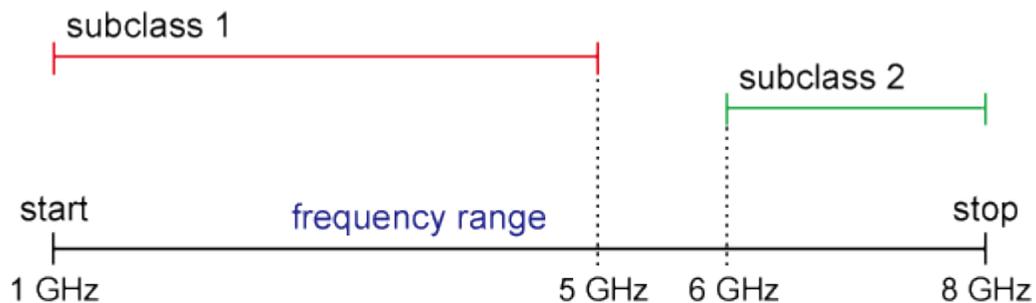**Example 1, When frequency ranges specified with subclasses overlap**

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).
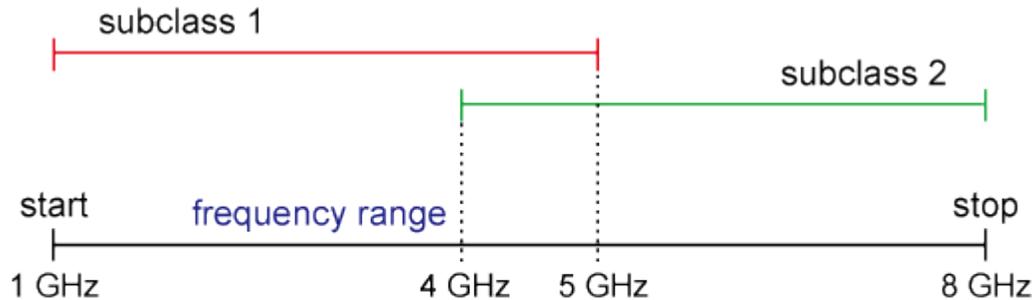
e5071c470

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).

e5071c468

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 is 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.
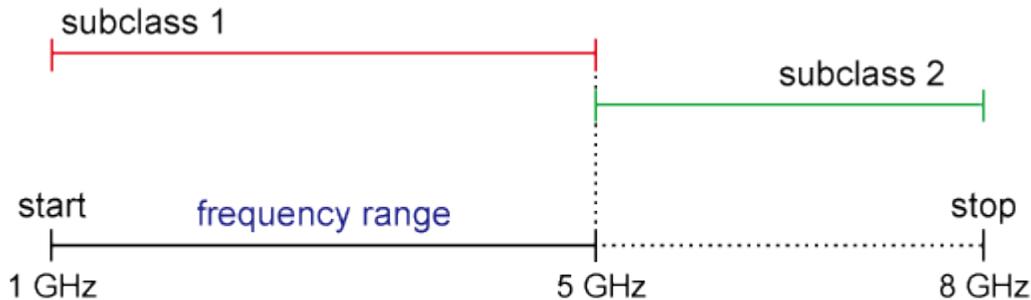
e5071c467

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.

e5071c469

144
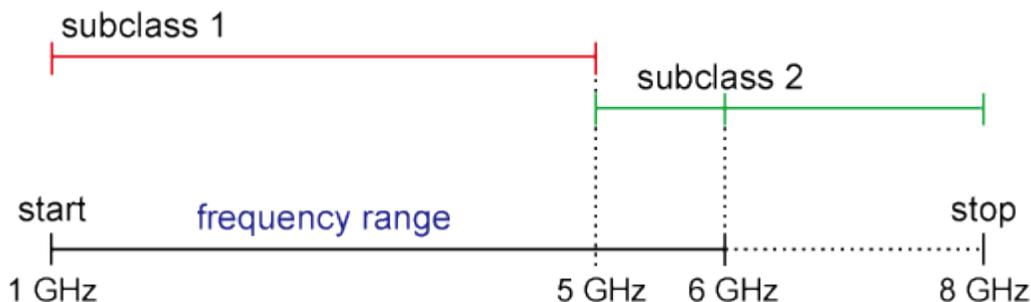
Partial Overwrite Calibration

- [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 the standards again.

- When the calibration coefficients change over time, or the status on the E5063A side from the calibration surface changes due to replacement of a cable or connector, you also need to perform thru measurement when partial overwrite is required for reflection or isolation measurement.

- The adapter removal and partial overwrite function is only available when calibration status is [**Cor**] 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 2-port calibration by performing additional calibration for 1 port after the execution of full 1-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 thru calibration only for exiting 2 port full calibration:

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which you already have performed full 2 port calibration.
2. Press **Cal** >**Calibrate** > **2-Port Cal**.
3. Click **Transmission**.
4. Make a thru connection between the ports 1 and 2.
5. Click **Port 1-2 Thru**.
6. Click **Return**.

7. Click **Overwrite** to finish the re-calibration for the full 2-port calibration. At this point, the calibration coefficients are re-calculated with the new thru calibration and saved.

.

Unknown Thru Calibration

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

Other topics about Advanced Calibration

Overview

Unknown Thru Cal is the preferred THRU method of calibrating the E5063A 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.

147

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** > **Calkit** , then select the desired calibration kit (e.g. 85033D or User Kit).

2. Press **Cal** > **Modify Cal Kit** > **Define STDs** > **No Name** > **Label**.

3. Type a name for unknown Thru Cal standard e.g. {Unknown Thru1}.

> **NOTE**   By changing the label to a different name (e.g. Unknown Thru1), the 'No Name' standard adopts to the specified new name.

4. Press **Cal** > **Modify Cal Kit** > **Define STDs** > **{Defined Name}** > **STD Type** > **Unknown Thru**.

5. Click **Offset Delay,** then enter the approximate offset delay value of the unknown Thru standard.

6. Press **Cal** > **Modify Cal Kit** > **Specify CLSs** > **Thru** > **Port 1-2** > **{Defined Name}**.

> **NOTE**   Different Unknown Thru can be used for different ports.

7. Press **Cal** > **Calibrate** > **2-Port Cal** > **Transmission** > **{Defined Name}** to execute mechanical Unknown Thru calibration.

8. Complete Reflection, and Isolation (optional) calibration for 2 full port Cal.

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 & E5063A 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 E5063A uses the following adapter removal process to remove the 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***
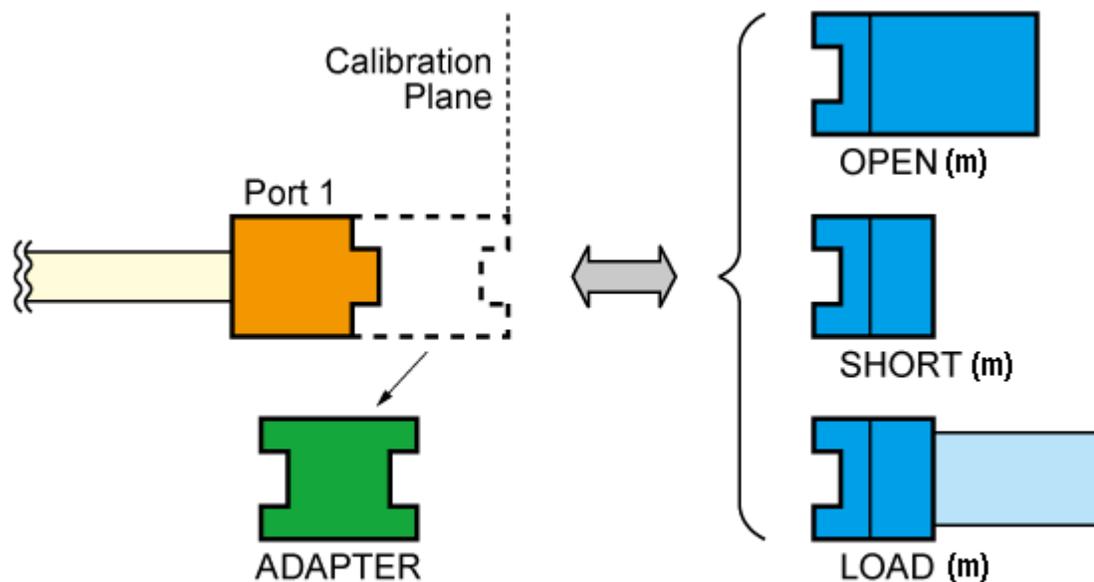
e5071c203

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

About Adapter Insertion

The 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. E5063A uses the following adapter insertion process to insert the 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
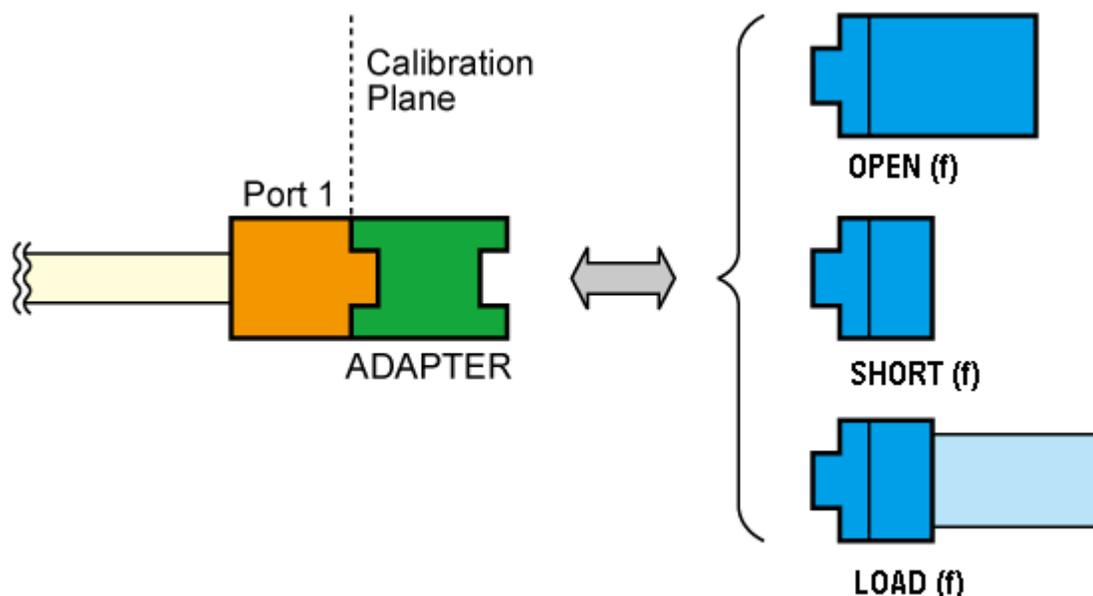
e5071c204

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, and independent of the direction from which it is used.

2. The electrical length of the adapter should be known with the accuracy of ±1/4 of wavelength.

About Adapter Waveguide & Rotate

E5063A has the functionality related to adapter waveguide and adapter rotation. Adapter waveguide length and Cut off frequency can be set using the following commands:

:SENS:CORR:COLL:ADAP:WAV:LENG

:SENS:CORR:COLL:ADAP:WAV:CUT

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.

:SENS:CORR:COLL:ADAP:ROT

> **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 E5063A stimulus setting is out of the operational frequency range, you cannot click the **Done** key or finish the calibration by remote control. In this case, use a calibration kit that has proper frequency range, or change the E5063A stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5063A 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).
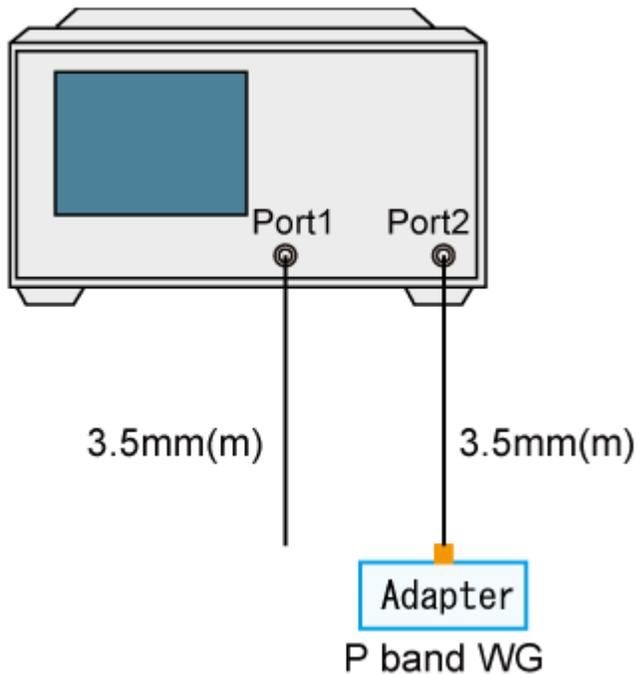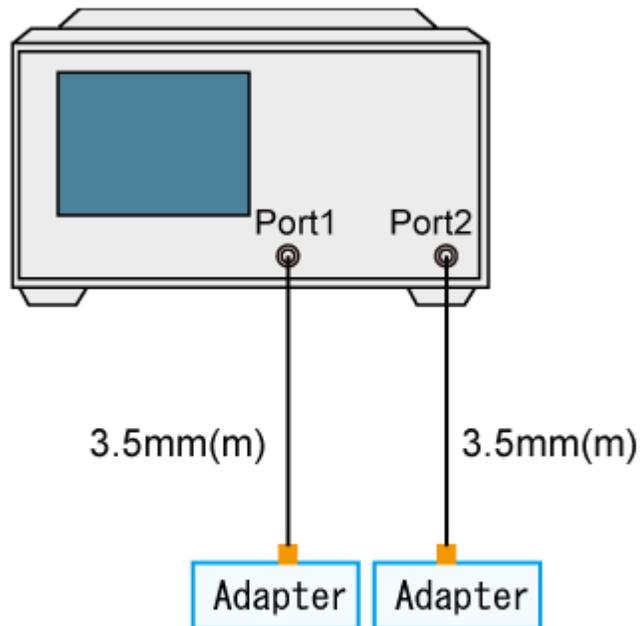
e5071c488

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
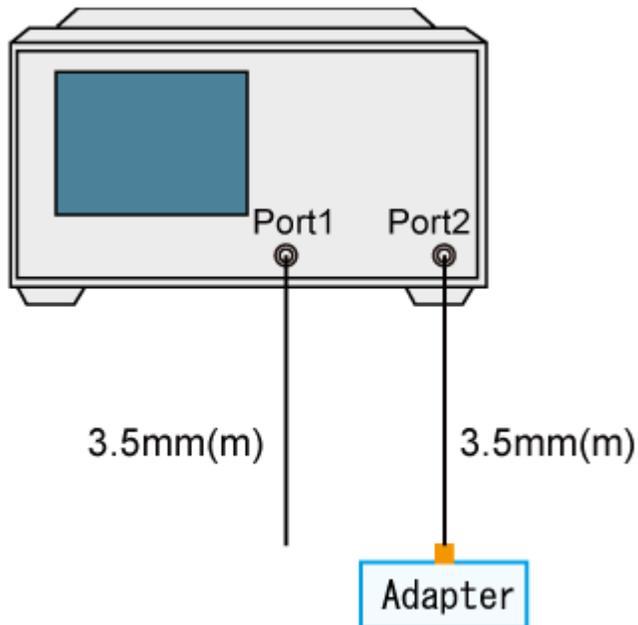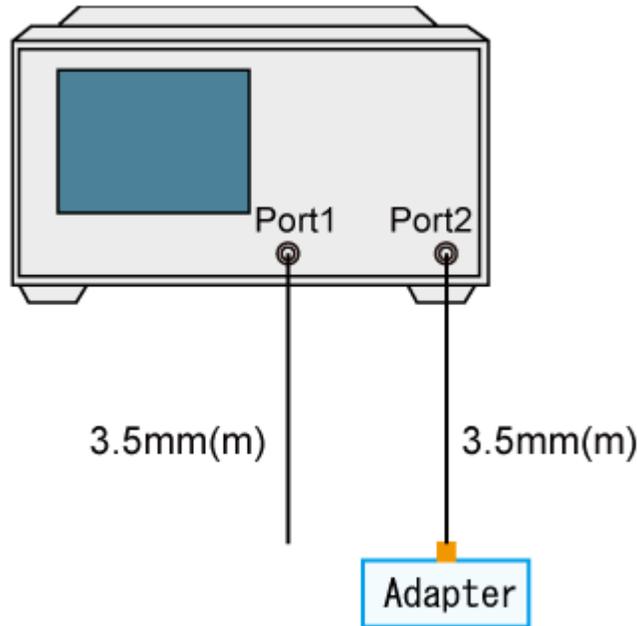

e5071c489

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 of 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:

- Open, Short, and Load measurements.

- Actual values derived from the **CalKit** definitions.

- An approximate length of the adapter.

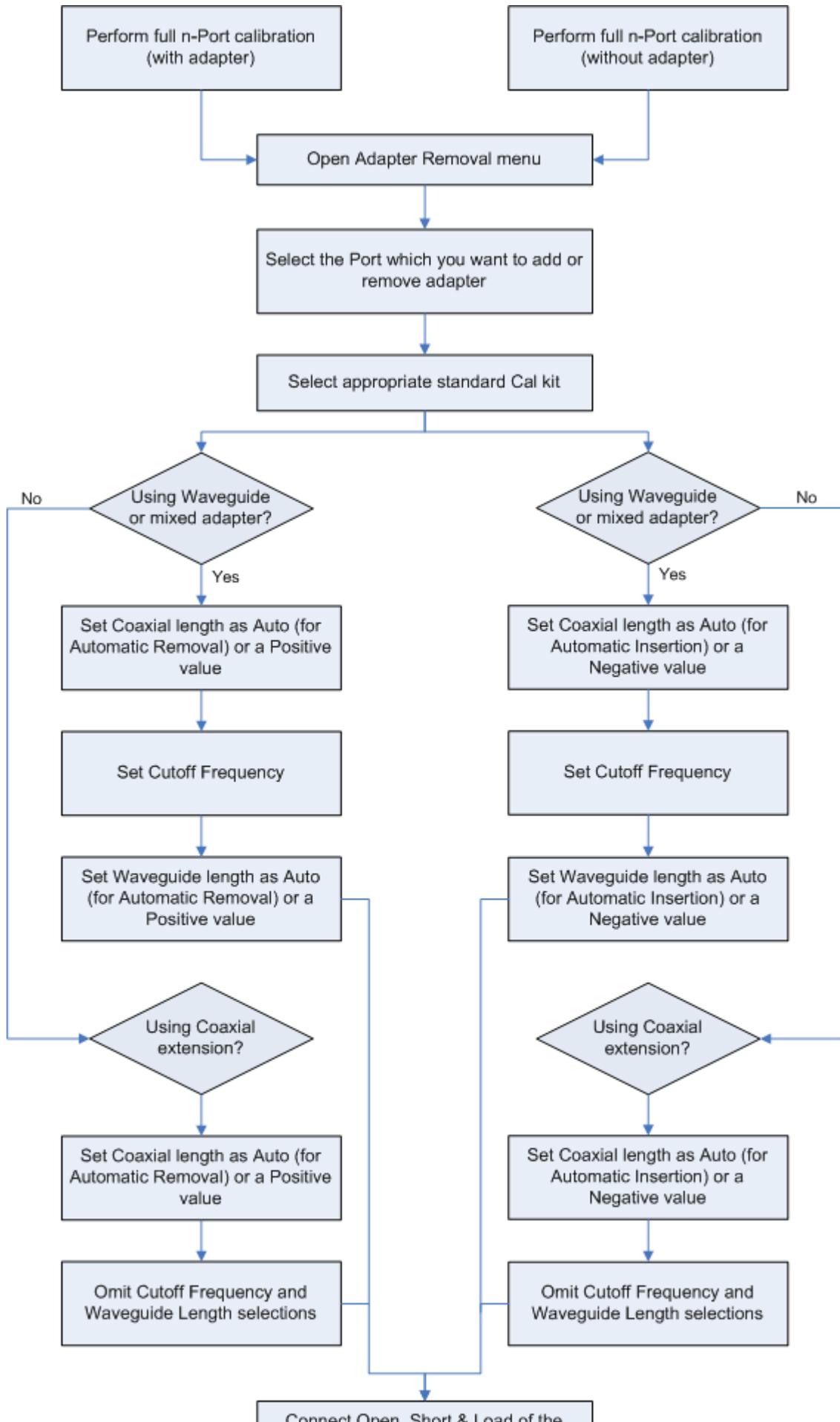- Nature of the intended operation: removal or insertion.

> 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.

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

```
┌─────────────────────────┐          ┌─────────────────────────┐
│ Perform full n-Port      │          │ Perform full n-Port      │
│ calibration (with adapter)│         │ 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

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**.
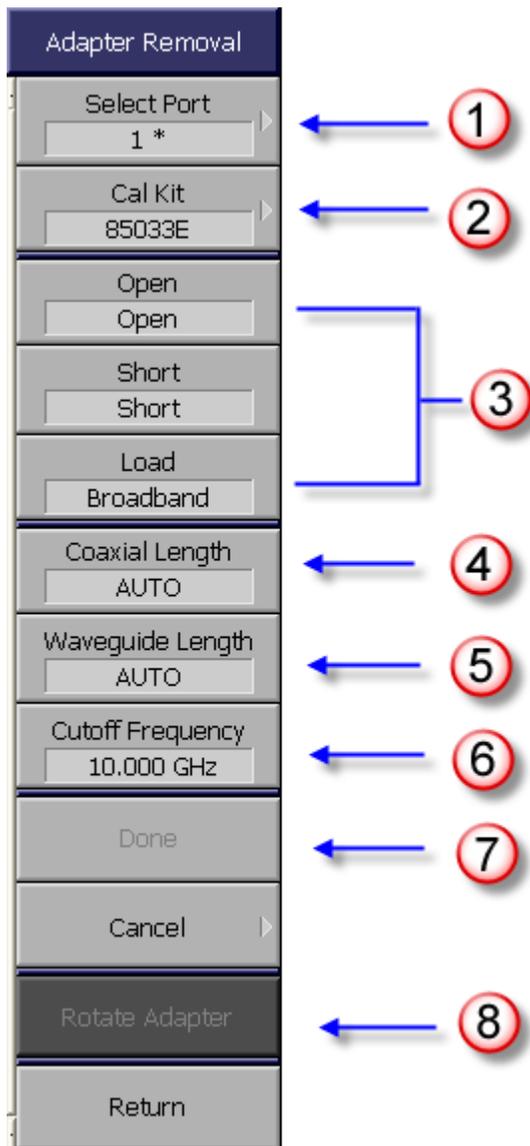
e5071c194

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 click/press **Open**, **Short**, and **Load** respectively. E5063A measures the cal kit standard, calculates the adapter characteristics, and then conducts adapter removal/insertion.

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

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 E5063A stimulus setting is out of the operational frequency range,

157

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 E5063A stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5063A 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.

   a. 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.

158

> **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.

1. 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 E5063A 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 E5063A stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5063A 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.

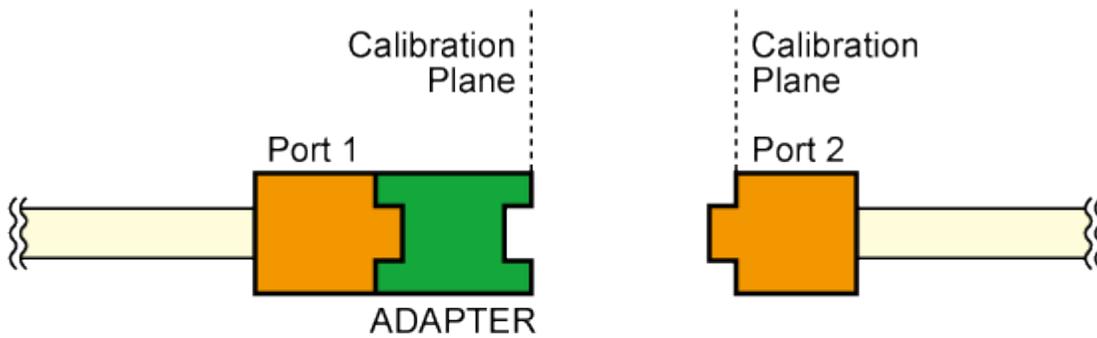**Difference between Traditional Network Analyzer & E5063A Adapter Calibration**

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

Calibration performed with the adapter connected to Port2
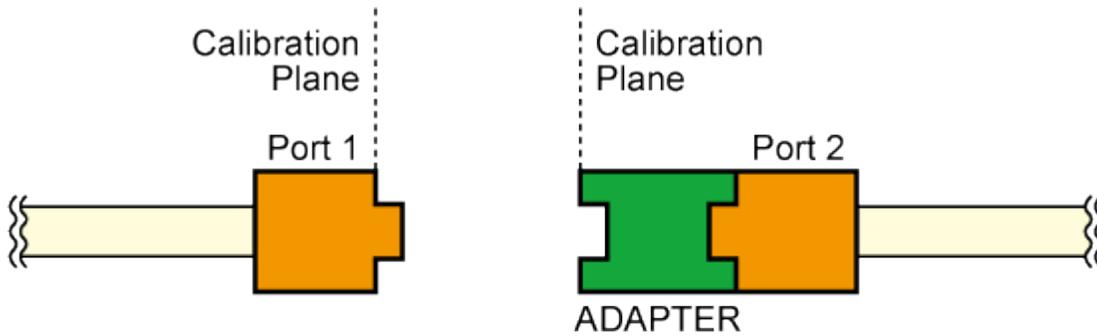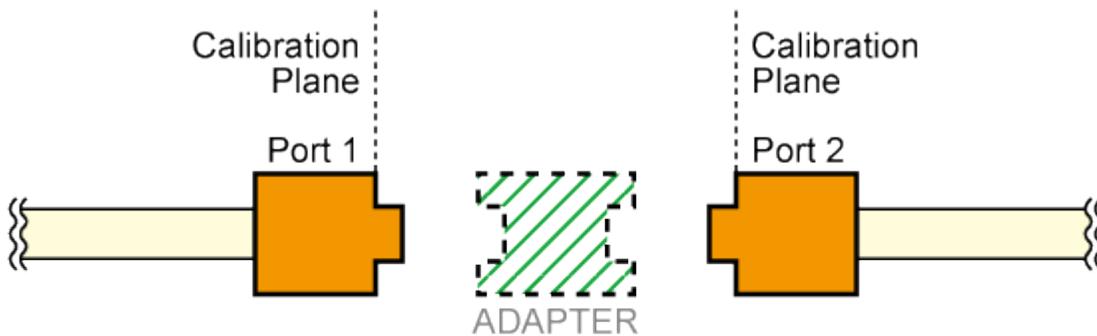
e5071c200

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 requires Full 2 Port Calibration as many as twice the number of port combinations. Therefore, the E5063A uses an advanced method to remove the Adapter characteristics described in <u>About Adapter Insertion</u>.

2-port TRL Calibration

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

Other topics about Advance 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. Connect the port.(In the following procedure, test ports you select here are expressed as test ports x and y.)

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** > **Port x-y Thru** to execute a Thru/Line measurement.
3. Click **Return**.

Reflection

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

Line/Match

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

4. Click **Done** to complete TRL 2-port calibration. At this point, the calibration coefficient is calculated and saved. The error correction function is automatically turned ON.

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 Ω 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** to select the channel for which you want to perform the calibration.
2. Press **Cal**.
3. Click **Calibrate** > **Cal Kit**.
4. Select the calibration kit with sliding load, for example, **85052B**.

5. Press **Cal**.

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 an 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** to select the channel for which you want to perform the calibration.

2. Press **Cal**.

3. Click **Calibrate** > **Cal Kit**.

4. Select the calibration kit with sliding load, for example, **X11644A**.

5. Press **Cal**.

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** > **Done** > **Return** twice.

17. Now, all **Short**, **Open** and **Load** should have check mark on their left.

18. Click **Done**.

166

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.

Performing 8 term Calibration using External PC
- [Overview](#)
- [Procedure](#)

[Other topics about Advanced Calibration](#)

Overview

To perform 8-term error model calibration using an external PC (measuring raw measurement data using the E5063A, 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.

- 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 the E5063A.

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

Procedure

Turning OFF the "System Correction"

In its default setting, the E5063A applies "system correction" to the measured S-Parameter data. As the "system correction" 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 the external computer. You need to turn OFF this state all the time for calibration and measurement.

The system calibration can be turned OFF by:

- **System** > **Service Menu** > **System Correction**
- :SYST:CORR

The setting of system correction is kept until preset. This state is saved in the state file.

Making Measurements

Making Measurements

- [Setting Up Trigger](#)
- [Making Averaging Measurement with Single Trigger](#)
- [Distributing Trigger to External Device](#)
- [Making Trigger](#)

Setting Up Trigger

Other topics about Making Measurement

Overview

The E5063A has one trigger source. When this trigger source detects a trigger signal that has occurred, a sweep or point measurement is performed for channels.

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.

For each channel, a sweep is performed only for the stimulus ports required to update the parameters of the displayed trace.

Selecting Trigger Source

The trigger source generates a cue signal that initiates a measurement process. Four types of trigger sources are available:

1. Press **Trigger** > **Trigger Source**, then select the desired **trigger source**.
2. When **External** is selected as a trigger source, click **Ext Trig Input** to select **trigger polarity**.
3. The setting for trigger polarity is NOT valid for the external trigger from the 24 Bit I/O (Hander).

Setting Trigger Mode (Single/Continuous)

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.

1. Press **Channel Next** (or **Channel Prev**) to select the channel for which the trigger mode will be set.
2. Press **Trigger**, then select the desired **trigger mode**.
3. Repeat the procedure until each channel is set to its trigger mode.

Making Trigger to Active Channel Only (Trigger Scope)

The trigger scope specifies the scope of the triggering, whether it is for all channels or for the active channel.

For example, when **Trigger** > **Continuous** is selected for all the channels and the trigger scope is set to active channel, a measurement channel is automatically changed by changing an active channel.

1. Press **Trigger** > **Trigger Scope**, then select the desired **trigger scope**.

Making One Point Measurement with Single Trigger (Point Trigger)

The point trigger provides a point measurement at every trigger, and it can be used to change the trigger event to point trigger mode.

1. Press **Trigger** > **Trigger Event**, then select the desired **trigger event**.

2. When the trigger source is the internal trigger, the point trigger does not work.

Setting Trigger Delay

Set the external trigger delay time at each point. The trigger delay works when the trigger source is set to external.

1. Press **Trigger** > **Trig Delay**.

2. Enter an external trigger delay time.

See the timing chart for sweep.

Trigger Delay Time and Point Trigger Interval

External trigger pulses which are 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 source = external)
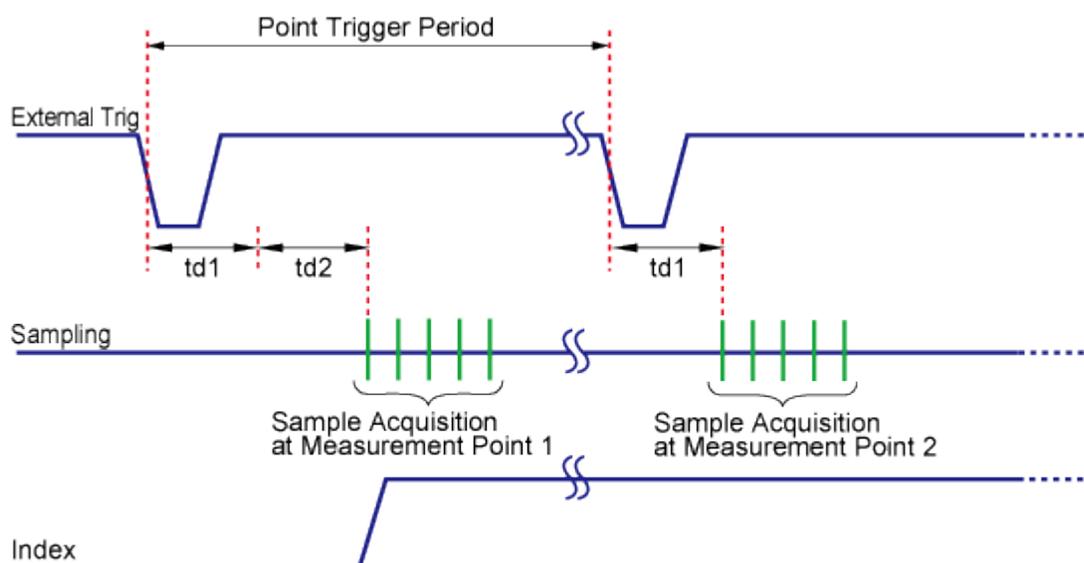
e5061b036

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 E5063A is actually performing measurement.
Index	Index signal of the handler I/O port. When the point trigger function is ON, it goes to the High level only before starting the measurement of the first sweep point and returns to the Low level after completing the measurement of all the measurement points.
Point Trigger Period	Time until the E5063A is ready to accept a trigger for the next measurement point. The value depends on the measurement conditions and the settings of the E5063A.
td1	Time set as the external trigger delay time.
td2	Time for sweep delay

Making Averaging Measurement with Single Trigger

- [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.

- 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)".

- When the sweep averaging function is OFF, sweep is performed only once even if the averaging trigger function is set to ON.

- The averaging trigger function is valid for all the channels. Note that you can set the sweep averaging function for each channel.

Averaging Trigger Function

Setting Averaging Trigger Function

When the sweep averaging function is **ON**, follow these steps to set the averaging trigger function.

1. Press **Avg** > **Avg Trigger**.

2. Click **ON** to activate the averaging trigger.

Executing Averaging Measurement

1. Press **Trigger**.

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.

Distributing Trigger to External Device

- 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 E5063A.

Setting External Trigger Output

1. Press **Trigger** key, and then set the value of **Ext Trig Output** as **ON**.

2. Click **Polarity**, and then select **polarity**.

3. Click **Position**, and then select **position**.

4. Click **Pulse Width** to define the pulse width of the signal.

Difference between After and Before Point Settings

External trigger output function displays the difference in results by setting the Point trigger as follows:
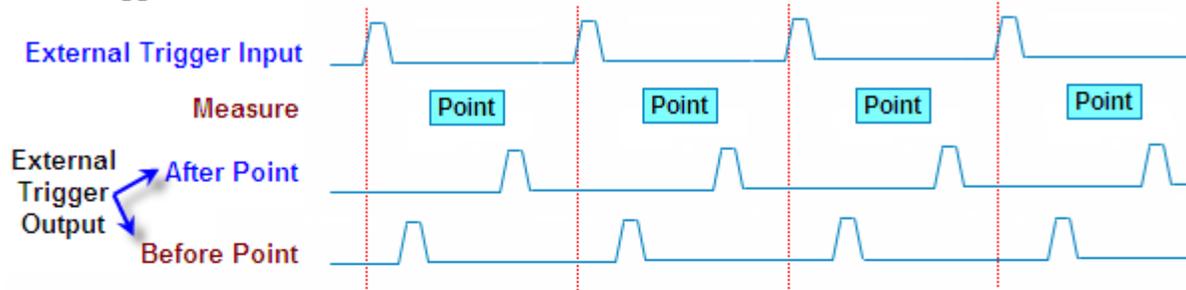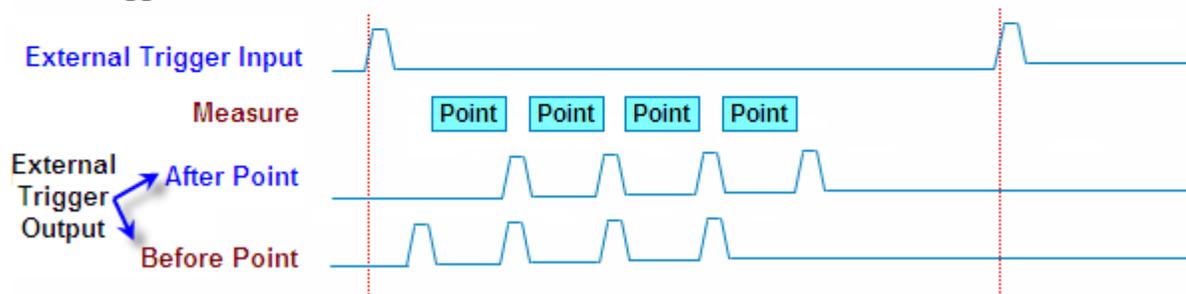
e5061b037

Making Trigger

- [Generating the Trigger](#)
- [Sweep Order in Each Channel](#)

Other topics about Making Measurement

Generating the Trigger

To make a measurement, it is necessary to generate a trigger by using the selected [trigger source](#). Once the internal trigger is selected, a series of triggers is continuously generated as soon as the setting becomes effective.

Pressing **Trigger** > **Restart** during a sweep forces the analyzer to abort the sweep.

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 accordingly.

Sweep Order	Stimulus Port	Updated Trace
1	Port 1	S11, S21
2	Port 2	S12, S22

- If full 2-port error correction is in effect, no trace between calibrated ports is updated until the last calibrated port is swept as the stimulus port.

- Sweep is not executed for stimulus ports that are not required for updating traces.

Analyzing Data

Analyzing Data

- [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](#)
- [Using Limit Test](#)
- [Using Bandwidth Test](#)
- [Using Ripple Test](#)
- [Using Point Limit](#)
- [Using 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 E5063A 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 have two marker response values each (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 of the marker response values (two values: main and auxiliary) can be selected from 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

To set up data formats, refer to Selecting a Data Format.

Activating the marker on the Trace

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the channel on which a marker is used.

2. 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.

3. 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.

- 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.
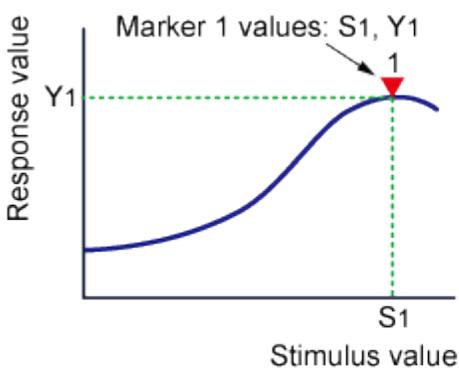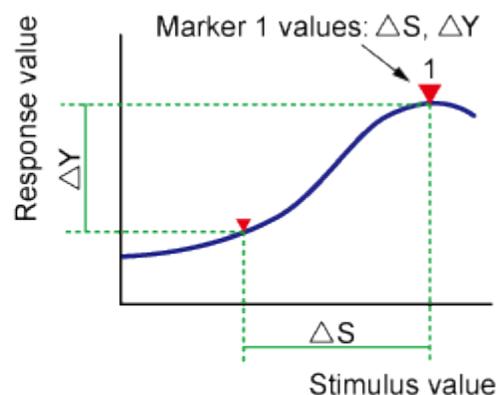
e5071c310

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** > **Marker -> Ref Marker** enables you to place the reference marker at the position of the currently active marker. The reference mode then turns 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's stimulus value is specified as a numerical value, the marker is placed at the measurement point closest to the specified value. A marker 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.
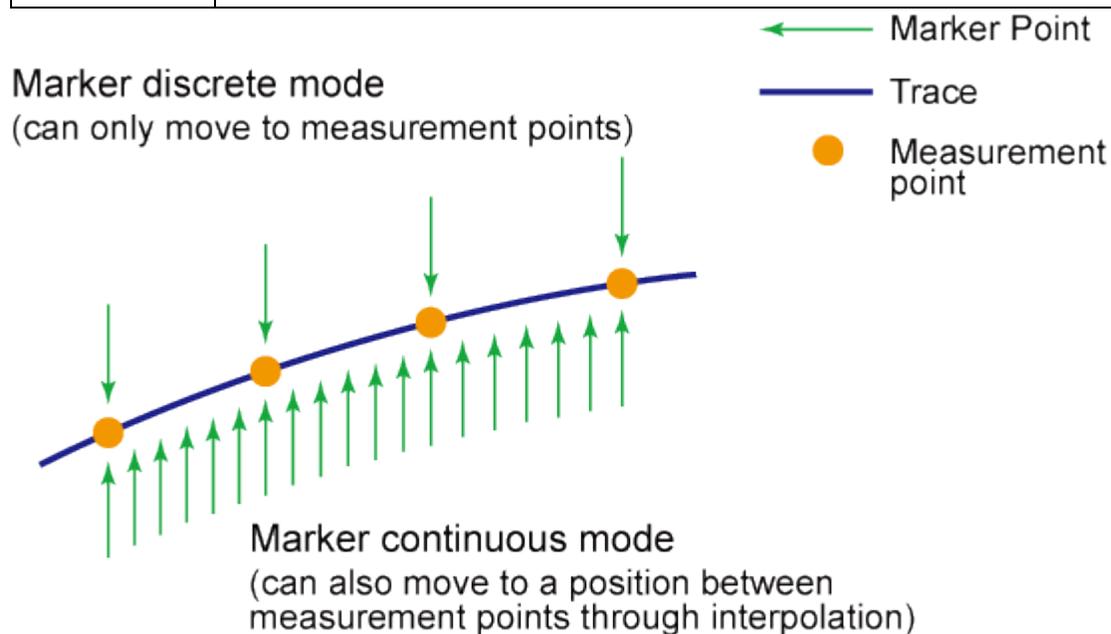
e5071c311

Turning Discrete Mode On or Off

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace on which the discrete mode is set up.

2. Press **Marker Fctn**.

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.
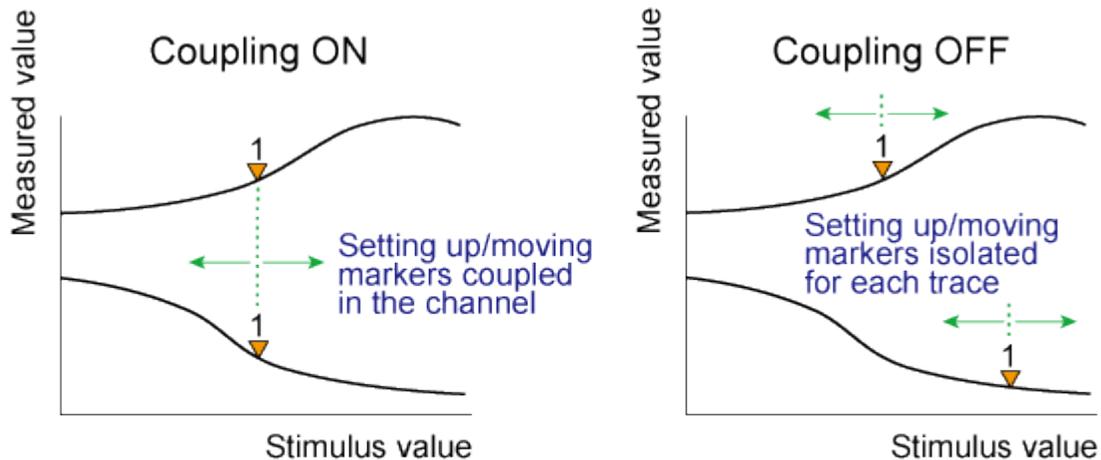
e5071c388

Value	Description
Marker Couple is ON (**Coupling ON**)	Markers are set up and moved in coupled operation on all the 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** (or **Channel Prev**) to activate the channel on which the marker couple is set.

2. Press **Marker Fctn**.

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
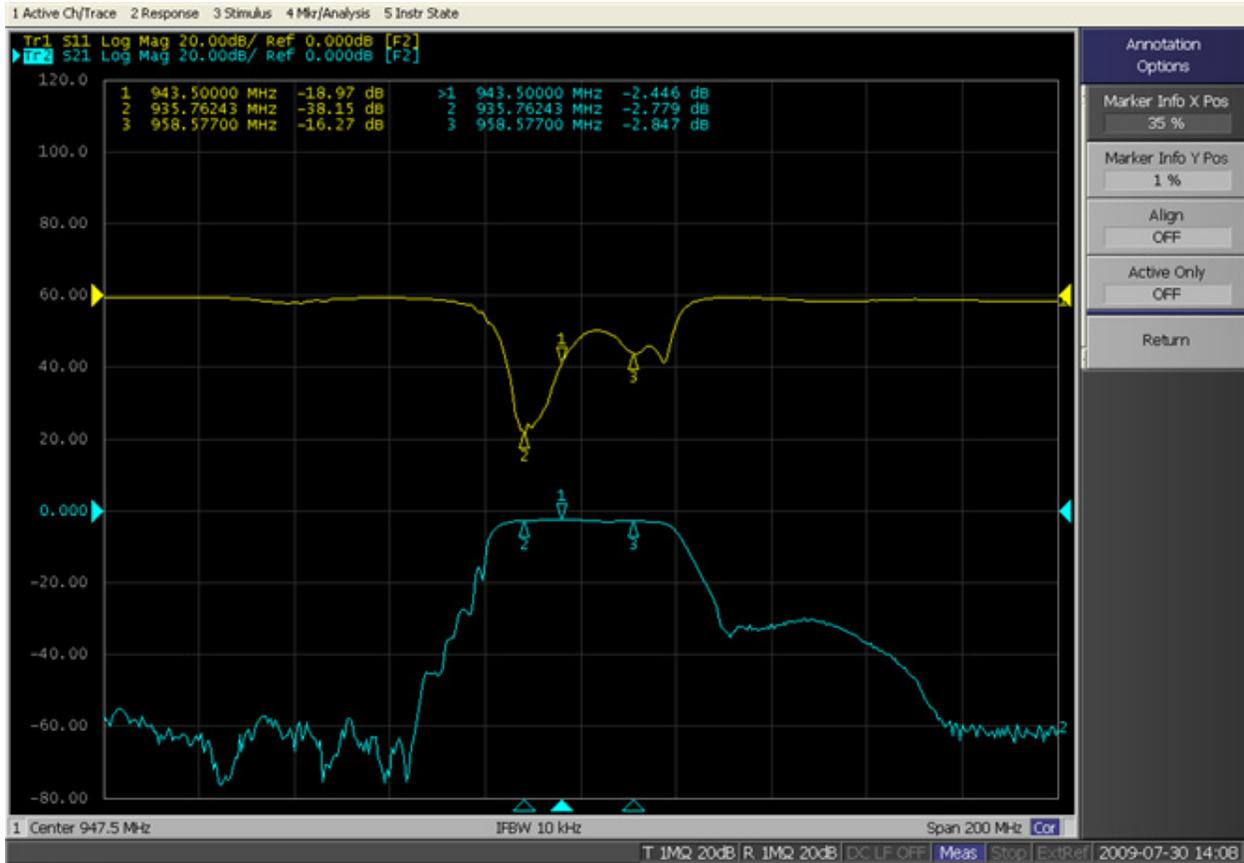
1. Press **Marker Fctn**.

2. Click **Marker Table** to turn ON the marker table display.

The marker table appears in the lower part of the screen.

Specifying Display Position of Marker Values

This section describes how to specify the marker value display position for each active trace.

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** (or **Channel Prev**) to activate the channel for which you want to set the marker coupling.

2. Press **Marker Fctn** > **Annotation Options**.

3. Click **Marker Info X Pos** to set the horizontal display position.

4. Click **Marker Info Y Pos** to set the vertical display position.

Aligning Marker Value Display

This section describes how to align maker value displays.
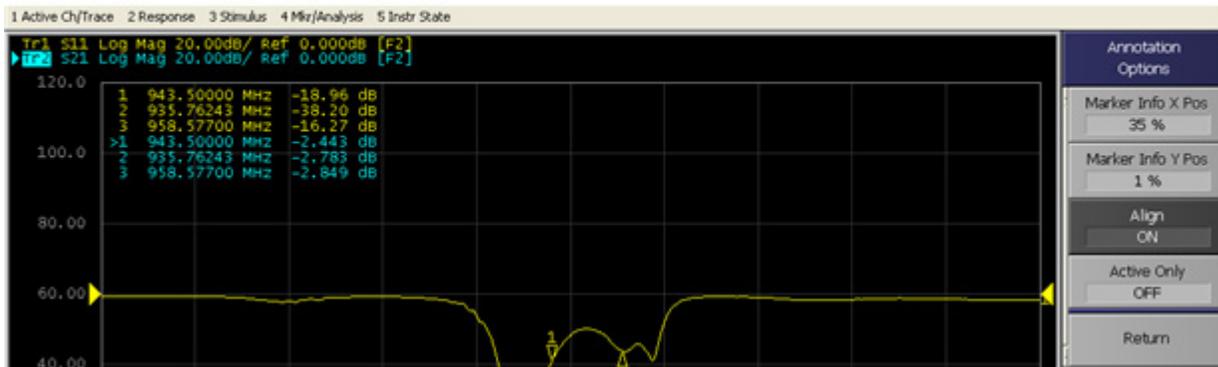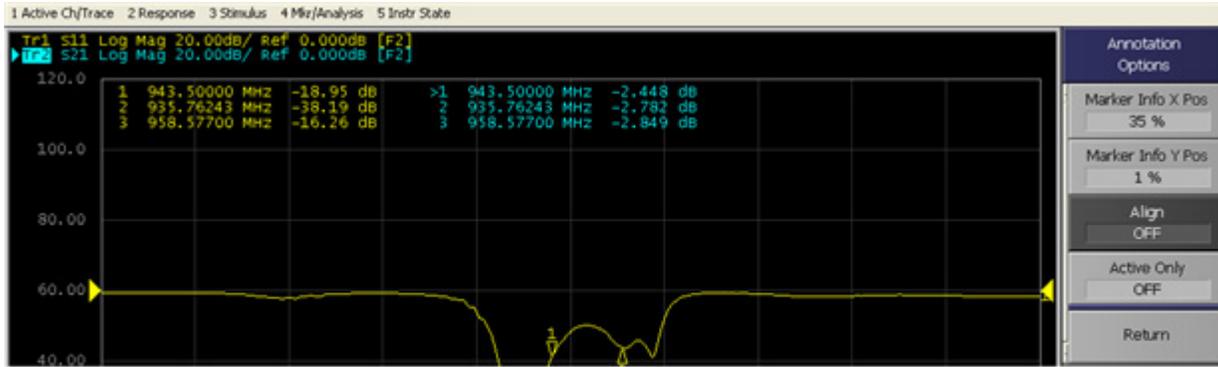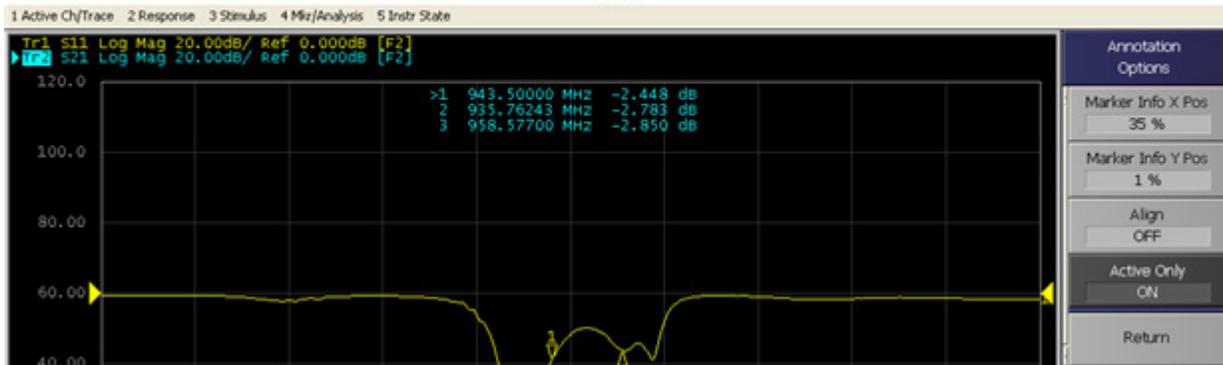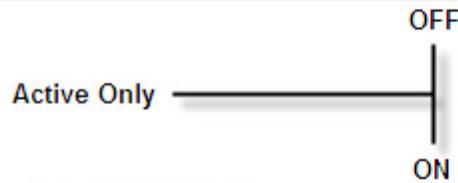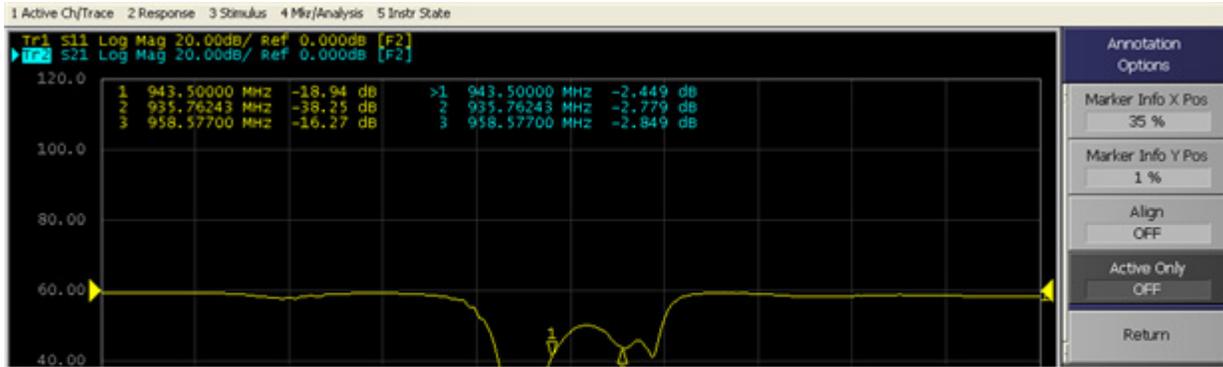
Value	Description
On (**Align ON**)	Displays marker values to align to the display position of trace 1.
Off (**Align OFF**)	Displays marker values in the display position defined for each trace.

1. Press **Marker Fctn** > **Annotation Options**.

2. Click **Align** to toggle ON/OFF.

Displaying All Marker Values for Displayed Traces

This section describes how to display all marker values for the displayed traces.

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** > **Annotation Options**.

2. Click **Active Only** to toggle ON/OFF.

Searching for Positions that Match Specified Criteria

- [Overview](#)
- [Setting Search Range](#)
    - [Setting Multiple Search Range](#)
- [Automatically Executing a Search (Search Tracking)](#)
- [Searching for Maximum and Minimum Values](#)
- [Searching for the Peak](#)
- [Searching for the Target Value (Target search)](#)

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](#)
- [Peak](#)

    - Maximum peak (for a positive peak), minimum peak (for a negative peak)
    - Peak on the left or right-hand side nearest to marker position
    - [Multiple Peak (Searching all peak points at one time)](#)

    - [Target (a point that has a target measurement value)](#)

        - Target nearest to the marker position
        - Target on the left or right-hand side nearest to marker position
        - [Multiple Target (Searching all target 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.

Procedure to Turn ON/OFF Trace Coupling within Search Range

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace for which you want to set the search range.

2. Press **Marker Search** > **Search Range**.

3. Click **Couple** to toggle ON/OFF trace coupling within the search range.

Procedure to Set Search Range

186

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace for which you want to set the search range.

2. Press **Marker Search** > **Search Range**.

3. Click **Search Range** to turn ON the Partial Search feature.

4. Click **Start**, then enter the start value (lower limit) of the search range.

5. Click **Stop**, then enter the stop value (upper limit) of the search range.

Setting Multiple Search 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 Search 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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace on which you want to set up the 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.
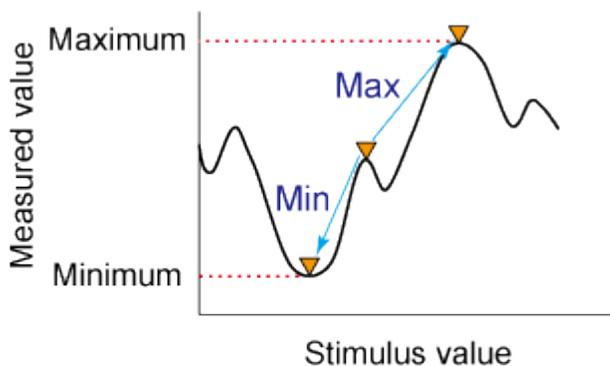
e5071c386

Search for maximum (**Max**)	Move active marker to point on the trace where measured value is the greatest
Search for minimum (**Min**)	Move active marker to point on the trace where measured value is the 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.

- When the data format is in Smith chart or polar format, execute the search only for the main response value.

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 of which the 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 differences in magnitude from the measurement points on either side of it.

Positive peak (**Positive**)	A peak of which the measured value is greater than the measurement points on either side of it (peak polarity: positive)
Negative	A peak of which the measured value is smaller than the

| peak (**Negative**) | measurement points on either side of it (peak polarity: negative) |

About Peak Excursion Value

The peak excursion value is smaller among the differences in the measured values from the adjoining peaks of the opposite polarity.
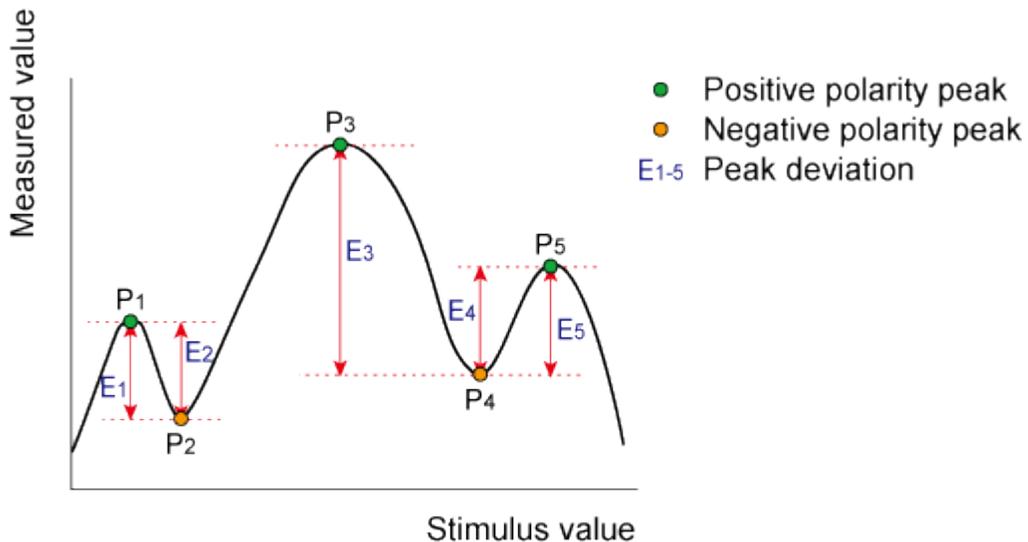

e5071c390

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.
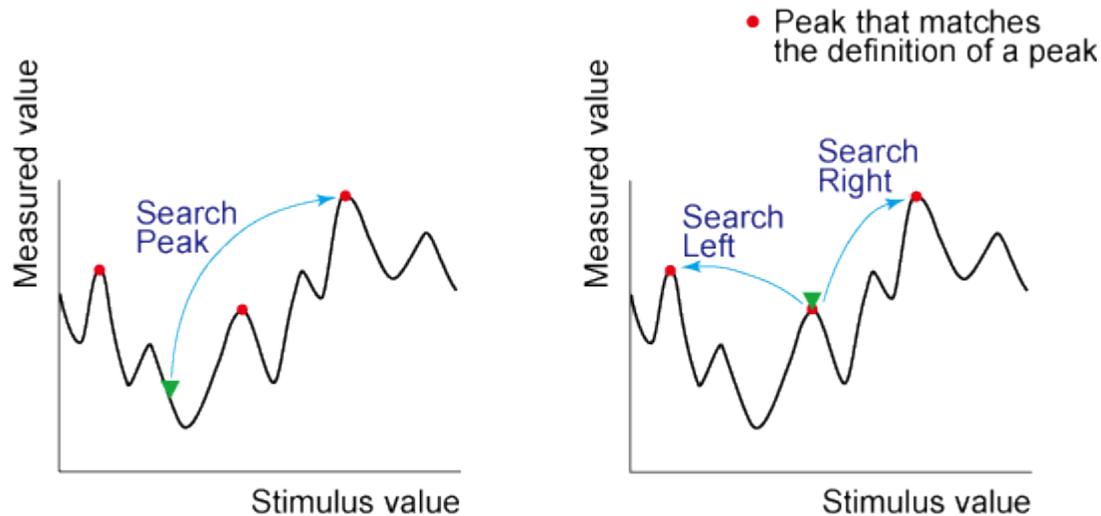
e5071c452

Procedure

1. [Activate the marker](#) you are using for the peak search.
2. Press **Marker Search** key.
3. Click **Peak** > **Peak Excursion**.
4. Enter the lower limit for the peak excursion value. This sets the peak search to be executed based on the definitions of the newly set lower limit for the peak excursion value and the currently set peak polarity.
5. Click **Peak Polarity**, then select a Peak Polarity.
6. Select a peak polarity. This sets the peak search to be executed based on the definitions of the currently set lower limit for the peak excursion value and the newly set peak polarity.
7. Click the corresponding softkey to move the marker to the peak.
8. 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.

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. Click **Peak Polarity**, then select a Peak Polarity.

3. Click **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 Search Range is not applied.

NOTE   Search tracking is valid for multiple peak search.

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 immediately precedes it (on the left side)

Positive target transition type
Negative target transition type
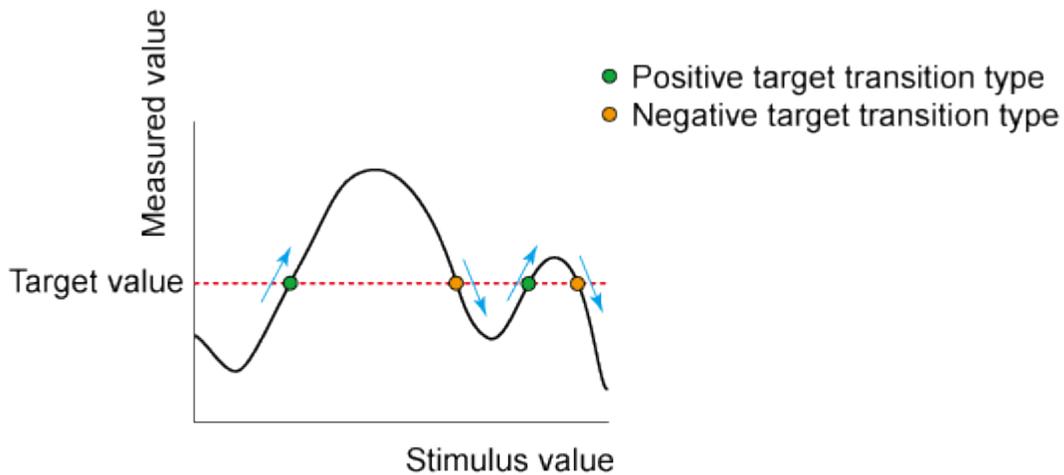
e5071c389

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 encountered target.
Search right (**Search Right**)	Executes the search from the current marker position to the larger stimulus values and moves the marker to first encountered target.
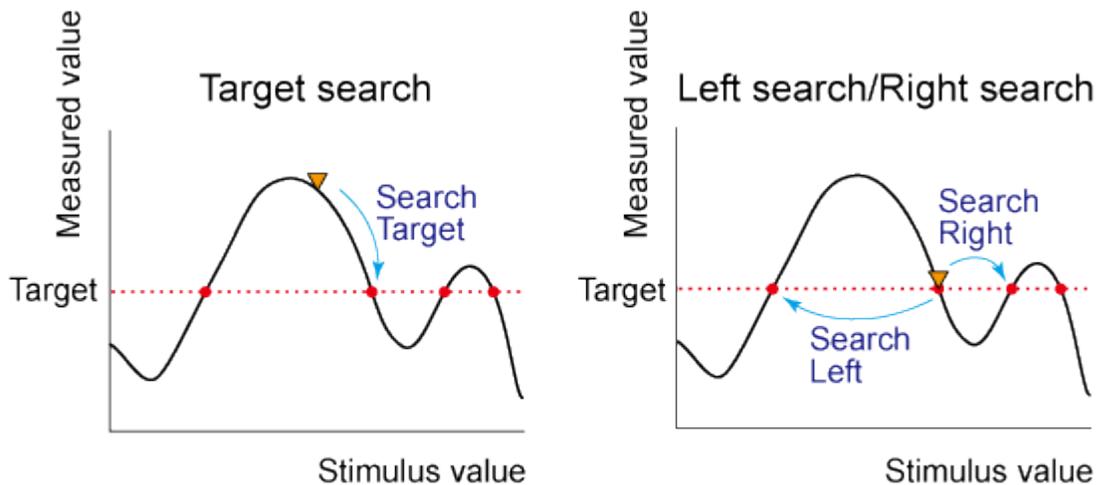
e5071c387

Procedure

1. <u>Activate the marker</u> you are using for the target search.

2. Press **Marker Search** key.

3. Click **Target** > **Target Value**

4. Enter the <u>target value</u> in the entry area that appears. This causes the target search to be executed based on the definitions of the newly set target value and the currently set transition type.

5. Click **Target Transition**.

6. Select a transition type. This sets the target search to be executed based on the definitions of the currently set target value and the newly set transition type.

7. Press the corresponding softkey to move the marker to the target.

- 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.

Searching all target points at one time (Multiple Target Search)

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 Search 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 below.

$$Q = \frac{cent}{BW}$$
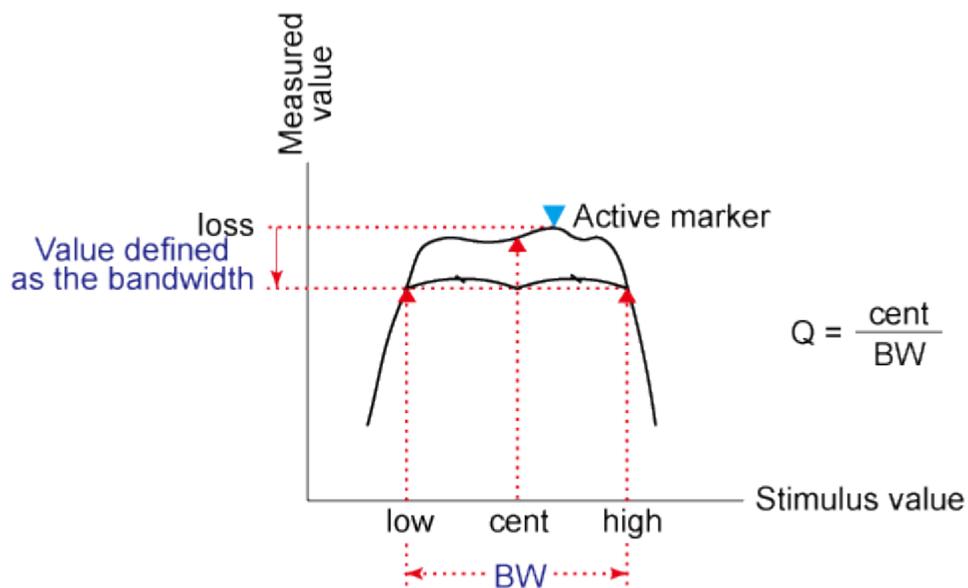
e5071c392

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 within two measurement points, both separated by the defined bandwidth value from the active marker position.
Higher frequency cut-off point (*high*)	Highest frequency within two measurement points, both separated by the defined bandwidth value from the active marker position.
Center frequency	Frequency at the midpoint between the lower

(*cent*)	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** > **Bandwidth Value**.

3. 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
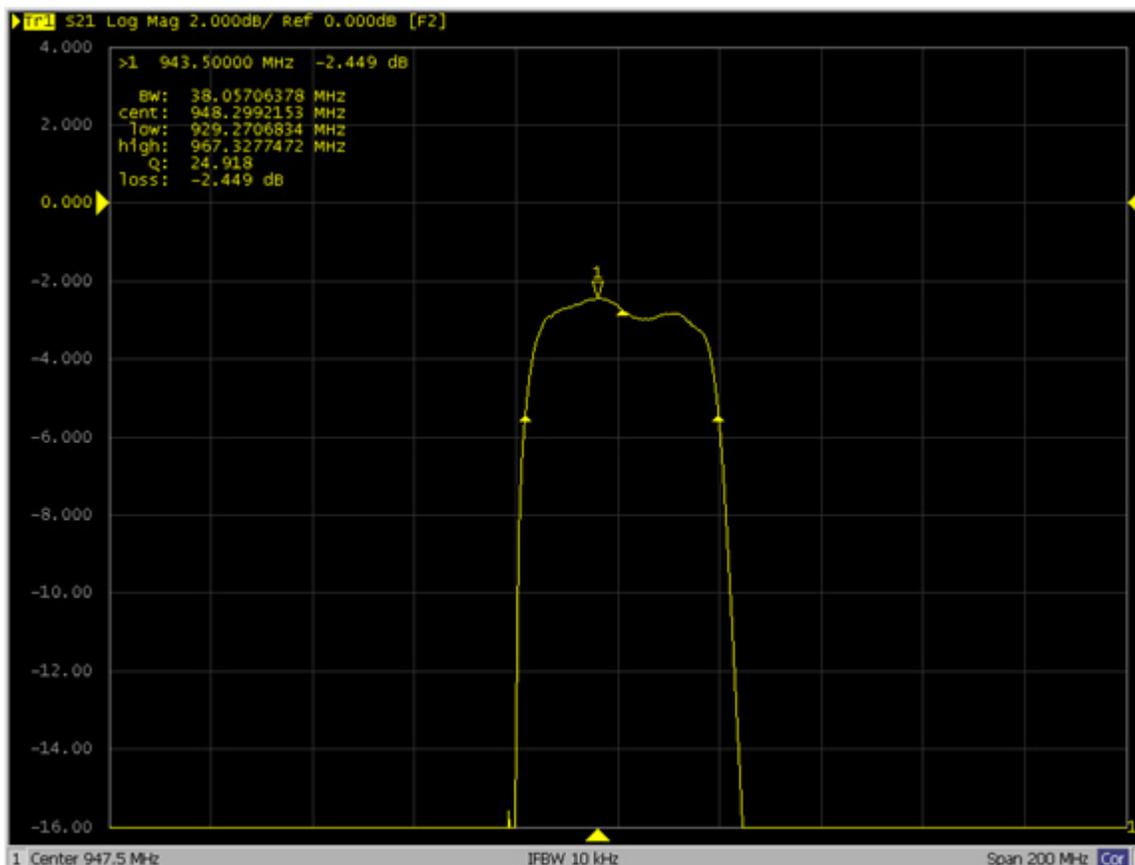
Determining the Bandwidth of the Trace (Notch Search)

- [Overview](#)
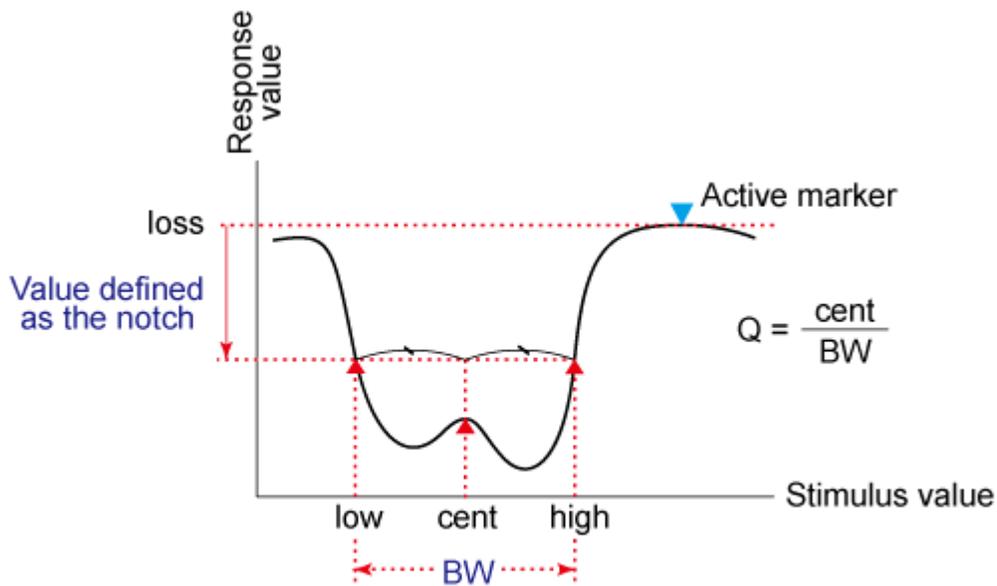- [Executing a Notch Search](#)

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 the points meet the conditions.

The figure and 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** > **Notch Value** and enter the notch value in the entry area that appears.

3. 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).
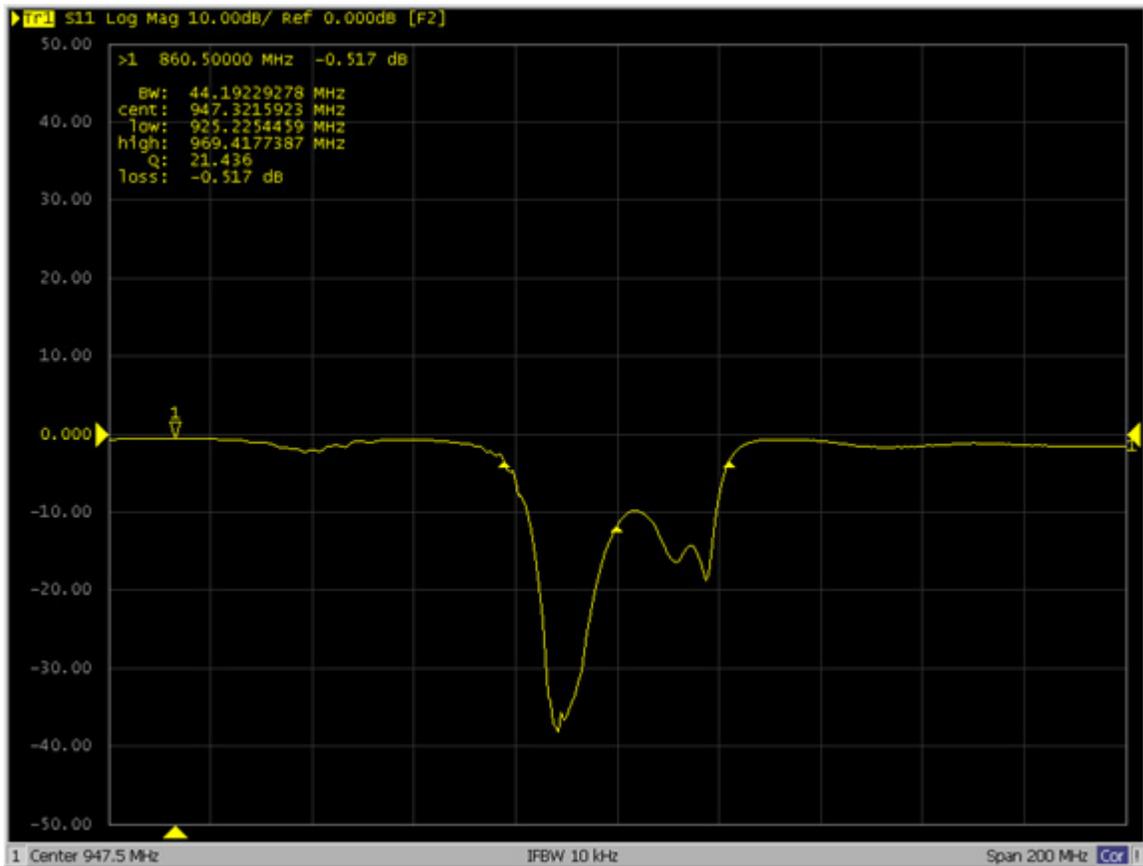
4. For more information on displaying the notch search result, see Notch Search
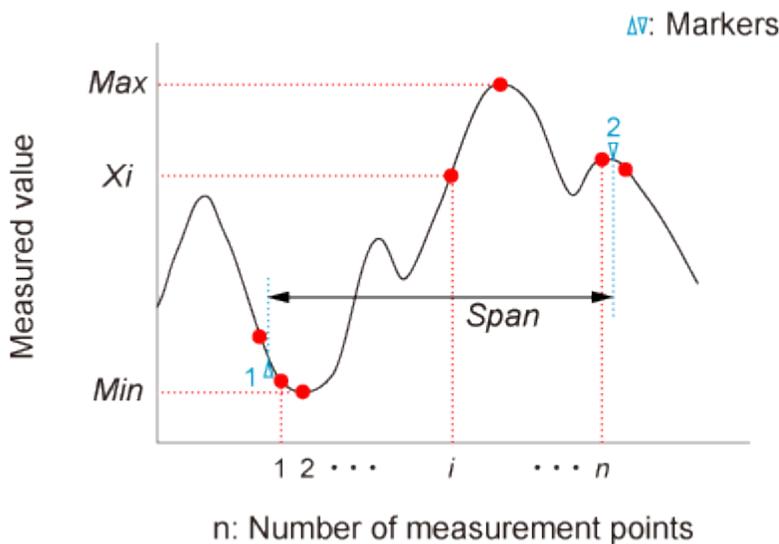
Determining the Mean, Standard Deviation, and p-p of the Trace

- 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. It is calculated within the marker search range.

e5061b063

Statistics data element	Definition
Mean (*mean*)	$$\frac{\sum_{i=1}^{n} x_i}{n}$$ (n: number of points between markers 1 and 2; x_i: measured value at the i-th measurement point between markers 1 and 2.)
Standard deviation (*s. dev*)	$$\sqrt{\frac{\sum_{i=1}^{n} (x_i - mean)^2}{n-1}}$$ (n: number of points between markers 1 and 2; x_i:

	measured value at the i-th measurement point between markers 1 and 2.; mean: Mean)
Peak-to-peak (*p - p*)	Max - Min (Max: greatest measured value between markers 1 and 2.; Min: smallest measured value between markers 1 and 2.)

- The search range does not affect this result.

Displaying Statistical Data

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace of which the statistical data is required.

2. Press **Marker Fctn** > **Statistics** to turn ON the display of statistics data.

3. The markers 1 and 2 are activated automatically. Move markers 1 and 2 to the position of the measurement.

Comparing Traces/Performing Data Math

- [Overview](#)
- [Performing Data Math Operations](#)

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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace to be stored in memory.
2. Press **Display** > **Data -> Mem** to store the measured data in memory.
3. Press **Data Math**.
4. Select the **data math** operation to perform.
5. Press **Display**.
6. Select the **type of data** to display ON the screen.
7. Send the trigger to make measurements.

Performing Parameter Conversion of Measurement Results

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

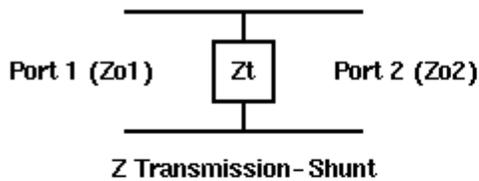$$Z_t = \frac{2 \times \sqrt{Z_{0a} \times Z_{0b}}}{S_{ab}} - (Z_{0a} + Z_{0b}), \ Y_t = \frac{1}{Z_t}$$

- Inverse S-parameter ($1/S_{ab}$)

  where:

  Z_{0a}: Characteristic impedance of port a
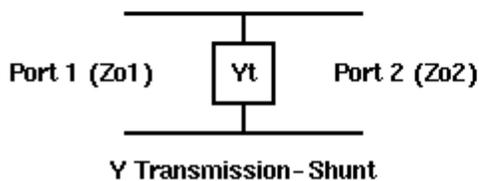
  Z_{0b}: Characteristic impedance of port b

- 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 Z_0 (preset value: 50 Ω) is used.

Turning Conversion ON

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate a trace of which you want to use the marker.

2. Press **Analysis** > **Conversion**.

3. Click **Conversion** to turn ON the conversion function.

Selecting Conversion Target Parameter

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate a trace of which you want to use the marker.

2. Press **Analysis** > **Conversion** > **Function**.

3. Select 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.

Using 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 exceeds, 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.

- The targets of the pass/fail judgment are measurement points only. Parts interpolated between the 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 with the following methods.

- Beep that occurs when the judgment result fails.
- Using the status register.

Displaying Judgment 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 section of the graph.
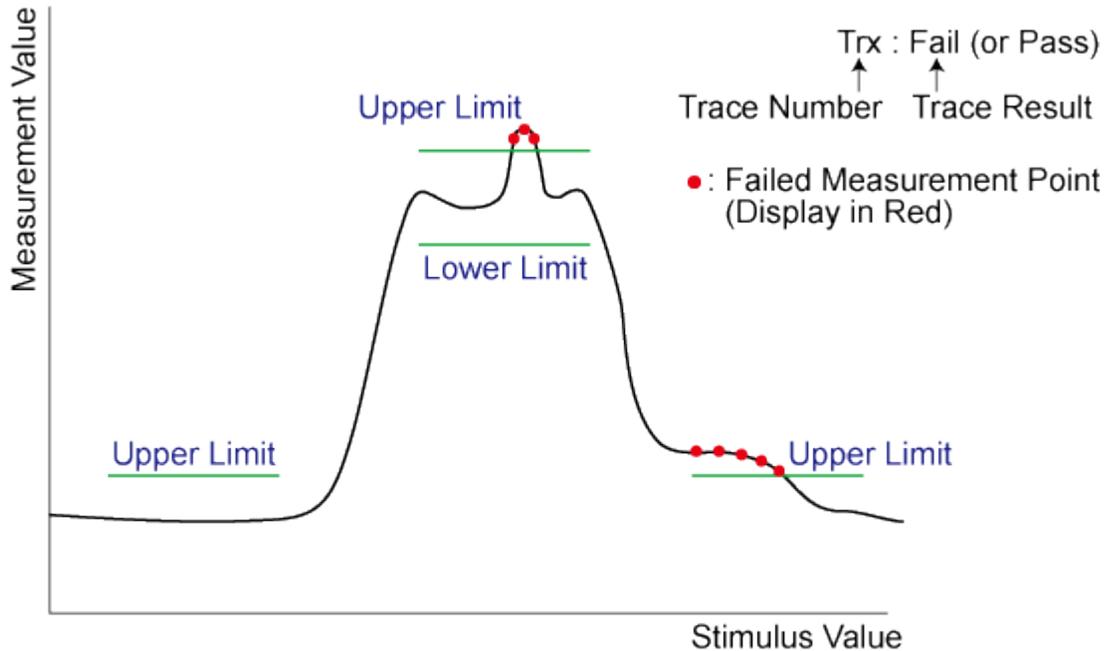
e5071c446

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 fail if one or more unsatisfactory trace exist in any of the limit test, the ripple test or the bandwidth test within the channel.)

Follow these steps to turn ON/OFF the display of the channel fail message.

1. Press **Analysis** > **Limit Test**.
2. 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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace on which the limit test function is used.

2. Press **Analysis** > **Limit Test**.

3. Click **Edit Limit Line** to display the limit table.

4. 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.

5. 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	

- 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 E5063A does not affect the stimulus value.

- 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.
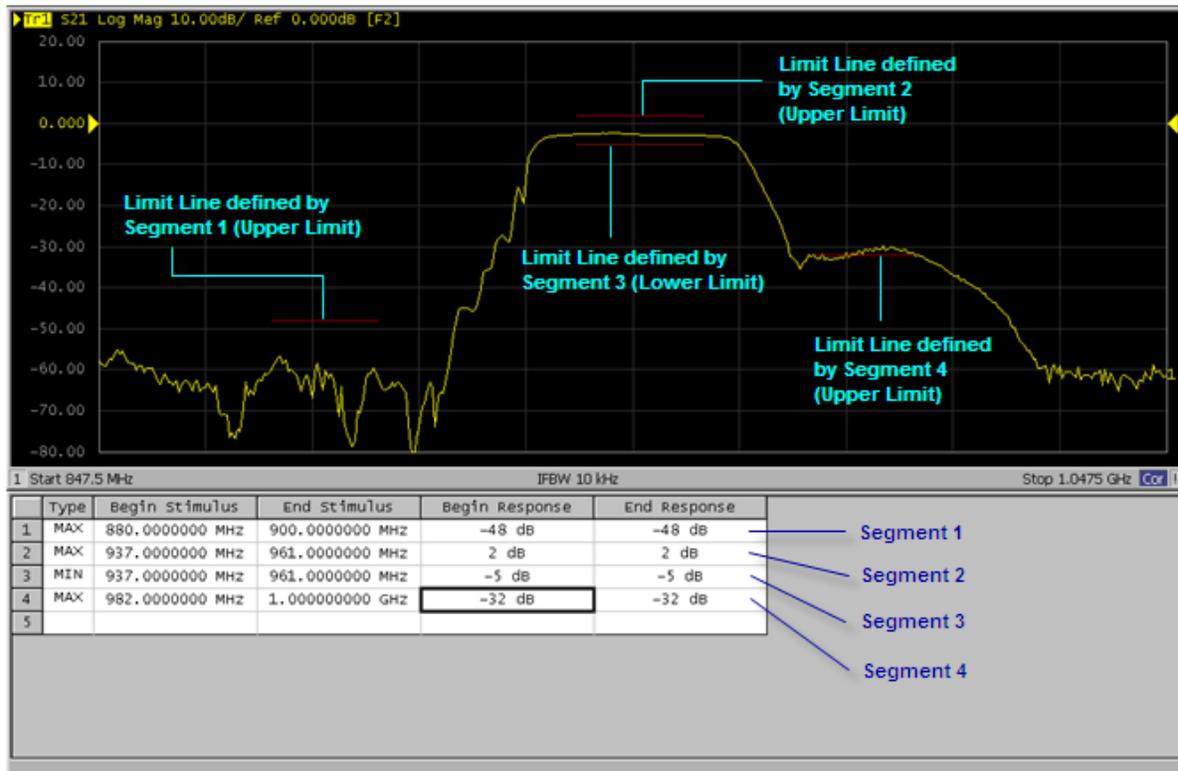
- You can define a limit line that is able to freely overlap the stimulus range of another limit line.

  Defining one limit line that has 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 of which the type is set to maximum (MAX) exist, the smallest one is used as the maximum.

    - When two or more limit values of which the type is set to minimum (MIN) exist, the largest one is used as the minimum.

- Even if the span of the sweep range on the E5063A is set to 0, enter the two parameters of Begin Stimulus and End Stimulus.

- 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 is saved as strings that includes its unit).
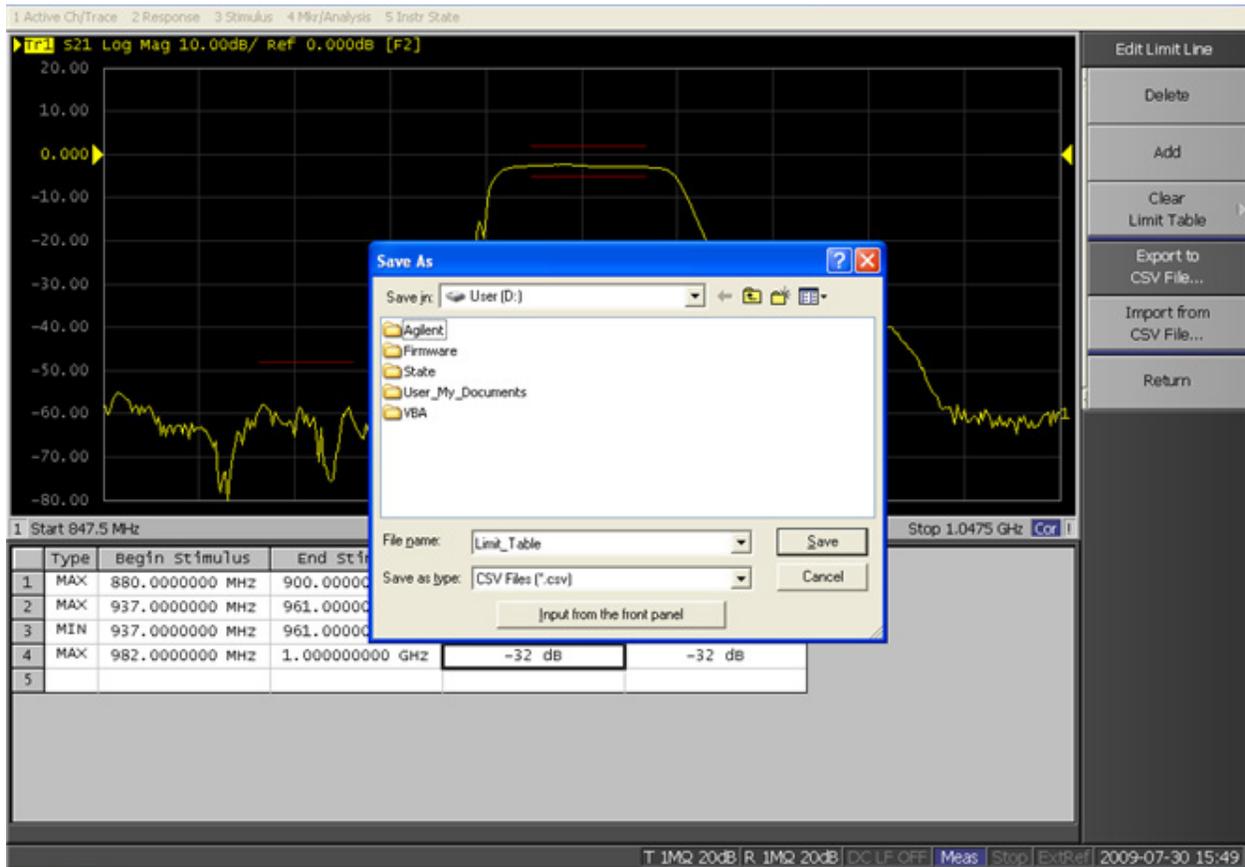
1. Display the limit table.

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 of the channel or trace.

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 is valid when the saved file is the output.

- On the second line, the trace number of the active trace that is valid when the saved file is the 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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace on which the limit test function is used.

2. Press **Analysis** > **Limit Test** to display the Limit Test menu.

3. Press **Limit Test** to set the limit test ON/OFF.

4. Press **Limit Line** to set the limit line display ON/OFF.

Adding Offset to Limit Line

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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the channel/trace on which the limit test function is used.

2. Press **Analysis** > **Limit Test** to display the softkeys for the limit test.

3. Click **Limit Line Offsets** to display the **limit line offset function menu**.

Stimulus offset
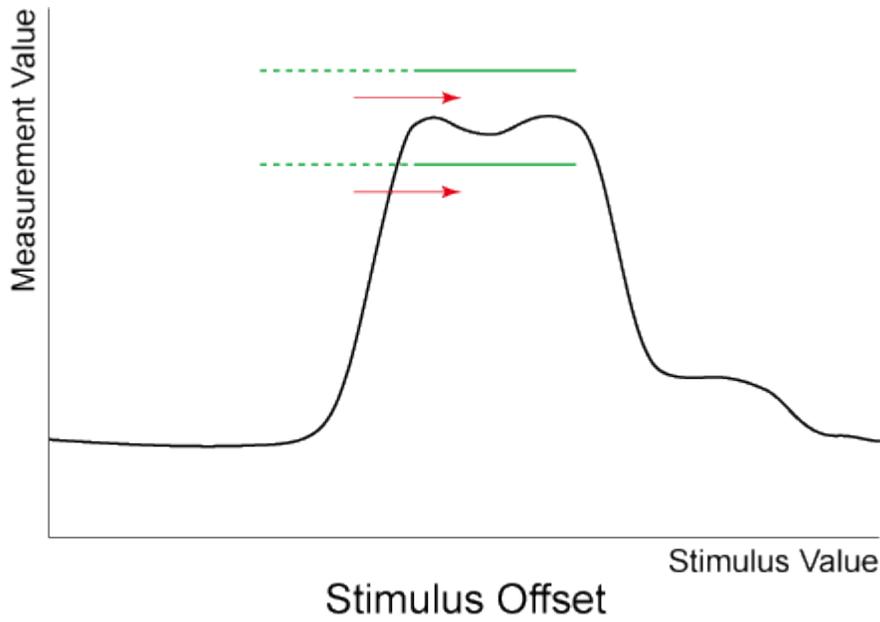
e5071c447

Amplitude offset
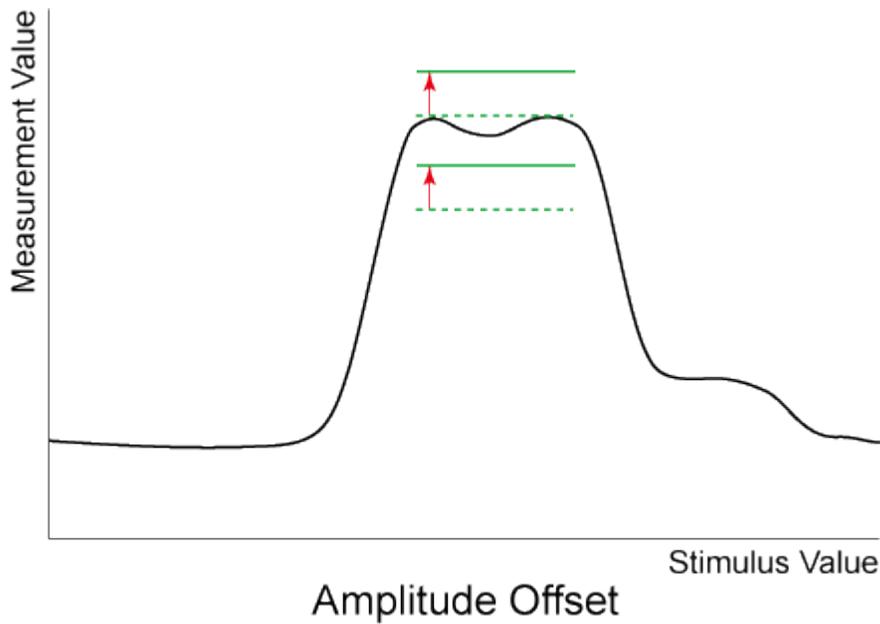
e5071c448

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

Using 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 to test 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.
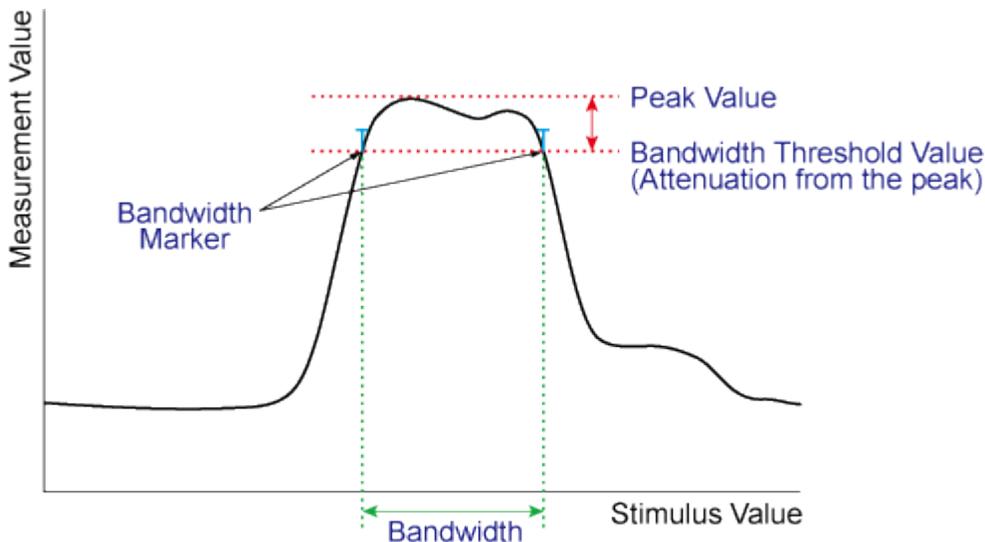
e5071c450

Displaying Bandwidth Test Results

Test Result for Trace

The test result of the trace is indicated in the upper-right section 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 choose to display the bandwidth value.

For information on how to display the results, see Turning On/Off Bandwidth Test and Displaying Results.
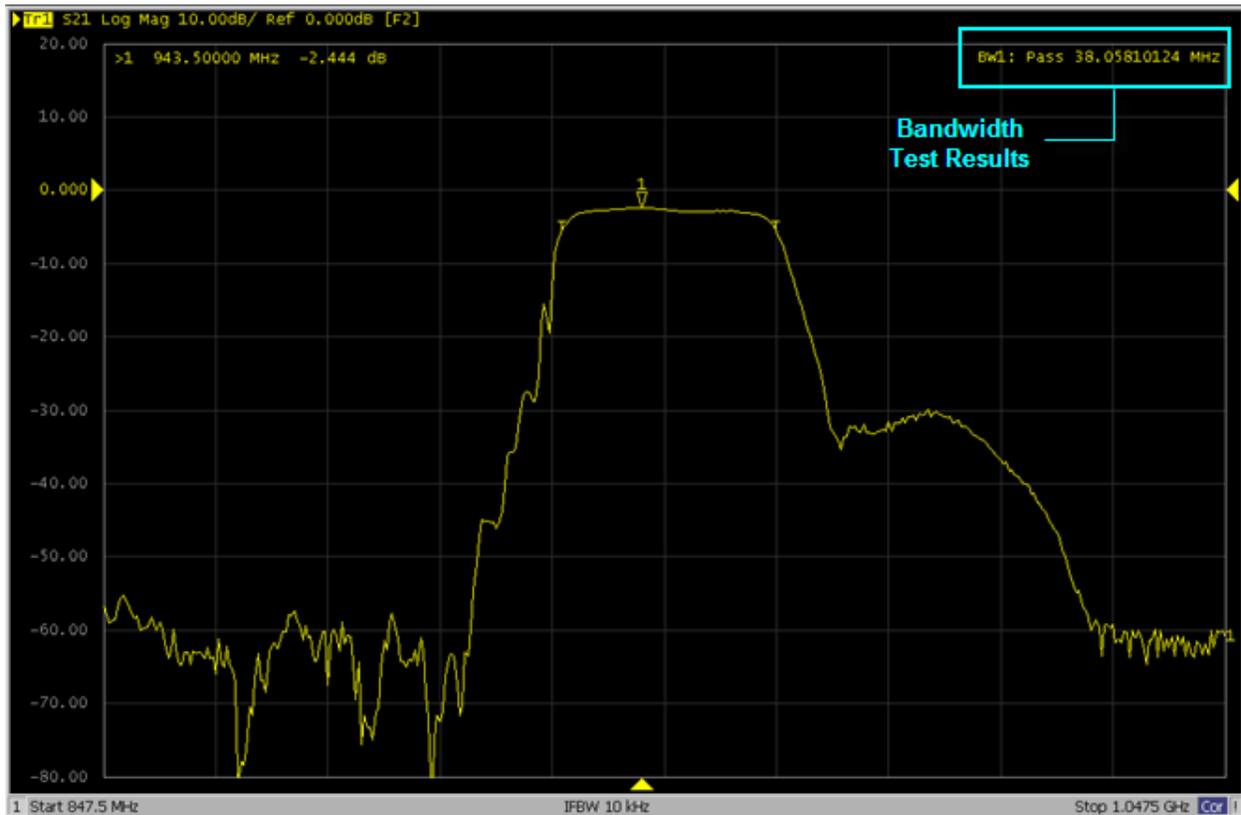
Test Results for Channel

If any channel is unsatisfactory, a message is displayed as shown in Judgment result of channels. (It is 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 that the result is 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.

Follow the steps below to set up the bandwidth test.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace for which you want to apply the bandwidth test function.

2. Press **Analysis** key to display the Analysis menu.

3. Click **Bandwidth Limit** > **N dB Points** to specify the bandwidth threshold. The unit is dB.

4. 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.

5. 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.

Follow the steps below to configure the ON/OFF setting of the bandwidth test.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to activate the trace of which you want to apply the bandwidth test function.

2. Press **Analysis** key to display the Analysis menu.

3. Click **Bandwidth Limit** > **BW Test** to turn ON the bandwidth test.

4. Click **BW Marker** to turn ON so that the bandwidth marker is displayed on the screen.

5. Click **BW Display** to turn ON so that the bandwidth value is displayed.

Using Ripple Test

Other topics about Data Analysis

Overview

Independent 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 does not exceed 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."

- 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 is indicated in red on the screen, and the trace's test results based on the results of each measurement point is 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 is displayed in red on the screen. The test result for the trace is indicated as Pass or Fail in the upper-right section of the graph. You can also display the ripple value at the selected frequency band.

The result is displayed as Ripln:Pass (or Fail) for each trace. n denotes the trace number. Bn is followed by the ripple value (if the ripple display is turned OFF, only Bn is 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 B1 is the ripple value at the first 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.

e5063a099
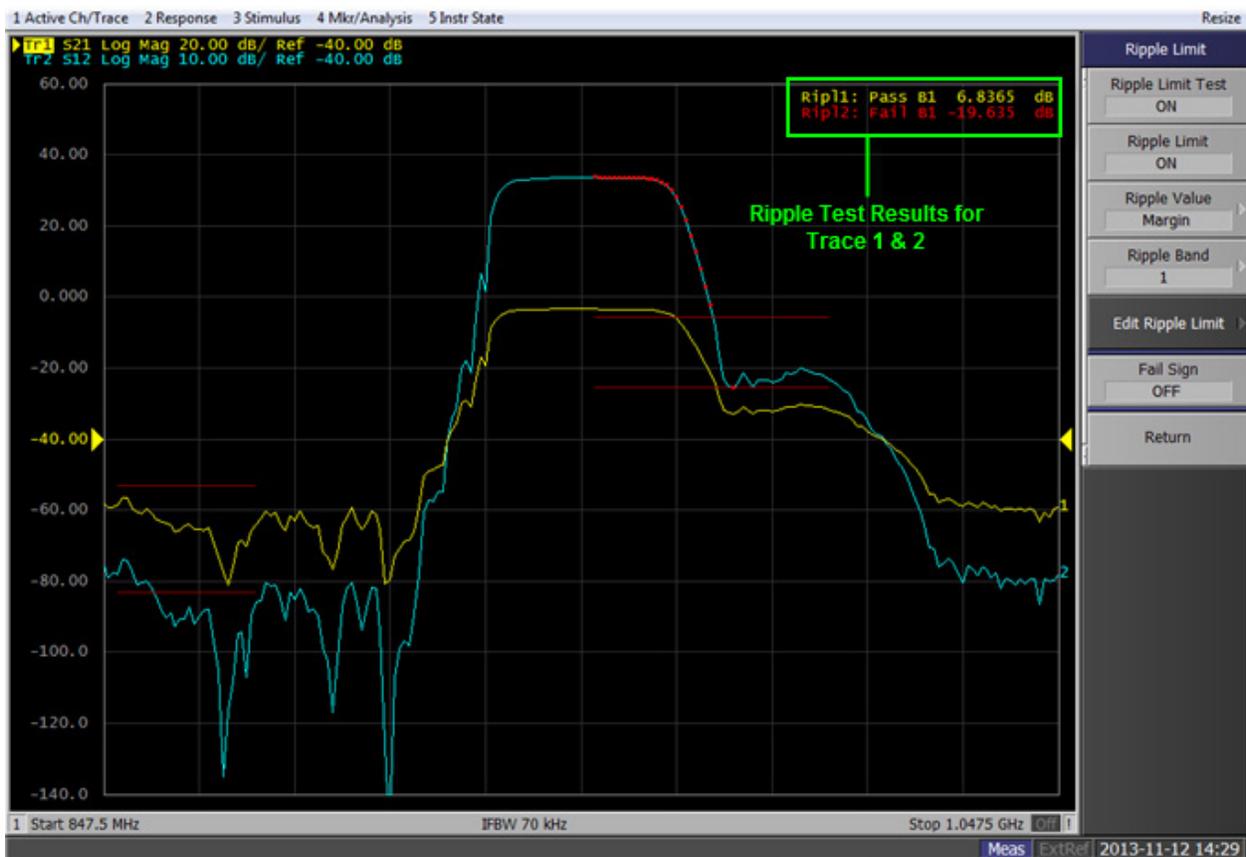
Test result for channel

If a channel has a judgment result of "fail", the message is displayed as shown in Test Result for Trace. (It is judged as failed if one or more failed traces are found in the limit test, ripple test, or bandwidth test within the channel.)

	Type	Begin Stimulus	End Stimulus	MaxRipple
1	ON	933.0000000 MHz	964.0000000 MHz	1.1 dB
2	ON	938.0000000 MHz	953.0000000 MHz	500 mdB
3	OFF	953.0000000 MHz	960.0000000 MHz	300 mdB
4	▾			

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** > **Ripple Limit**.

2. 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 is "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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace on which you want to apply the ripple test function.

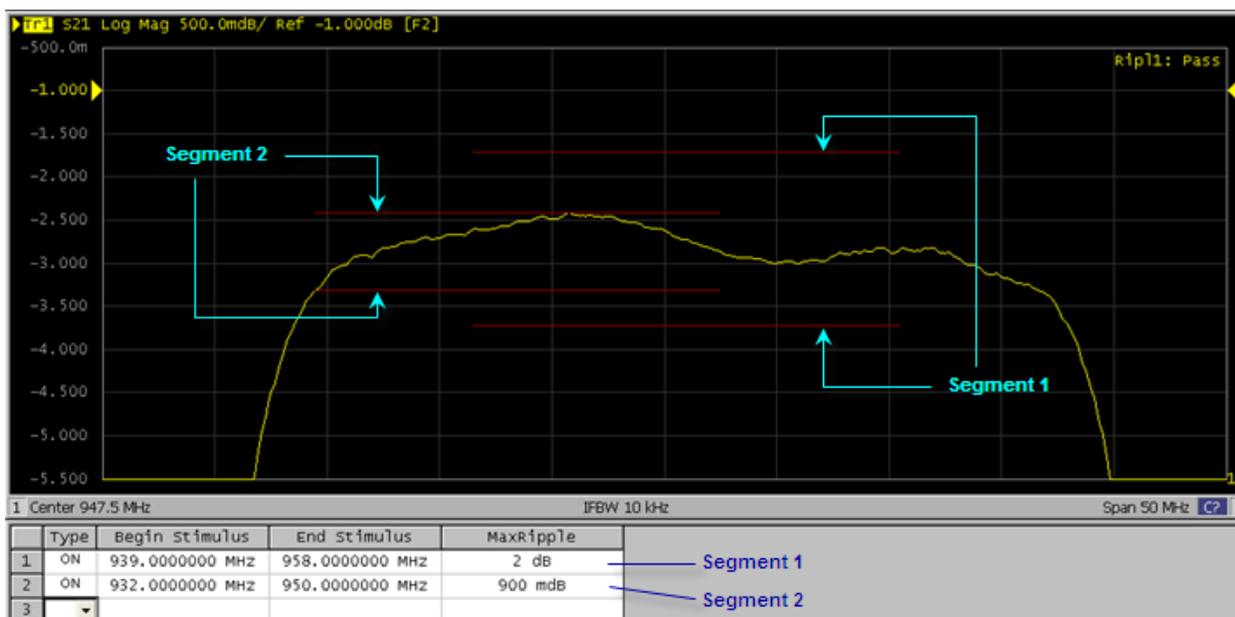2. Press **Analysis** > **Ripple Limit**.

3. Click **Edit Ripple Limit** to display the ripple limit table shown below.

	Type	Begin Stimulus	End Stimulus	MaxRipple
1	ON	933.0000000 MHz	964.0000000 MHz	1.1 dB
2	ON	938.0000000 MHz	953.0000000 MHz	500 mdB
3	OFF	953.0000000 MHz	960.0000000 MHz	300 mdB
4	▼			

(Labels: Frequency Band; Type of Frequency Band; Beginning Point of Stimulus Value; End Point of Stimulus Value; Ripple Limit Value)

4. Click **Add** to add a frequency band to the ripple limit table and then specify the **parameters** for the frequency band. Note that no frequency band is provided in the ripple limit table by default. Click **Delete** to delete the selected frequency band and **Clear Ripple Limit Table** to clear the table setting.

- Acceptable range for the stimulus value: -500G to +500G. If any outranging value is specified, it is reset to fall within the range.

- Even if the E5063A's sweep range is changed after the stimulus value has been set, the stimulus value is not susceptible.

Example of ripple limit configuration

	Type	Begin Stimulus	End Stimulus	MaxRipple	
1	ON	939.0000000 MHz	958.0000000 MHz	2 dB	Segment 1
2	ON	932.0000000 MHz	950.0000000 MHz	900 mdB	Segment 2
3	▼				

- 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.

222

- Even if the E5063A's span value is set to zero, you must enter a parameter for both Begin Stimulus and End Stimulus.
- 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 file (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 file (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.
6. You can recall a limit table from a trace on any channel independently of the channel and trace that are active when the limit table is 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 onwards.
- From the fourth line onwards, 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** (or **Channel Prev**) and **Trace Next** (or **Trace Prev**) to select the trace on which you want to apply the ripple test function.

2. Press **Analysis** > **Ripple Limit**.

3. Press **Ripple Limit Test** to set the ripple test ON.

4. Press **Ripple Limit** to display the ripple limit line.

5. Press **Ripple Value** to shows how the ripple values are displayed, then select **Absolute** (difference between maximum and minimum values within the band), **Margin** (difference between absolute value of ripple and ripple limit), or **OFF**.

6. Press **Ripple Value Band** to select the band for which you want to display the ripple value.

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 Ripple Limit Table** > **OK** is clicked in the Edit Ripple Limit menu

Using Point Limit

-

-

-

Overview

The point limit function allows you to define the upper and lower limits at a certain frequency point. Limits are defined in segments, where each segment is a portion of the stimulus span. Each limit segment has an upper and a lower starting limit value.

This function compares the measured data with the defined limits, and provides pass or fail information for each measured data point.

Turning ON/OFF Point Limit

Follow the steps below to set the point limit segment and turn it ON/OFF:

1. Press **Analysis** > **Point Limit** > **Edit Point Limit**.

2. At the entry table at the bottom of the screen, enter the stimulus, lower limit and upper limit value of the desired segment.

3. Click Type **ON/OFF** to toggle between turning ON and OFF the segment.

To enter a new segment:

1. Click **Edit Point Limit** > **Add**.

2. A new row is added to the entry table with information identical to the previous row.

3. Change the stimulus, lower limit, upper limit value and type of the desired segment.

To delete a segment:

1. Click or highlight the segment to be deleted.

2. Click **Edit Point Limit** > **Delete**.

To delete all the point limit segments:

1. Click **Edit Point Limit** > **Clear Point Limit Table** > **OK**.

2. A new row is added to the entry table with information identical to the previous row.

The point limit and limit test can be used simultaneously or independently. If the **Point Limit** is turned ON and the **Point Limit Test** is turned OFF, the point

limit segments (segments with TYPE set to ON) are displayed on the screen for visual comparison and adjustment of the measurement trace. However, no pass or fail status is provided. If the **Point Limit** is turned OFF and **Point Limit Test** is turned ON, the specified limits are still valid and the pass or fail status is indicated even though the point limit lines are not displayed on the screen. If both **Point Limit** and **Point Limit Test** are turned ON, the point limit segments (segments with TYPE set to ON) and the pass or fail status are displayed on the screen.

When the **Point Limit** > **Fail Sign** is set to **ON**, a fail sign is displayed at the centre of the screen if the measurement data point is out of the specified segment range and the test has failed.

> **NOTE** If the stimulus of the point limit test is not the same as the measurement points, the closest measurement point is used for judgement. In such cases, interpolation between the measurement points is not applied. However, if the stimulus of the point limit test is the center of the two measurement points, the measurement point on the right side is used for judgement.

> **NOTE** If the stimulus of the point limit test is out of the measurement span, it will not be considered.

Importing/Exporting Point Limit Data File

The test point limit data can be saved in *.csv format and recalled as well.

To export the test point limit data:

1. Click **Edit Point Limit** > **Export to CSV File...**
2. Default file name is LimitXX.csv.
3. Click **Save** to save the file at your preferred location.

To import the test point limit data:

1. Click **Edit Point Limit** > **Import from CSV File...**
2. Select the desired data file.
3. Click **Open** to open and load the data file.

Using 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 E5063A. 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 E5063A 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.

Using Equation Editor

The step-by-step procedure of using Equation Editor is described below:

e5063a086
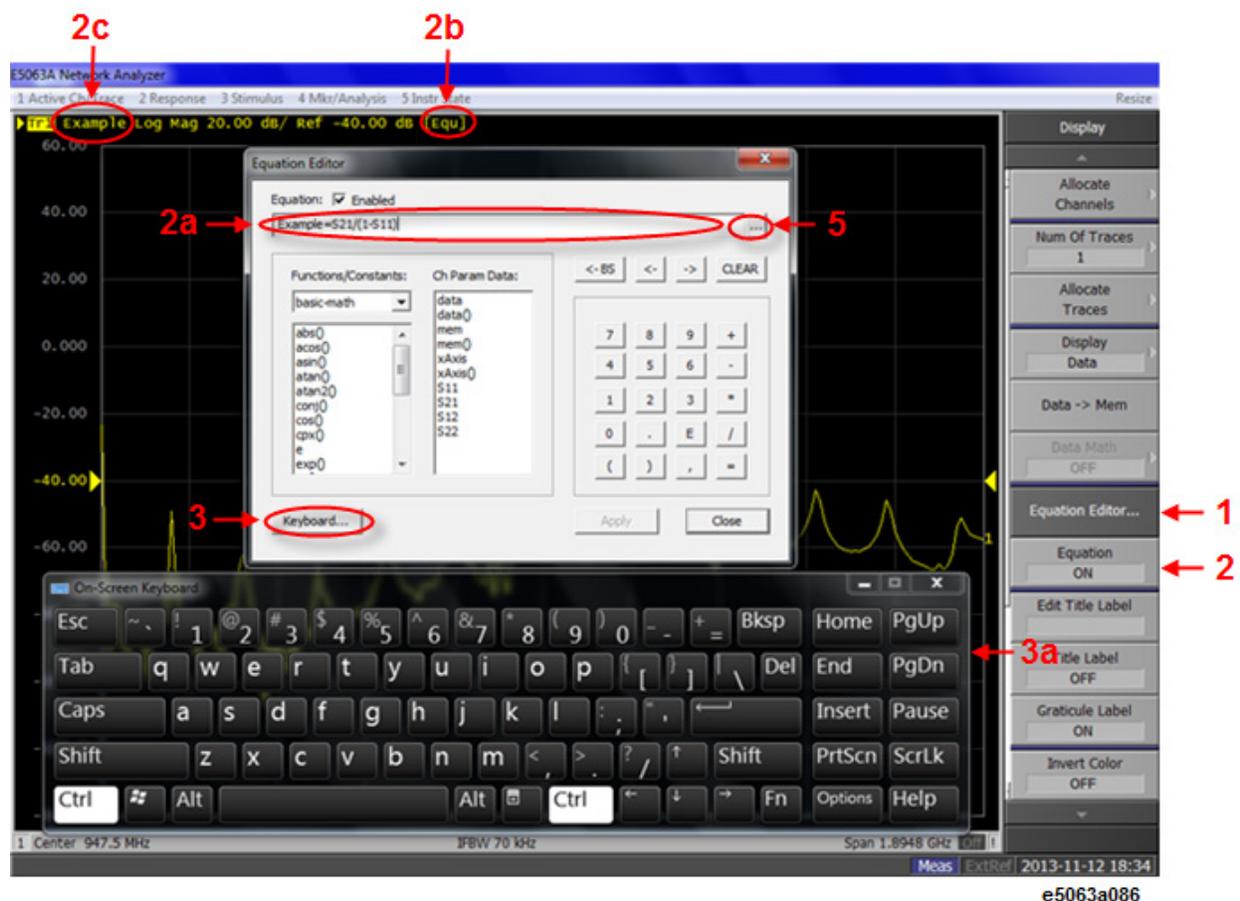
NOTE   The parameter of **Z** is available in **Ch Param Data**.

Equation Editor Dialog box

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.

3. Follow the steps below to enter an equation:

    a. Press **Display**.

    b. Click **Equation Editor** (**1** in the figure above). The Equation Editor dialog box appears.

    c. 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).

4.   Follow the steps below to apply the defined equation. When a valid equation is entered, the Equation Enabled check box becomes available for checking.

  a. Check **Equation Enabled** check box (**2a** in the figure above).

  b. Click **Apply**. The equation becomes visible and annotation of [Equ] (**2b** in the figure above) is displayed in the trace title area.

  c. Click **Close** to hide the dialog box.

  NOTE   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. Input the equation example in the equation field (**4** in Equation Editor dialog box).

Description	Equation Example
Offset each data point in trace 2 from trace 1 by XdB	*Offset= data(1)*pow(10,X/20)*
Multiplying by X-times in log format	*XTimes= pow(data,X)*
Differential Return Loss	*Sdd11= (S11-S21-S12+S22)/2)*
CMRR of a Balanced Component	*CMRR = data(2) / data (1)* *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. 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 to save the defined equation in the directory of the E5063A. To view a list

of saved equations, click the … button (**5** in [Equation Editor Dialog box](#)) to open the Equation History dialog box.
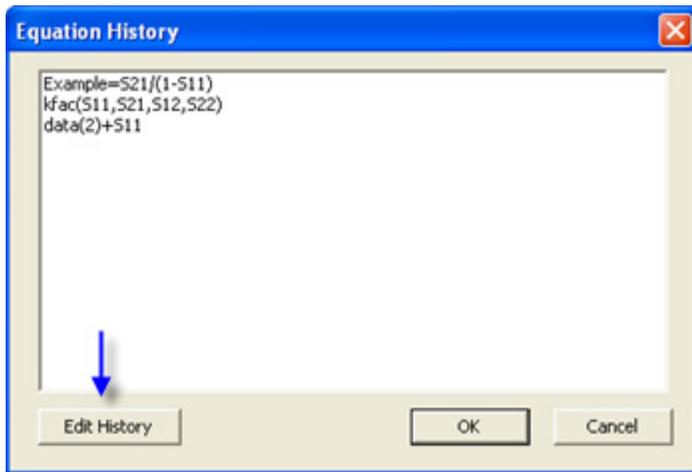
3. To store an equation in the History List, the equation must be applied first. This can be done by clicking on the **Apply** button.

4. To edit the equations in the list, click **Edit History**. The text file of **history list** is opened with Notepad.
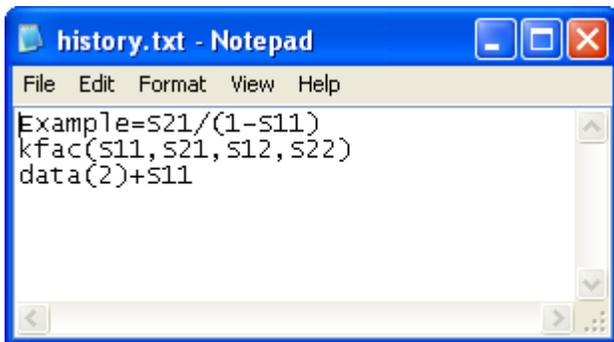
5. The **History** List is stored as a text file **D:\Agilent\Equation\history.txt** and can save a maximum of 50 lines (equations) with a maximum of 254 character per line (equation).

Functions and Constants

The following table describes the different functions and constant available in the E5063A 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^2+a.im^2)
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^2+a.im^2)
phase(complex **a**)	returns atan^2(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 (-п/2, п/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** correspond to Sii,Sji, Sij, Sjj respectively.

- The functions can also be defined by scalar arguments with port numbers of the E5063A. For example, the function, kfac(1,2) returns the K-factor of 2-port measurement between port 1 and port 2.

Z_0 refers ti the system characteristic impedance which can be accessed under Cal.

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 = Z_0 × ((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 = $(1/Z_0)$×((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 = Z_0×((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 = $(1/Z_0)$×((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 = $(1/Z_0) \times ((1-a) \times (1+d) + b \times c)/((1+a) \times (1+d) - b \times c)$
Y11(scalar i, scalar j)	returns Y11(Sii, Sji, Sij, Sjj)
Y12(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{12} conversion = $(1/Z_0) \times (-2 \times c)/((1+a) \times (1+d) - b \times c)$
Y12(scalar i, scalar j)	returns Y12(Sii, Sji, Sij, Sjj)
Y21(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{21} conversion = $(1/Z_0) \times (-2 \times b)/((1+a) \times (1+d) - b \times c)$
Y21(scalar i, scalar j)	returns Y21(Sii, Sji, Sij, Sjj)
Y22(complex **a**, complex **b**, complex **c**, complex **d**)	Y_{22} conversion = $(1/Z_0 0) \times ((1+a) \times (1-d) + b \times c)/((1+a) \times (1+d) - b \times c)$
Y22(scalar i, scalar j)	returns Y22(Sii, Sji, Sij, Sjj)
Z11(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{11} conversion = $Z_0 \times ((1+a) \times (1-d) + b \times c)/((1-a) \times (1-d) - b \times c)$
Z11(scalar i, scalar j)	returns Z11(Sii, Sji, Sij, Sjj)
Z12(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{12} conversion = $Z_0 \times (2 \times c)/((1-a) \times (1-d) - b \times c)$
Z12(scalar i, scalar j)	returns Z12(Sii, Sji, Sij, Sjj)
Z21(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{21} conversion = $Z_0 \times (2 \times b)/((1-a) \times (1-d) - b \times c)$
Z21(scalar i, scalar j)	returns Z21(Sii, Sji, Sij, Sjj)
Z22(complex **a**, complex **b**, complex **c**, complex **d**)	Z_{22} conversion = $Z_0 \times ((1-a) \times (1+d) + b \times c)/((1-a) \times (1-d) - b*c)$
Z22(scalar i, scalar j)	returns Z22(Sii, Sji, Sij, Sjj)

1. 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 - S22	S-parameter data

- When a trace number is out of range, the E5063A indicates an error message, "Equation runtime error" and equation is not reflected on the trace.

Converting S-Parameters to H, Y, Z, F & T-Parameters

The following section provides definition of the two port parameters in Equation Editor.

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:

$D_s = S_{11}S_{22} - S_{12}S_{21}$

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)

- Priority of operators is:

  1. ^
  2. *, /
  3. +, -

Outputting Data

Outputting Data

- [Saving and Recalling Instrument State](#)
- [Saving/Recalling Instrument State for Each Channel into/from Memory](#)
- [Compatibility of State File](#)
- [Saving Trace Data to a File](#)
- [Saving the Screen Image to a File](#)
- [Printing Displayed Screen](#)

Saving and Recalling Instrument State

- Overview
- 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 E5063A 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 E5063A and reproduces the state when it is saved by recalling it later into the E5063A.
State and calibration data (State & Cal)	Saves the setting of the E5063A and calibration data (calibration coefficient array) to reproduce the state when it was saved by recalling it later into the E5063A. 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 E5063A 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 E5063A. At this time, the traces are also recalled and displayed on the screen.
State, calibration data, and traces (All)	Saves the setting of the E5063A, calibration data, and traces to reproduce the state when it was saved by recalling it later into the E5063A. 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.

Saving Data

Selecting Content to be Saved

- 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** > **Save Type**.

241

2. 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.

Saving Instrument State

Follow the procedure below to save internal data from the E5063A.

1. Press **Save/Recall** > **Save State**.

2. 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 E5063A 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 E5063A 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.

- If **D:\Autorec.sta** is found on the system at startup, the E5063A 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 E5063A 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.

- 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 E5063A.

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** > **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 warning messages may appear when recall fails:

- 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 occurs if the user file is not located in the same folder as when the state is saved.

- Pressing **Save/Recall** > **Explorer** executes Windows Explorer. This helps you to browse the files in the storage.

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 save in the **D:\State** folder. This function lets you recall a file you have named freely and save by simple softkey operation, eliminating annoying operation using the Open dialog box.

- 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

"...".

- 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** > **Recall by File Name**.

    2. 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 E5063A, 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

Other topics about Data Output

Overview

The E5063A 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 the 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.

- 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** (or **Channel Prev**) to activate a channel of which the state you want to save.

2. Press **Save/Recall** > **Save Channel**.

3. Click one of **State Only A** to **D**, **State & Cal A** to **D**, **State & Trace A** to **D** or **All A** to **D**, to save the (instrument state and) calibration data of the active channel to the specified register.

4. 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** (or **Channel Prev**) to activate a channel of which the state you want to recall and restore.

2. Press **Save/Recall** > **Recall Channel**.

3. 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** > **Save Channel**.

2. Click **Clear States**. The contents of all the registers are deleted.

Compatibility of State File

- [Overview](#)
- [Compatibility of Files (Saving and Recalling)](#)

Other topics about Data Output

Overview

As mentioned in [Saving and Recalling Instrument State](#), you can save the instrument state of the E5063A into a file on the mass storage and then recall it later to reproduce that state.

Compatibility of Files (Saving and Recalling)

Compatibility from other ENA 6/7 series products

There is no compatibility with other ENA series products.

Compatibility between different options

- The E5063A with option 011 (Time domain/Test wizard) can recall the files saved by the E5063A without option 011.

- The E5063A without option 011 (Time domain/Test wizard) can recall the files saved by the E5063A with option 011. The settings for Transform /Gating are not used but they are kept.

Compatibility between different frequency models

		Recalling by E5063A		
		245	**285**	**2H5**
Saved file by E5063A	**245**	Y	Y	Y
	285	Y#	Y	Y
	2H5	Y#	Y#	Y

Y: Recall is possible, Y# Warning message prompts "Incompatible recall file". Some state data may be lost.

Compatibility when the Firmware revision is different

- Files that are saved with later firmware revisions may not be recalled by prior firmware revisions.

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 default setting.

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 E5063A 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 is saved.

The second line shows the number of the active trace at the time the data is 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 E5063A.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to select the trace to be saved.

2. Press **Save/Recall** > **Save Trace Data** to open the Save As dialog box.

3. Select the destination folder and input a file name.

4. Click **Save** to save the file.

5. This function is inactive when the [security level](#) is set at low/high.

Saving Data in Touchstone Format

You can also save trace data of a E5063A active channel of S-parameter to a Touchstone format file.

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 the 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.

File types of Touchstone files

File types of the E5063A Touchstone files are s1p and s2p. 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 returned value of *IDN?, file created date, calibration state, list of all S parameters of a specified port, and format information.

The header parts of s1p and s2p are shown below.

Header of s1p

```
!Agilent Technologies,E5063A,<ID>,<FW Revision>
!Date <Date>
!Data & Calibration Information
!Freq Syy:Method(Stat)
# Hz S FMT R Z0
```

Header of s2p

```
!Agilent Technologies,E5063A,<ID>,<FW Revision>
!Date <Date>
!Data & Calibration Information
!Freq Syy:Method(Stat) Syz:Method(Stat) Szy:Method(Stat) Szz:Method(Stat)
# Hz S FMT R Z0
```

Each item has the following meaning:

Parameter	Description
Syy to Szz	S parameters of the selected test port; corresponds in ascending order, beginning with y 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
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.

- When the S parameter of a port on which calibration is not performed is specified, if the 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

a : Selected test port number
Freq(n) : Frequency at measurement point n [Hz]
Saa. pri(n) : Real part(RI), linear magnitude(MA) or dB(DB) of measured parameter Saa at measurement point n
Saa. sec(n) : Imaginary part(RI) or phase(MA,DB) of measured parameter Saa at measurement point n
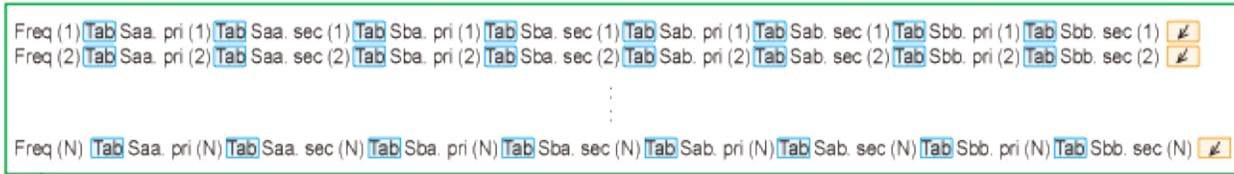N : Number of measurement points
Tab : Tab
⤶ : Line break

e5071c346

2-port Touchstone file

Data

a - b : Selected test port number (corresponding in ascending order, beginning with 1 to a)

Freq(n) : Frequency at measurement point n [Hz]

Sxy. pri(n) : Real part(RI), linear magnitude(MA) or dB(DB) of measured parameter Sxy at measurement point n

Sxy. sec(n) : Imaginary part(RI) or phase(MA,DB) of measured parameter Sxy at measurement point n

N : Number of measurement points

Tab : Tab

↙ : Line break

e5071c347

Restrictions when saving data in Touchstone format

The following restrictions apply in saving measurement data into Touchstone format.

- When both fixture simulation and port impedance conversion are ON, all Z_0 of the ports to be saved must be set to the same value. If Z_0 is different among the ports, no error occurs, but only the Z_0 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.

Saving procedure

Follow the steps below to save trace data in Touchstone format.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to select the trace to be saved.

2. Press **Save/Recall** > **SnP** > **SnP Format**.

3. 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** or **s2p** 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.

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.

9. 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.

10. This function is inactive when the <ins>security level</ins> is set at low/high.

Saving the Screen Image to a File

- [Overview](#)
- [Saving Screen Image as File](#)

Other topics about Data Output

Overview

Along with printing, the E5063A 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.

4. Select the file type from "24-Bit Bitmap (*.bmp)" or "Portable Network Graphics (*.png)".

5. Select the destination folder and type a file name. (Clicking **Input from the front panel** on the Save As dialog box activates the on screen keyboard.)

6. Press **Save** to save the screen image of E5063A 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 E5063A, you can print the displayed screen of the E5063A.

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.

- 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 E5063A.

2. Turn ON the printer and connect it to E5063A.

3. Turn ON the E5063A.

4. Press **System** > **Printer Setup**.

5. 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. 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.

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](#)
- [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 E5063A 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

Data processing flow diagram of fixture simulator

De-embedding

Embedding
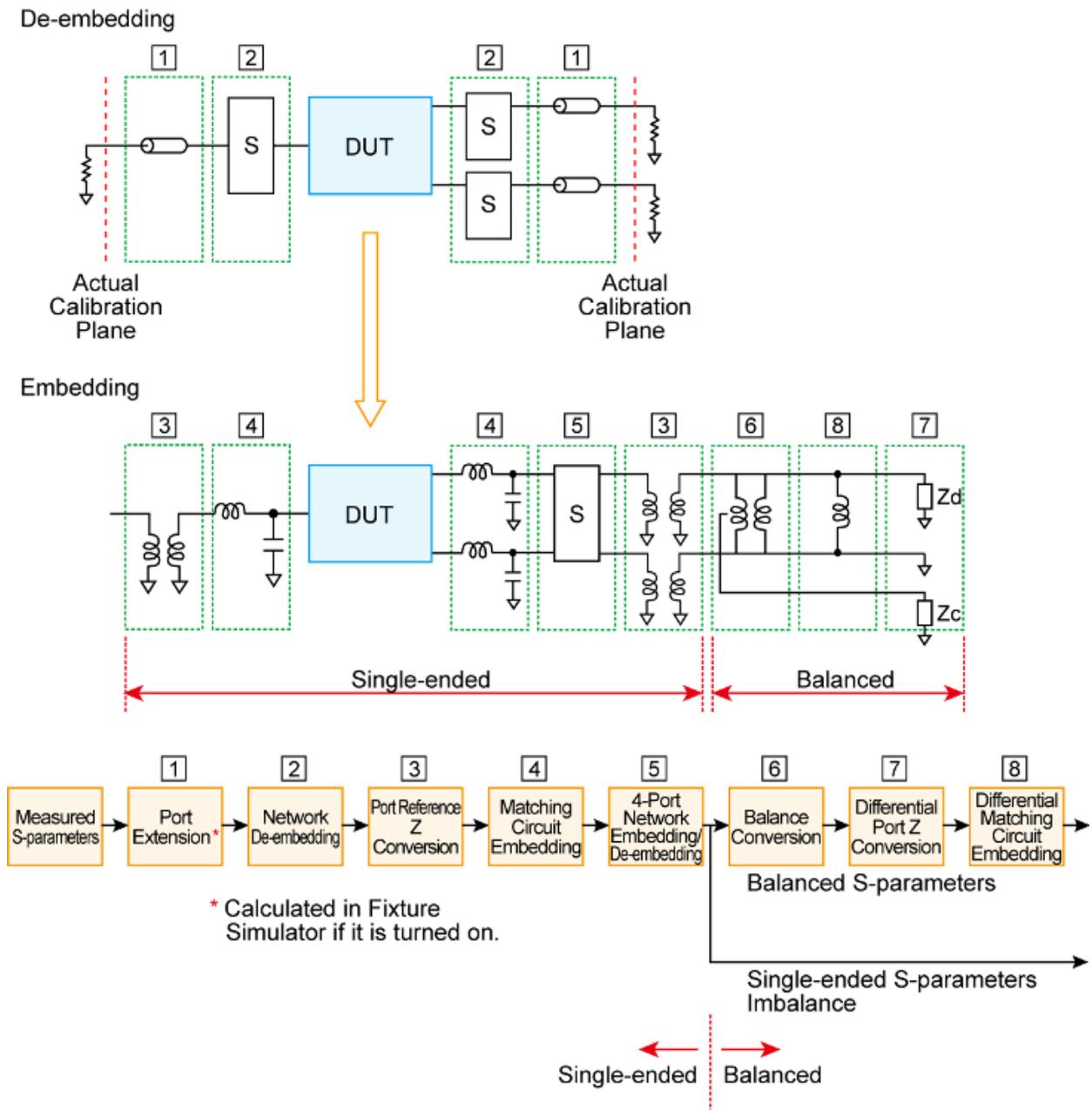
e5072c545.png

> **NOTE**   The E5063A 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 Ω 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 Ω 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.

## 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
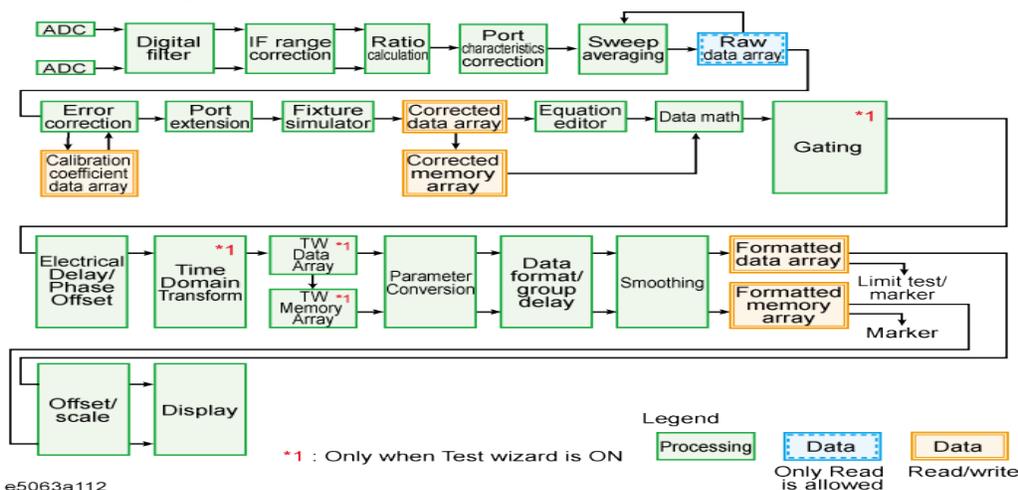
e5063a112

Using Network De-embedding Function

1. Prepare a two-port Touchstone data file (.s2p format) corresponding to the network to be removed.

2. Press **Analysis** > **Fixture Simulator** > **De-Embedding**.

3. Click **Select Port**.

4. Click **1**, or **2** to select the test port to perform network de-embedding.

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** > **ON**.

9. Click **Return**.

10. If **Fixture Simulator** is **OFF**, press the key again to turn it **ON**.

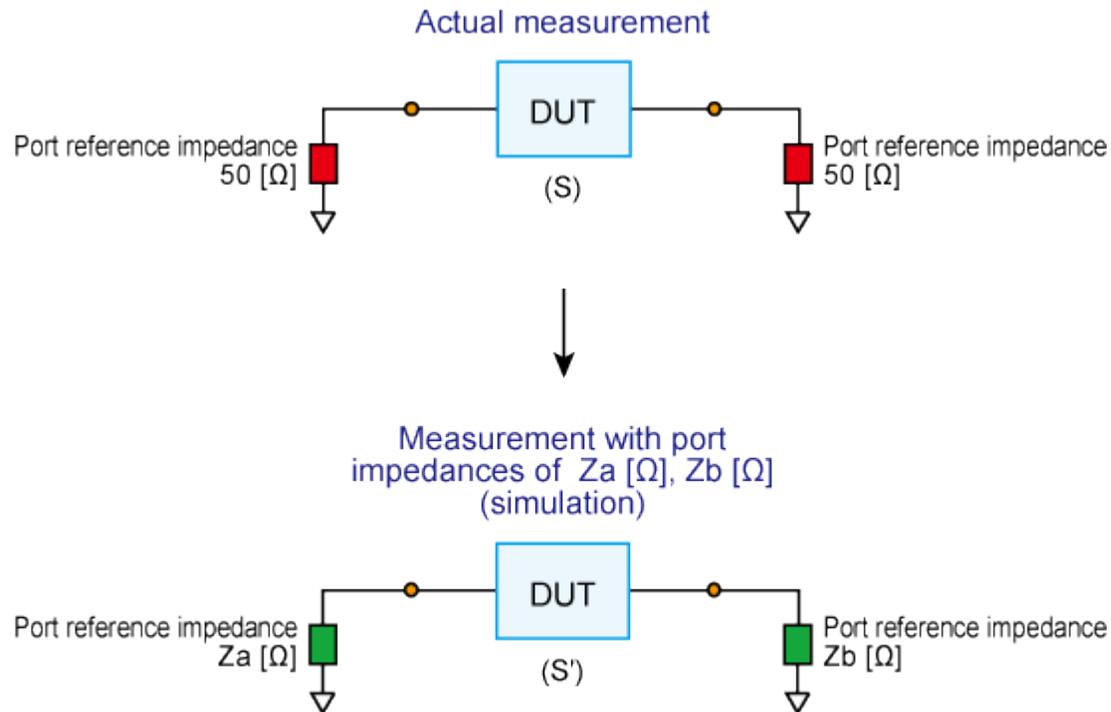## Converting the Port Impedance of the Measurement Result

- Overview
- Converting Port Impedance

Other topics about Fixture Simulator

Overview

The measured value obtained by using a port impedance of 50 Ω can be converted into a measured value at an arbitrary port impedance.

Port impedance conversion function

e5071c377

Converting Port Impedance

1. Press **Analysis** > **Fixture Simulator** > **Port ZConversion**.

2. Specify the port reference impedance for the required port.

   - Select **Port 1 Z0 Real**, or **Port 2 Z0 Real** to set the port reference impedance in Real format.

   - Select **Port 1 Z0 Real** and **Port 1 Z0 Imag**, or **Port 2 Z0 Real** and **Port2 Z0 Imag** to set the port reference impedance in Complex format.

3. Click **Port ZConversion** > **ON.**

4. Click **Return**.

5. If **Fixture Simulator** is **OFF,** click the key again to turn it **ON**.

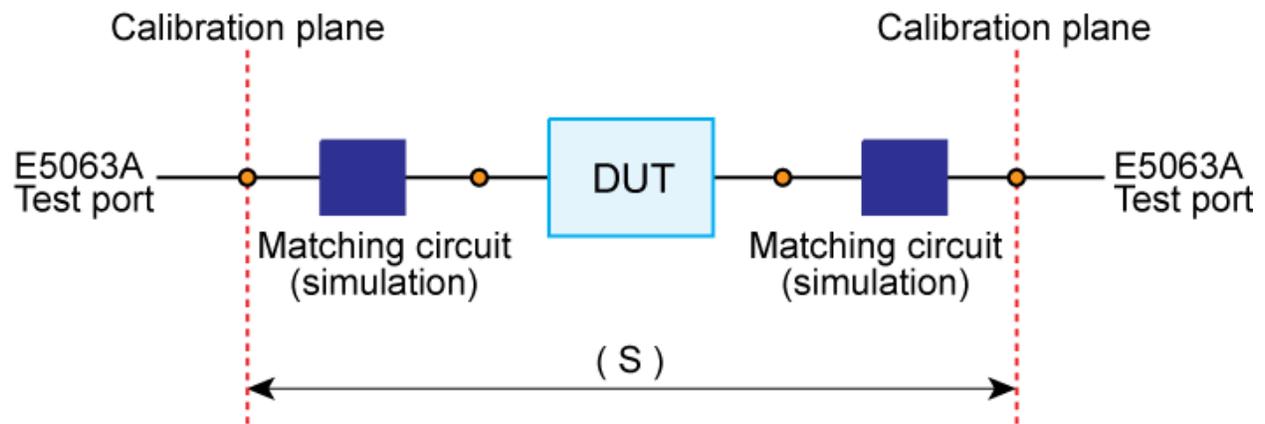## Determining Characteristics After Adding a Matching Circuit

- [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

e5063a077

Define the matching circuit to be added by one of the following methods:

- Select one of the five 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

Test port   L   R   DUT port

C   G

Shunt C - Series L

Test port   L   R   DUT port

C   G

Shunt L - Series C

Test port   C   DUT port

L

G

R

Series C - Shunt L

Test port   C   DUT port

G

L

R

Shunt L - Shunt C

Test port   DUT port

L

C   G

R

User

Test port   Circuit defined   DUT port
1   by a Touchstone   2
            data file

Series C - Shunt C

Test port   C1   DUT port

G1
G2   C2

Shunt C - Series C

Test port   C1   DUT port

G1
G2   C2

Series L - Shunt L

Test port   L1   R1   DUT port

Shunt L - Series L

Test port   L1   R1   DUT port

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**.
2. Click **Fixture Simulator** > **Port Matching** > **Select Port**.
3. Click **1**, or **2** to select the port to add a matching circuit.
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 [Ω]
C2	Specifies capacitance [F]
G2	Specifies conductance [S]
L2	Specifies inductance [H]
R2	Specifies resistance [Ω]

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**.

Evaluating Balanced Devices (Balance-Unbalance Conversion Function)

-
-
-
-
-

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 equal impedances for the two unbalanced ports. For more details on setting up port impedance for unbalanced ports, refer to Converting the Port Impedance of the Measurement Result.

Balance-unbalance conversion
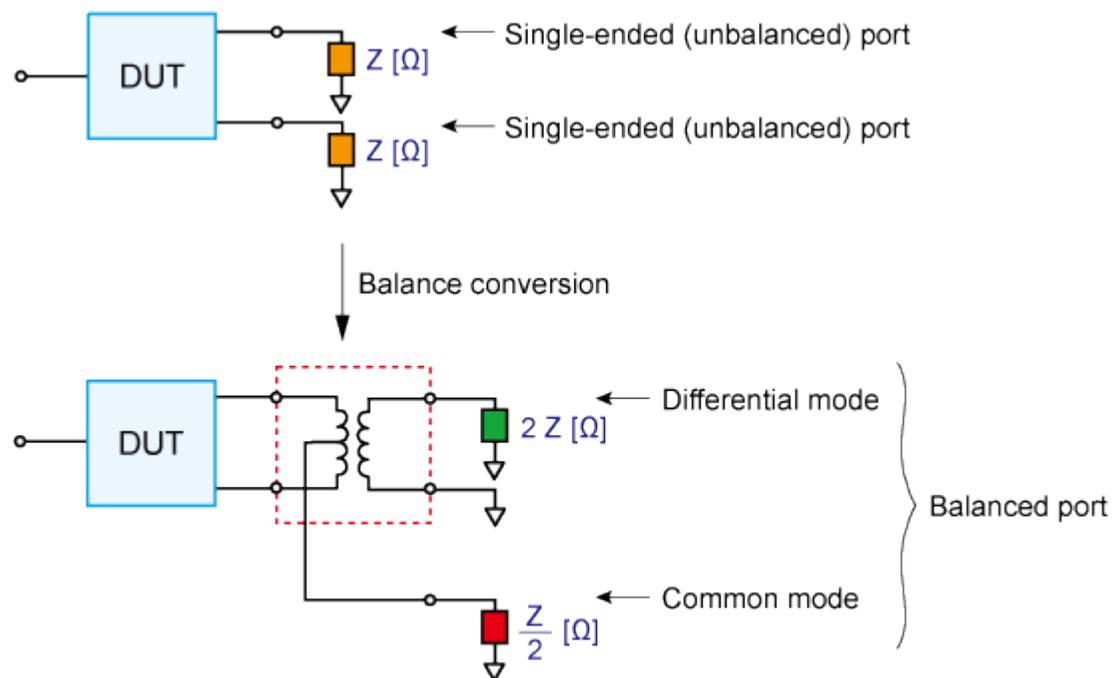
e5071c406

The types of devices that can be evaluated using the E5063A are shown below.

Types of balanced devices that can be evaluated with E5063A

Balanced Devices (Bal)

Balanced port

Logical port 1
[Port1 (bal)]

Port a

Port b

DUT

Number of test ports required for analyzer : 2
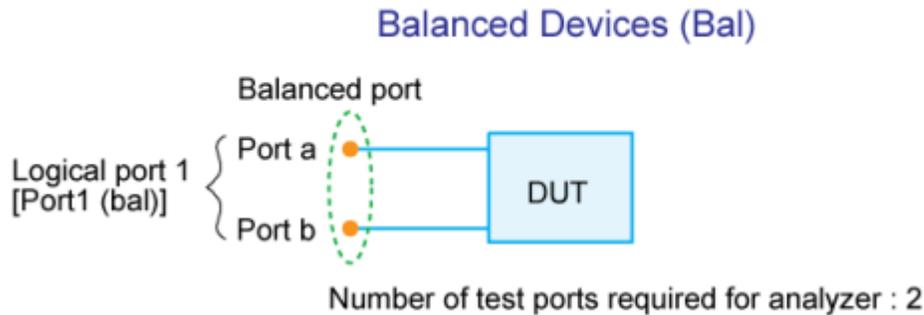
Enclosed with [ ] are softkey labels.

e5072a012.png

In the terminology of the E5063A, ports after the balance conversion are called logical ports (or DUT ports).

Measurement Parameters of Balanced Devices

Turn on the balance-unbalance conversion function to measure the following parameters.

- Mixed mode S-parameter

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

$$S_{xyAB}$$

- Stimulus port number
- Response port number
- Stimulus port mode
- Response port mode
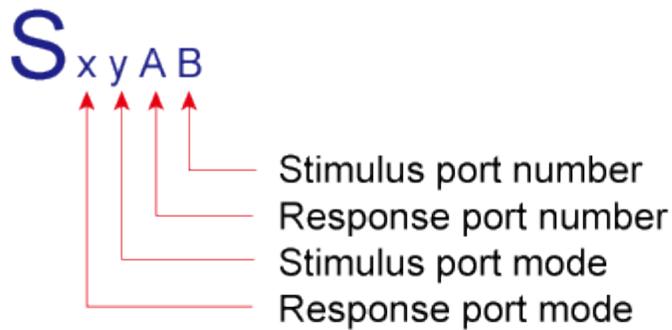
(All ports should be noted as logical ports)

$$\text{Mode} \begin{cases} s: & \text{Single-Ended (Unbalanced)} \\ d: & \text{Differential Mode (Balanced)} \\ c: & \text{Common Mode (Balanced)} \end{cases}$$

$$S_{xyAB} = \frac{x \text{ mode signal output on port A}}{y \text{ mode signal input on port B}}$$

e5071c381

Mixed mode S-parameter when measuring a balanced device
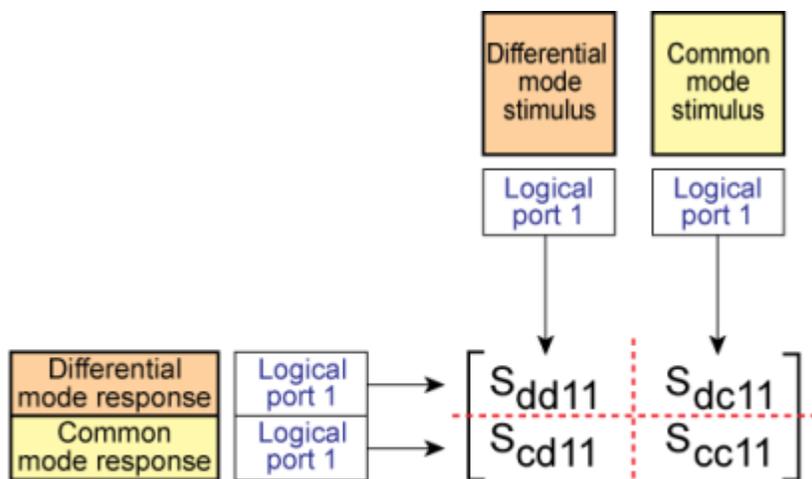
e5071c540

Procedure for Balance-Unbalance Conversion

> **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. Only Bal can be selected.

Softkey	Function
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.

Softkey	Function
Port 1 (bal)	Selects two ports on the analyzer from between **1-2** and **2-1** 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. 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.

e5063a114

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**   When the port reference Z conversion is a complex, you cannot use the differential/common port Z conversion.

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
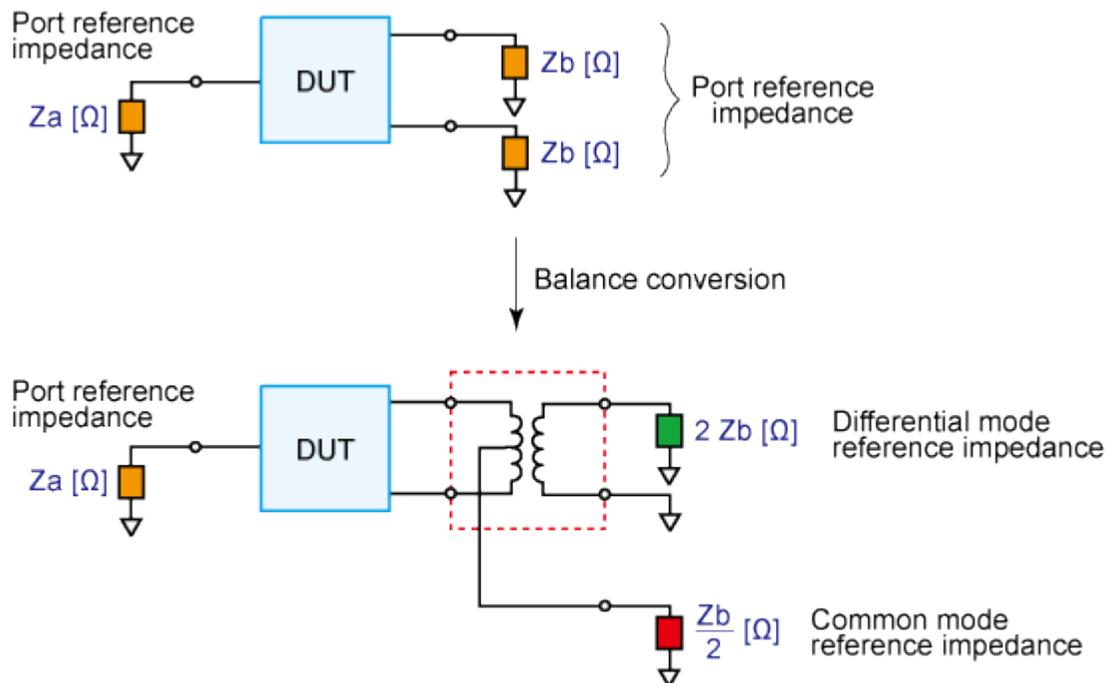
e5071c403

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** > **Fixture Simulator** > **Diff ZConversion**.

2. Click **Diff ZConversion** > **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** > **Fixture Simulator** > **Diff ZConversion**.

2. Click **Port 1 (bal) Real** to specify the port reference impedance in "Real" format for differential mode.

3. Click **Port 1 (bal) Real** and **Port 1 (bal) Imag** to set the differential port reference impedance in "Complex" format.

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** > **Fixture Simulator** > **Cmn ZConversion**.

2. Click **Cmn ZConversion** > **ON**.

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 figure

Procedure to set common port reference impedance

1. Press **Analysis** > **Fixture Simulator** > **Cmn Z Conversion**.

2. Click **Port 1 (bal) Real** to specify the port reference impedance in "Real" format for common mode.

3. Click **Port 1 (bal) Real** and **Port 1 (bal) Imag** to set the common port reference impedance in "Complex" format.

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.

NOTE When the port reference Z conversion is a complex, you cannot use the differential matching circuit embedding.

Balance matching circuit function
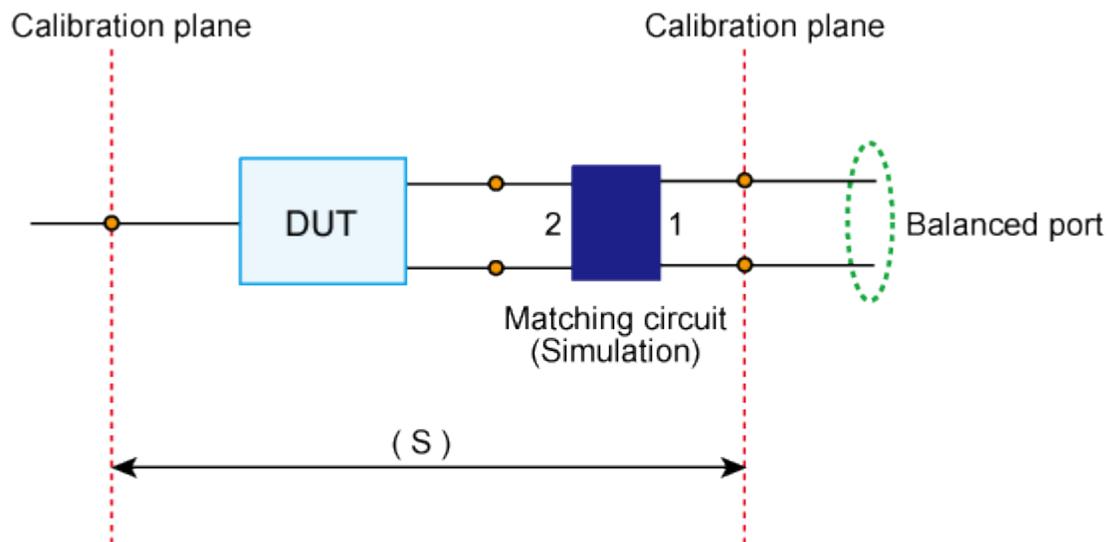
e5071c376

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:

276

- 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
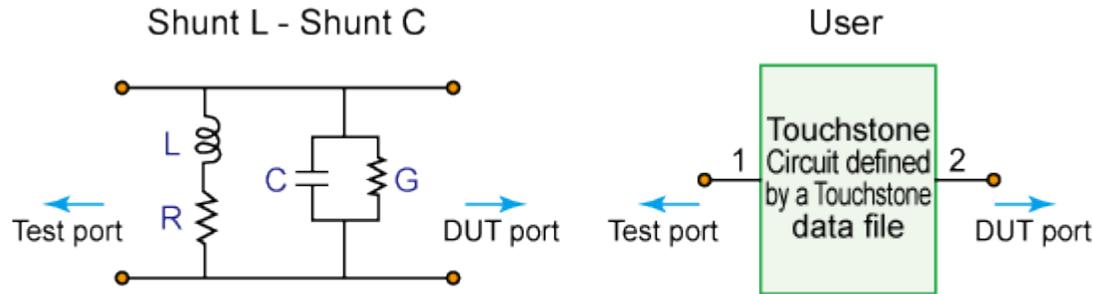
e5071c379

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** > **Fixture Simulator** > **Diff. Matching**.

2. 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.

3. Click **Select Circuit**.

4. 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.

5. 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]

6. **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."

7. Repeat the procedure to set up the differential matching circuit to be added to the selected ports on the DUT.

8. Click **Diff. Matching** to turn the differential matching circuit **ON**.

9. Click **Return**.

10.   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.

> **NOTE**   This example can not be performed by the E5063A which has only two ports. This section provides the general information of fixture simulator.

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

$$R_s = 50\ \Omega$$
$$R_L = 200\ \Omega$$
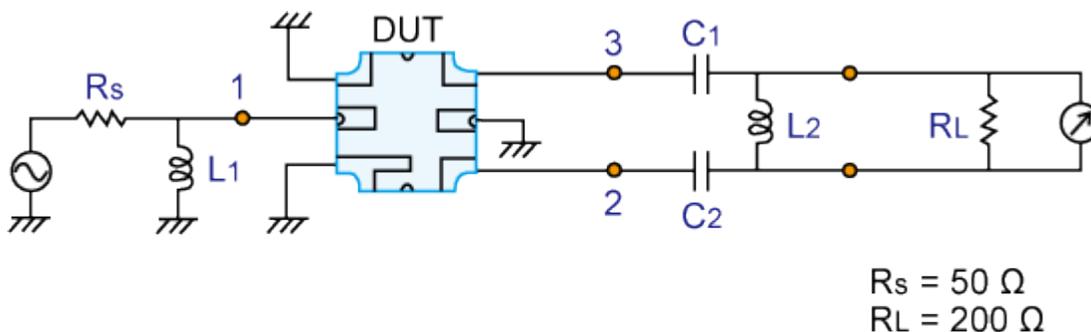
e5071c420

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.

279

In typical use, a network analyzer performs measurements at a 50 Ω 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
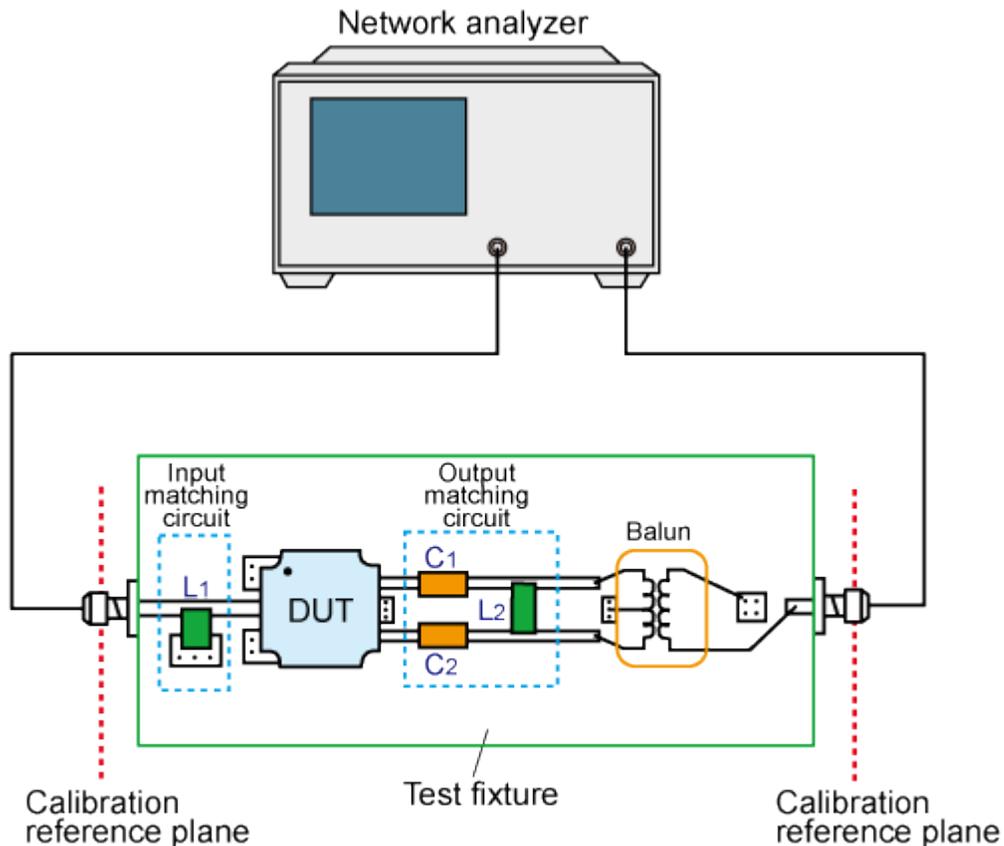
e5071c404

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 errors cannot be avoided. Furthermore, a common mode signal evaluation cannot be performed when an actual balun is used.

DUT Evaluation using Fixture Simulator

The 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 network analyzer's fixture simulator function. The unbalanced port of the DUT should be directly connected to a test port of the network analyzer, and the balanced port of the DUT should be connected to two other test ports of the network analyzer. The actual measurement by the is performed at single-ended ports with a 50 Ω port reference impedance.

DUT connection when fixture simulator is used
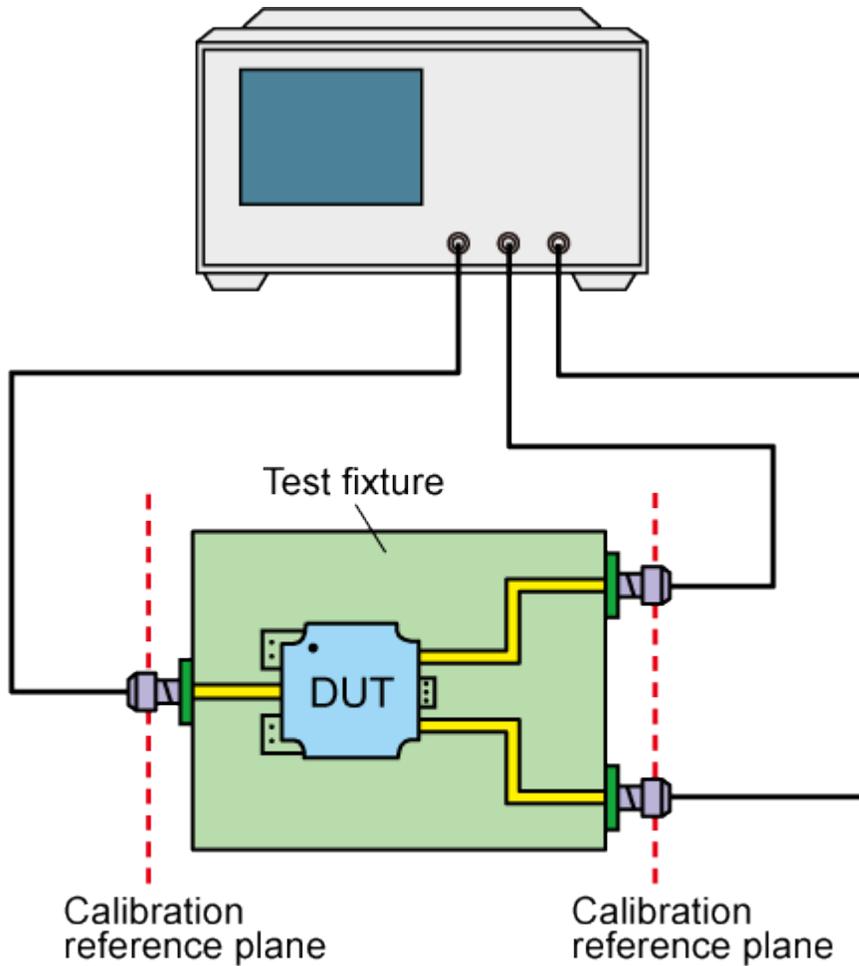
e5071c362

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
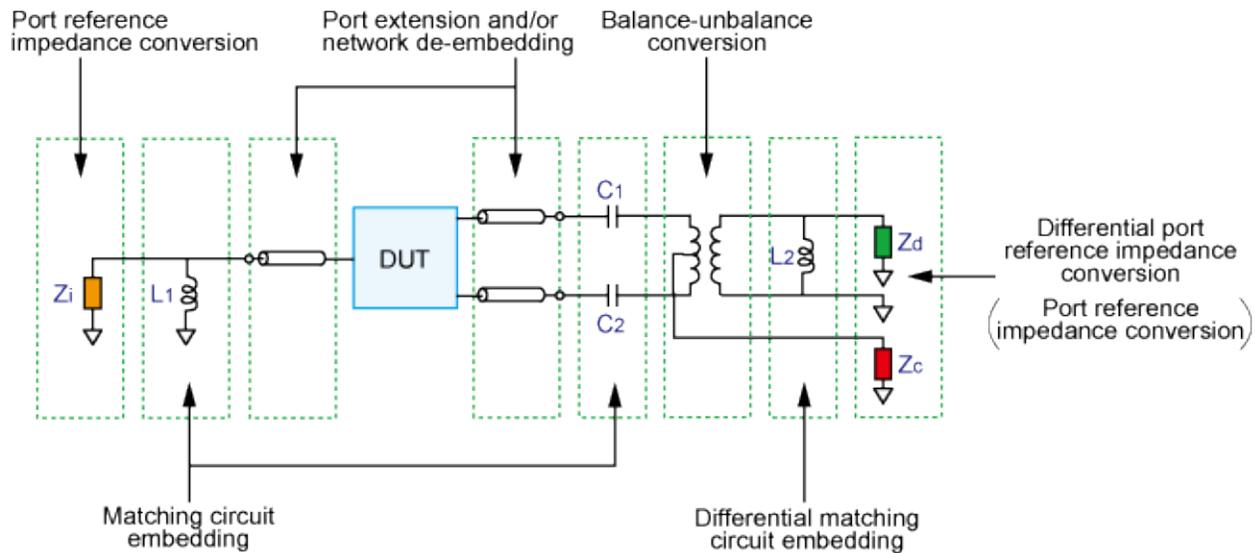
e5071c405

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 network analyzer's test port (50 Ω , 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 [Ω] at the two single-ended ports before balance conversion is automatically converted to 2Z [Ω] for differential mode port and Z/2 [Ω] 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 Ω × 2 = 100 Ω , and common mode port reference impedance Zc becomes 50 Ω / 2 = 25 Ω . Since the differential port is terminated with 200 Ω in the figure,Measurement circuit of balance SAW filter, differential port reference impedance Zd should be set to 200 Ω.

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).

Time Domain Analysis (Opt.011)

Time Domain Analysis (Opt.011)

Test Domain Analysis is available when option 011 is installed.

- [Overview](#)
- [Characteristics of Response in Time Domain](#)
- [Transformation to Time Domain](#)
- [Deleting Unnecessary Data in Time Domain (Gating)](#)

[Other topics about Measurement](#)

Overview of Time Domain Measurement

- Overview
- Comparison to Time Domain Reflectometry (TDR) Measurement
- Time Domain Function of E5063A

Other topics about Time Domain Analysis

Overview

The E5063A provides the time domain function, which is used to mathematically transform waveforms in the frequency domain that can be measured with a general network analyzer to waveforms in the time domain.

The following figure shows waveforms in the frequency domain and in the time domain when using the same cable. The waveform in the frequency domain shows ripples due to mismatches, but it is difficult to estimate their locations. On the other hand, from the waveform in the time domain, you can determine the locations and magnitudes of mismatches.
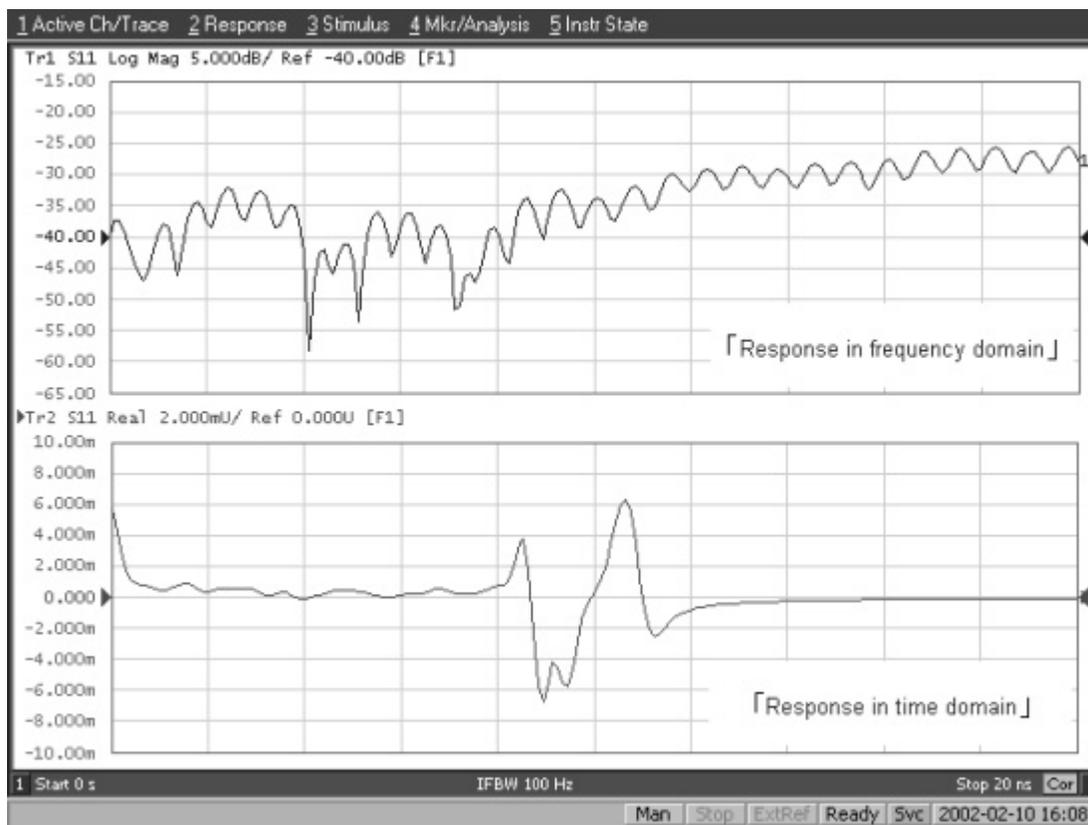
e5070aue202

Comparison to Time Domain Reflectometry (TDR) Measurement

The time domain function of the E5063A is similar to the time domain reflectometry (TDR) measurement in that it displays the response in the time domain.

In the TDR measurement, a pulse or step signal is inputted to the DUT and the change of the reflected wave over time is measured.

On the other hand, the time domain function of the E5063A changes the frequency of the input signal to the DUT, performs measurement in the frequency domain, and transforms the result to the response in the time domain using the inverse Fourier transform.

Time Domain Function of E5063A

The time domain function of the E5063A is divided into the following two types of functions. You can use them at the same time.

Transformation function

Transforms measurement data in the frequency domain to data in the time domain. For more information, refer to Transformation to time domain.

Gating function

Deletes unnecessary data in the time domain from original data in the time domain. For more information, refer to Deleting Unnecessary Data in Time Domain (Gating).

Characteristics of Response in Time Domain

- Overview
- Masking
- Mismatch Type

Other topics about Time Domain Analysis

Overview

This section describes masking and the identification of the mismatch type, which are important for analyzing the response in the time domain.

Masking

Masking is a phenomenon in which a mismatch at a location near the calibration surface affects the response at the next mismatch location. This occurs because energy reflected by a mismatch at the location nearest to the calibration surface does not reach the next mismatch location. For example, when you measure the reflection of a cable that has 2 mismatch locations reflecting 50% of the input voltage, the first mismatch reflects 50% of the measurement signal. The remaining 50% reaches the next mismatch and its 50%, which is 25% of the entire measurement signal, is reflected. Therefore, in the response in the time domain, the second mismatch looks smaller.

- In this example, the transmission line is assumed to have no loss. However, because there actually is loss, the signal is attenuated as the distance from the calibration surface becomes greater.

The following figure shows an example of masking due to loss. It compares masking when a short termination is directly connected to the calibration surface and masking when a 6-dB attenuator is inserted. In either case, total reflection occurs at the short termination. In the latter case, the signal is attenuated in both ways, and the return loss appears to be - 12 dB.

Example of masking

Short termination   6dB attenuator and short termination
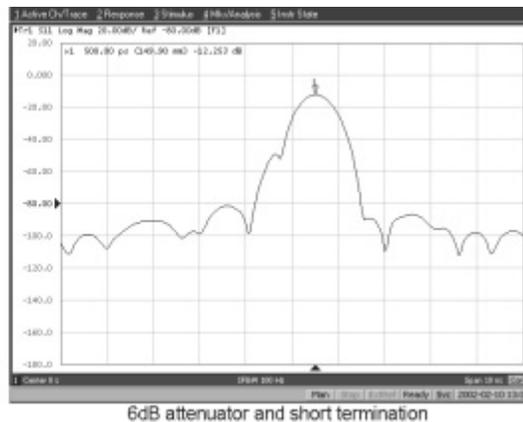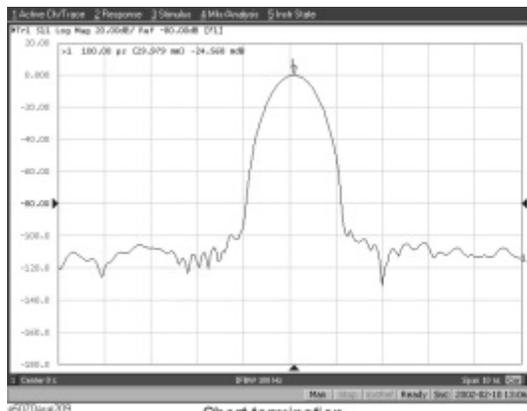
Mismatch Type

The transformation in the low pass mode simulates the response in the TDR measurement. In addition to mismatch locations, the response includes information on the mismatch type.

The following shows each mismatch type and the response waveform corresponding to it. In the low pass mode, you can simulate the response of the step signal and the impulse signal. From the viewpoint of mathematics, the response of the impulse signal is the waveform obtained by differentiating the response of the step signal.

	Step response	Impulse Response
Open	Unity Reflection	Unity Reflection
Short	Unity Reflection, -180°	Unity Reflection, -180°
Resistor $R > Z_0$		
Resistor $R < Z_0$		
Inductor		
Capacitor		

e5071c415

Deleting Unnecessary Data in Time Domain (Gating)

- [Measurement Flow](#)
- [Setting Gate Type](#)
- [Setting Gate Shape](#)
- [Setting Gate Range](#)
- [Setting Unit for X-Axis](#)
- [Enabling Gating Function](#)

Other topics about Time Domain Analysis

Measurement Flow

Item	Description
1. Measurement in frequency domain	Executes measurement in frequency domain
2. Transformation to time domain	Enables transformation function and transforms measurement data to data in time domain
3. Setting the gate	Makes the following settings of the gate to select the necessary domain: • Gate type • Gate shape • Gate range
4. Transformation back to frequency domain	Disables transformation function and displays response in frequency domain corresponding to the data selected with the gate

The following figure shows the change in the waveform at each step of the flow.
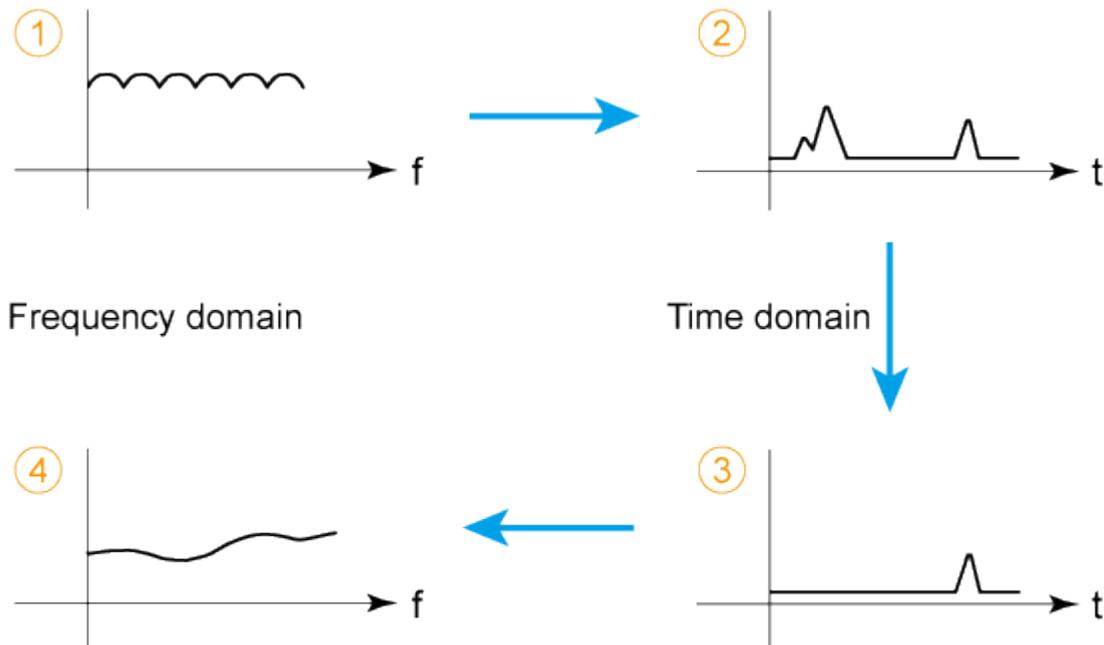
e5071c414

Setting Gate Type

The E5063A allows you to choose from the following two gate types:

Gate type	Description
Band pass	Deletes response outside the gate range
Notch	Deletes response inside the gate range

Operational procedure

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the gate type.

2. Press **Analysis**, then click **Gating** to display the Gating menu.

3. Click **Type** to toggle between band pass (**Bandpass**) and notch (**Notch**).

Setting Gate Shape

The gate is a filter whose shape looks like a band pass filter. There are several parameters that indicate the gate shape. The following figure shows the definition of the gate shape parameters.
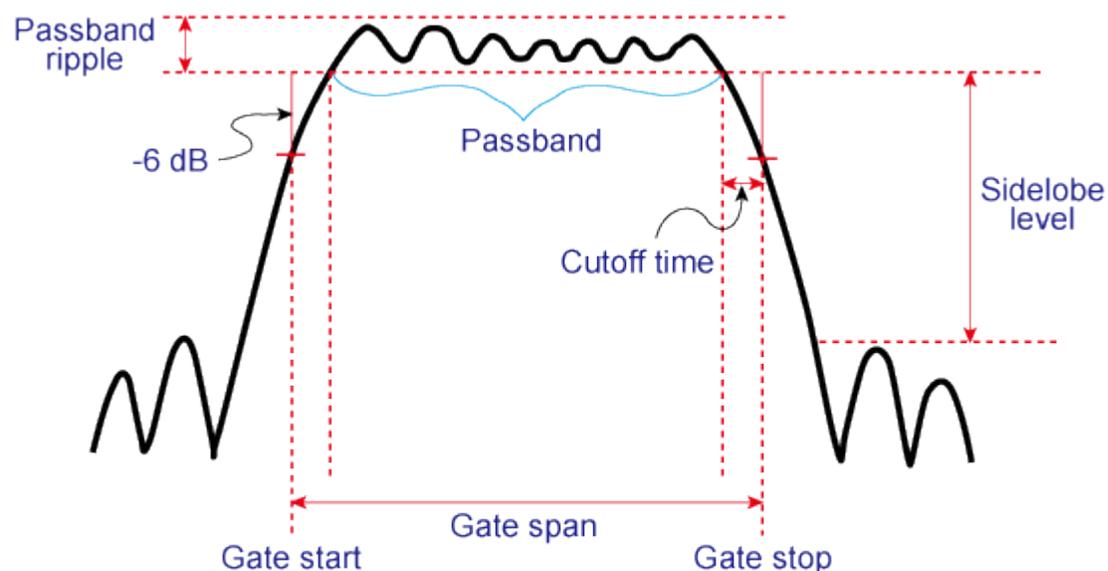
e5071c416

The following table compares the characteristics according to the gate shape. When the shape is "minimum," the cut-off time is shorter and the response is deleted abruptly, but the sidelobe level and band pass ripples become larger. When it is "maximum," cut-off is gentler, but the sidelobe level and the band pass ripple become smaller. The minimum gate span in the following table is the minimum gate range you can set. This value is defined as the minimum gate span necessary for the existence of the pass band and is equal to 2 times the cut-off time.

Gate shape	Passband ripple	Sidelobe level	Cutoff time	Minimum gate span
Minimum	± 0.13 dB	- 48 dB	1.4/frequency span	2.8/frequency span
Normal	± 0.01 dB	- 68 dB	2.8/frequency span	5.6/frequency span
Wide	± 0.01 dB	- 57 dB	4.4/frequency span	8.8/frequency span
Maximum	± 0.01 dB	- 70 dB	12.7/frequency span	25.4/frequency span

Operational procedure

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the gate shape.

2. Press **Analysis**, then click **Gating** to display the Gating menu.

3. Press **Shape** and then select the gate shape.

Setting Gate Range

Specify the gate range in time. The ends of the range are defined as the - 6 dB attenuation points shown in the figure above. You can set the gate range by specifying the start and stop times or the center and span. The E5063A has the following limitations on the gate range you can set.

Lower limit $-T_{span}$

Upper limit T_{span}

T_{span} is the measurement range expressed in time obtained in Measurement range.

Operational procedure

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the gate range.

2. Press **Analysis**, then click **Gating** to display the "Gating" menu.

3. Press each of the following softkeys to specify the gate range. Distance corresponding to the setting time is displayed at the side of the set value area in the data entry bar. The displayed distance is a value that takes the velocity factor into consideration.

Softkey	Function
Start	Sets the start time
Stop	Sets the stop time
Center	Sets the center of the gate in time
Span	Sets the gate span in time

NOTE   No Hardkey is provided for this setting. The hardkeys are dedicated to setting the sweep range.

NOTE   You can set the center and span by dragging and dropping flags indicating the gate range.

Setting Unit for X-Axis

Operational procedure

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the gate range.

2. Press **Analysis**, then click **Gating** to display the "Gating" menu.

3. Click **Units**. The following options are available. These options will represent the X-axis of the graph. Meters, feet and inches are calculated by taking velocity factor into consideration.

Softkey	Function
Seconds	Sets the x-axis to seconds.
Meter	Sets the x-axis to meter (distance).
Feet	Sets the x-axis to feet (distance).
Inches	Sets the x-axis to inches (distance).

Enabling Gating Function

When you enable the gating function, data within the specified range is deleted. When the transformation function is enabled, the flags indicating the gate range is displayed as shown in the following figure.

> **NOTE**   In the figure, the gate type is set to band pass. When it is set to notch, the directions of the flags indicating the ends of the gate range are reversed.
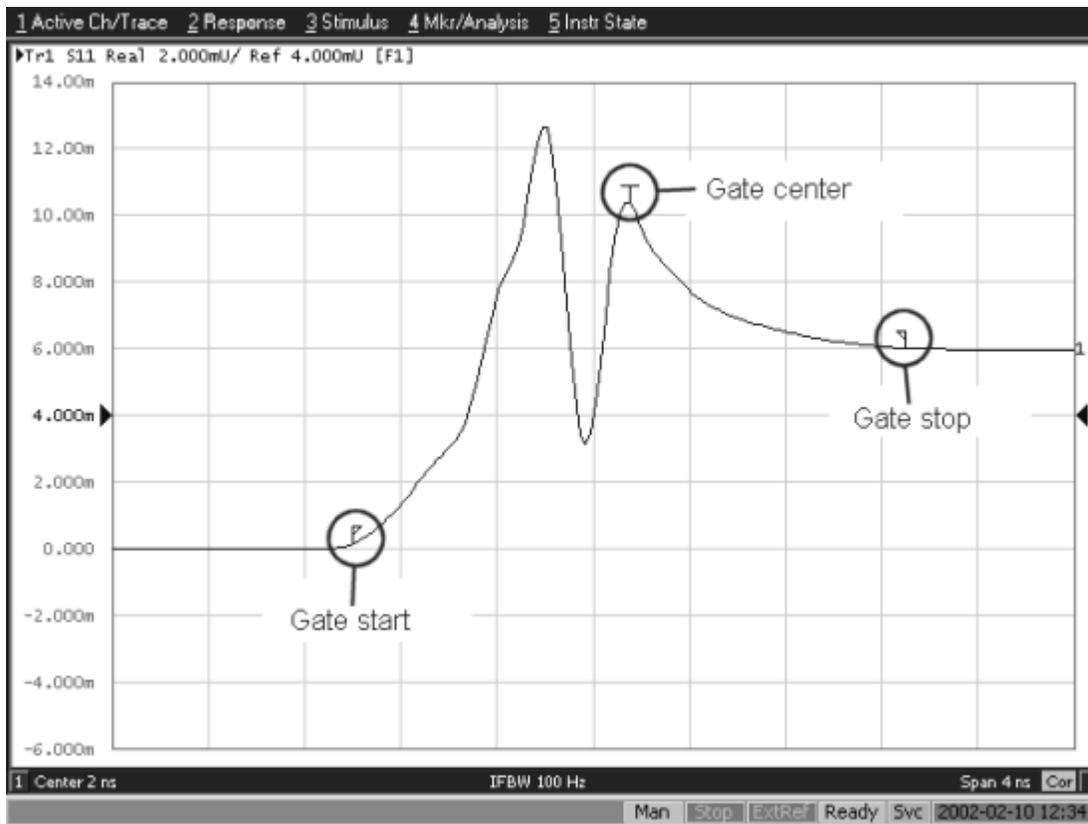
e5070aue205

Operational procedure

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to use the gate function.

2. Press **Analysis** key, then click **Gating** to display the "Gating" menu.

3. Use **Gating** to enable (**ON**) the gate function.

Transformation to time domain

- Overview
- Measurement Flow

Other topics about Time Domain Analysis

Overview

The transformation function lets you transform a response in the frequency domain to the corresponding response in the time domain.

Measurement Flow

Item	Description
Selecting a type	Selects the transformation type from the following: • Band pass mode You can set the sweep range arbitrarily. This is suitable for devices through which signals within a certain frequency range pass such as filters. • Low pass mode Simulates the TDR measurement. This mode is suitable for devices through which dc current or signals of many frequencies pass, such as cables. The low pass mode provides two types of modes: low pass step and low pass impulse.
Setting the window	Sets the window to reduce the ringing phenomenon, in which a waveform's waves are caused due to the fact that the frequency domain is finite.
Calculating necessary measurement conditions	Calculates the following values to obtain the necessary resolution and measurement range in the time domain: • Sweep range • Number of points • Window width
Setting the frequency	Sets the sweep range and the number of points to

range and the number of points	the values calculated above.
Setting display range	Sets the range displayed on the graph.
Enabling transformation function	Enables the transformation function.

Selecting a type

There are two types of transformation to the time domain: band pass and low pass. The appropriate transformation type depends on the DUT.

Comparison between band pass mode and low pass mode

Item	Band pass	Low pass
Appropriate DUT	DUTs that do not operate with dc current such as band pass filters.	DUTs that operate with dc current such as cables.
Input signal	You can simulate the response to the impulse signal.	You can simulate the TDR measurement. You can simulate the response to both the impulse signal and step signal.
Selection of the sweep range	You can select the sweep range arbitrarily.	Because dc data are estimated from the first few points, the frequencies of measurement points must be multiples of the start frequency.
Transmission/ reflection measurement	You can perform both transmission and reflection measurements.	You can perform both transmission and reflection measurements.
Identification of mismatches	You can identify the locations of mismatches.	You can identify the locations of mismatches and the type of impedance (capacitive or inductive).
Resolution		The resolution in the time domain increases by two times over that in the band pass mode.

Available data format	• Liner magnitude format	Real format
	• In the reflection measurement, it indicates the mean of the reflection coefficient within the frequency sweep range.	In the low pass mode, the real format is useful because the time axis data do not have phase information.
	In the transmission measurement, it indicates the mean of the transmission coefficient within the frequency sweep range.	
	• Log magnitude format	
	• In the reflection measurement, it indicates the mean of the return loss within the frequency sweep range.	
	In the transmission measurement, it indicates the mean of the transmission gain within the frequency sweep range.	
	• SWR format	
	• In the reflection measurement, it indicates the mean of SWR (standing wave ratio) within the frequency sweep range.	

Impulse signal and step signal

The E5063A lets you simulate the response from the DUT to two types of signals: impulse signal and step signal. The impulse signal is a pulse-shape signal in which the voltage rises from 0 to a certain value and returns to 0 again. The pulse width depends on the frequency sweep range. The step signal is a signal in which the voltage rises from 0 to a certain value. The rise time depends on the maximum frequency within the frequency sweep range.

- For more information on how the frequency span setting affects the pulse width and the rise time, refer to [Calculating necessary measurement conditions](#).

Step signal and impulse signal

e5071c417

Operation

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate a trace for which you want to set the transformation type.

2. Press **Analysis** > **Transform** to display the Transform menu.

3. Click **Type**, then select one of the **softkeys** to specify the type.

4. Press **Format** key to display the "Format" menu, then select the data format.

Setting the window

Because the transforms data within a finite frequency domain to data in the time domain, an unnatural change of data at the end points within the frequency domain occurs. For this reason, the following phenomena occur.

- Width of impulse signal and rise time of step signal
  A time width occurs in the impulse signal and a rise time occurs in the step signal.

- Sidelobe
  Sidelobes (small peaks around the maximum peak) occur in the impulse signal and the step signal. Ringing occurs on the trace due to sidelobes, which reduces the dynamic range.

By using the window function, you can lower the level of sidelobes. However, the width of the impulse and the rise time of the step become larger as a penalty. You can select from three types of windows: maximum, normal, and minimum. The following table shows the approximate relation between the window and the sidelobe/impulse width.

Window	Sidelobe level of impulse signal	Width of impulse (**50% in low pass mode**)	Sidelobe level of step signal	Rise time of step signal (10 - 90%)
Minimum	- 13 dB	0.60/frequency span	- 21 dB	0.45/frequency span
Normal	- 44 dB	0.98/frequency span	- 60 dB	0.99/frequency span
Maximum	- 75 dB	1.39/frequency span	- 70 dB	1.466/frequency span

The window function is available only when the response in the time domain is displayed. It does not have any effect when the response in frequency domain is displayed. The following figure shows the effect of the window when measuring the reflection of a short circuit in the time domain.
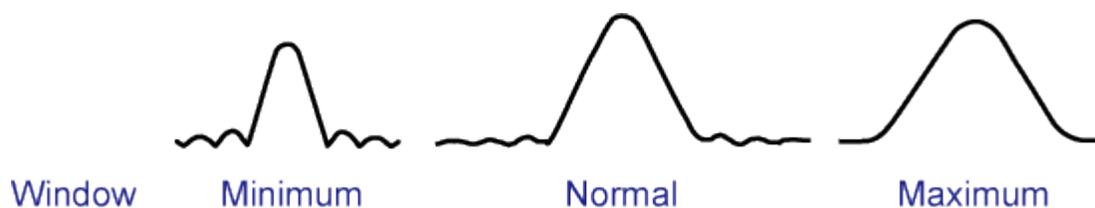
e5071c419

Operation

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate a trace for which you want to set the window.
2. Press **Analysis**, then click **Transform** to display the Transform menu.
3. Press **Window** and then select a window type.

Softkey	Function
Maximum	Sets the window type to maximum. b of the Kaiser Bessel function is set to 13.
Normal	Sets the window type to normal. b of the Kaiser Bessel function is set to 6.
Minimum	Sets the window type to minimum. b of the Kaiser Bessel function is set to 0.

Impulse Width	Sets the window by specifying the impulse width or the step rise time. The lower limit can be set to the value when the window is set to minimum, the upper limit when the window is set to maximum.
Kaiser Beta	Sets the window by specifying the b value of the Kaiser Bessel function. The Kaiser Bessel function determines the shape of the window. The allowable setting range is 0 to 13.

- By specifying **Kaiser Beta**, or **Impulse Width**, you can specify a window that is not classified into the three window types. Conversely, when you specify a window type, these values are set automatically.

Calculating necessary measurement conditions

To use the transformation function efficiently, you need to make the following three settings appropriately.

- Resolution of the response

- Measurement range

- Sidelobe

This section describes the details of these settings.

Effect of frequency sweep range on response resolution

The following figure shows an example when measuring the same cable while changing the sweep span. When measured in a narrower sweep range, the overlap between two peaks is larger than when measured in a wider sweep range. By performing measurement in a wider sweep range, adjacent peaks can be clearly separated, which means that the response resolution is smaller.

Effect of frequency sweep range on resolution
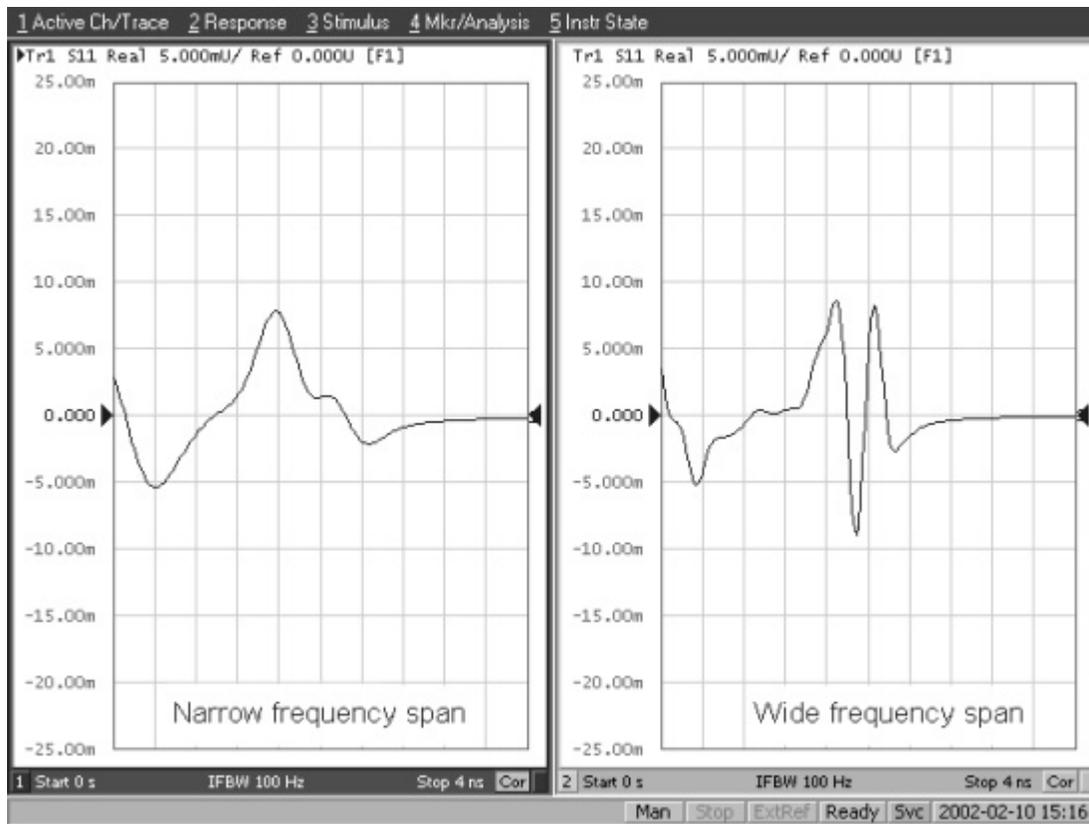
e5070aue208

The sweep range affects the width of the impulse signal and the rise time of the step signal. The width of the impulse signal and the rise time of the step signal are inversely proportional to the sweep range. Therefore, the wider the sweep range is, the shorter these times are.

The resolution is equal to the width defined at the point of 50% of the impulse signal or the rise time defined at the points of 10% and 90% of the step signal.

Definitions of impulse width and step rise time

e5071c418

302
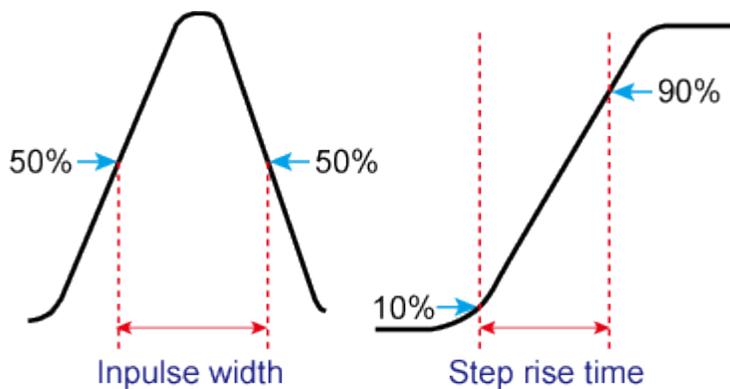
Effect of the window function on the response resolution

Lowering the sidelobe level with the window function elongates the width of the impulse signal and the rise time of the step signal. As described in Effect of frequency sweep range on response resolution, because the response resolution is equal to the width of the impulse signal and the rise time of the step signal, lowering the sidelobe level enlarges the response resolution. The following table shows the approximate relationship between the response resolution and the window setting.

Correspondence of Window setting and response resolution

Window	Low pass step	Low pass impulse	Band pass
Minimum	0.45/stop frequency	0.60/stop frequency	1.20/frequency span
Normal	0.99/stop frequency	0.98/stop frequency	1.95/frequency span
Maximum	1.46/stop frequency	1.39/stop frequency	2.77/frequency span

The following figure shows how the response changes as the window shape changes. If the magnitudes of adjacent peaks are comparable, you need to make the resolution higher; if they differ significantly, you need to set the window so that smaller peaks with lower sidelobes appear.

Effect of window on response resolution
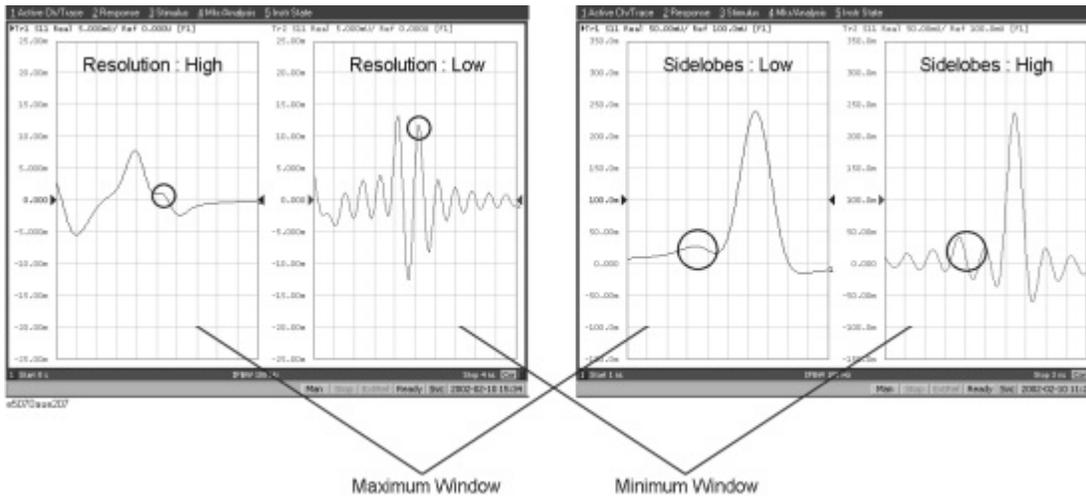
Effect of the transformation type on the response resolution

Although both transformation types, band pass and low pass impulse, simulate the response of the impulse signal, the impulse width in the low pass impulse mode is half the width of that in the band pass mode as shown in the table above. Therefore, the resolution is better in the low pass mode.

303

If the DUT can be measured in the low pass mode, response data with better resolution is obtained in the low pass mode.

Measurement range

In the time domain function, the measurement range means the range within which the response can be measured without repetition. The repetition of the response occurs because measurement in the frequency domain is performed discretely instead of continuously. The measurement range is inversely proportional to the frequency difference between adjacent measurement points. The frequency difference between measurement points $Delta\ F$ is expressed as follows by using the span of the sweep frequency F_{span} and the number of points N_{meas}.

$$\Delta F = \frac{F_{span}}{N_{meas} - 1}$$

Therefore, the measurement range is proportional to the number of points-1 and inversely proportional to the span of the sweep range. To enlarge the measurement range, use one of the following methods:

- Increase the number of points.

- Narrow the span of the sweep range.

- When you change the above settings after performing calibration, you need to perform calibration again.

The sweep range is expressed as time or distance. The time of the measurement range T_{span} is as follows:

$$T_{span} = \frac{1}{\Delta F}$$

The distance of the measurement range Lspan is expressed as follows using the velocity factor V and the speed of light in a vacuum c ($3*E^8$ m/s).

$$L_{span} = \frac{Vc}{\Delta F}$$

- The maximum length of the DUT that can be measured in the transmission measurement is L_{span}. On the other hand, in the reflection measurement, because the signal that applies returns, it is 1/2 of L_{span}.

The velocity factor varies depending on the material through which the signal propagates: 0.66 for polyethylene and 0.7 for PTFE.

Changes in the settings and changes in the response

The following table shows the effect of changes in the measurement conditions on the response resolution and measurement range.

Change in setting	Response resolution	Measurement range	Sidelobe
Widening the sweep range	Becomes smaller	Becomes narrower	Does not change
Setting the window type to maximum	Becomes larger	Does not change	Becomes lower
Increasing the number of points	Does not change	Becomes wider	Does not change

Setting the frequency range and the number of points

Operation

1. Press **Channel Next**/**Channel Prev** to activate the channel you want to set. The frequency range and the number of points are common to all of the traces in the channel. If you want to use different settings, make them on another channel.

2. Press **Sweep Setup**, then click **Sweep Type** > **Lin Freq** to set the sweep type to linear sweep. When the sweep type is set to other than the linear sweep, the time domain function is not available.

3. Set the sweep range using the **Start**/**Stop** or **Center**/**Span**.

4. Press **Sweep Setup**, then click **Points** and enter the number of points. When performing measurement in the low pass mode, select **Analysis** > **Transform** > **Set Freq Low Pass** to adjust the frequency range so that it is appropriate for the low pass mode. The frequency changes depending on the stop frequency as shown below.

Condition of stop frequency	Frequency setting
>= F_{low} * the number of points	Start frequency = stop frequency/number of points
< F_{low} * the number of points	Start frequency: F_{low} Stop frequency = F_{low} * Number of points

5. F_{low} =9 kHz or 100 KHz (Depending upon the installed option)

When the frequency settings satisfy the conditions shown above, the **Set Freq Low Pass** key is displayed in gray.

Setting display range

The E5063A has the following limitations on the display range you can set.

- Lower limit: $-T_{span}$
- Upper limit: T_{span}

T_{span} is the measurement range expressed in time obtained in [Measurement range](#).

The number of response points displayed on the graph is the same as the number of points regardless of the response resolution.

Operation

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the display range.
2. Press **Analysis**, then click **Transform** to display the "Transform" menu.
3. Press each of the following softkeys and then specify the display range. Distance corresponding to the setting time is displayed at the side of the set value in the data entry bar. The displayed distance is a value taking the velocity factor into consideration.

Softkey	Function
Start	Sets the start value of the display range in time
Stop	Sets the stop value of the display range in time
Center	Sets the center value of the display range in time
Span	Sets the span of the display range in time

- You cannot use hardkeys to set the display. The hardkeys are dedicated to specifying the sweep range.

Setting Unit for X-Axis

Operational procedure

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the gate range.
2. Press **Analysis**, then click **Transform** to display the "Transform" menu.
3. Click **Units**. The following options are available. These options will represent the X-axis of the graph. Meters, feet and inches are calculated by taking velocity factor into consideration.

Softkey	Function
Seconds	Sets the x-axis to seconds.
Meters	Sets the x-axis to meter (distance).

Feet	Sets the x-axis to feet (distance).
Inches	Sets the x-axis to inches (distance).

Enabling transformation function
Operation

1. Press **Channel Next**/**Channel Prev** and **Trace Next**/**Trace Prev** to activate the trace for which you want to use the transformation type.

2. Press **Analysis**, then click **Transform** to display the "Transform" menu.

3. Click **Transform** to turn ON the transformation function.

4. The following requirements must be met to enable the transformation function.

    - The sweep mode is the linear sweep.
    - There are three or more measurement points.

Optimizing Measurements

Optimizing Measurements

- [Expanding Dynamic Range](#)
- [Reducing Trace Noise](#)
- [Improving Phase Measurement Accuracy](#)
- [Improving Measurement Throughput](#)
- [Performing Segment-by-Segment Sweep (segment sweep)](#)

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.
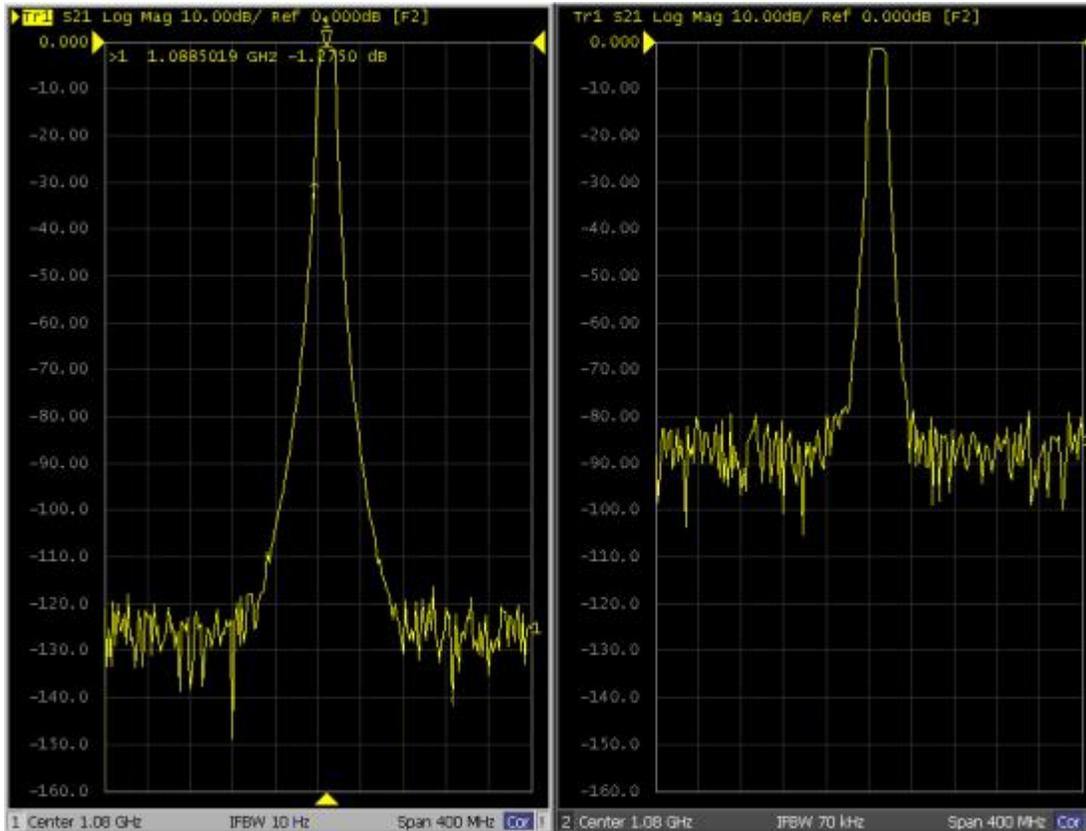
e5071c080

To specify the IF bandwidth, follow the steps described below.

1. Press **Channel Next** (or **Channel Prev**) 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.

Setting IF Bandwidth Automatically

The IF bandwidth auto mode helps you to improve the measurement speed, when the sweep mode is set at log especially. When you make a measurement with wide span from lower frequency and the IF bandwidth is set at fixed frequency, the measurement would be slow or higher noise floor in lower frequency. This function allows you to measure narrower IF bandwidth (lower noise) in lower frequency and wider IF bandwidth (faster) in higher frequency.

1. Press **Avg** key.

2. Click **IFBW Auto** to turn ON the IF bandwidth auto mode.

3. If you want to set an upper limit of IF bandwidth in auto mode, press **IFBW Auto Limit**, and enter the upper limit with enter keys.

4. In IF bandwidth auto mode, the IF bandwidth is automatically set to equal to or less than 1/5 of each measurement frequency. (The E5063A does not change IFBW for each test points in IF BW auto mode. The E5063A divides its measurement span into several segments which are defined internally. The IFBW is set at equal to or less than 1/5 of the start frequency of each segment.)

5. IFBW Auto On may make the sweep time be longer than IFBW Auto Off. This is because IFBW auto operation requires some additional time.

Turning ON Sweep Averaging

Using sweep averaging also enables you to reduce the effects of random noise on the 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 the 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)
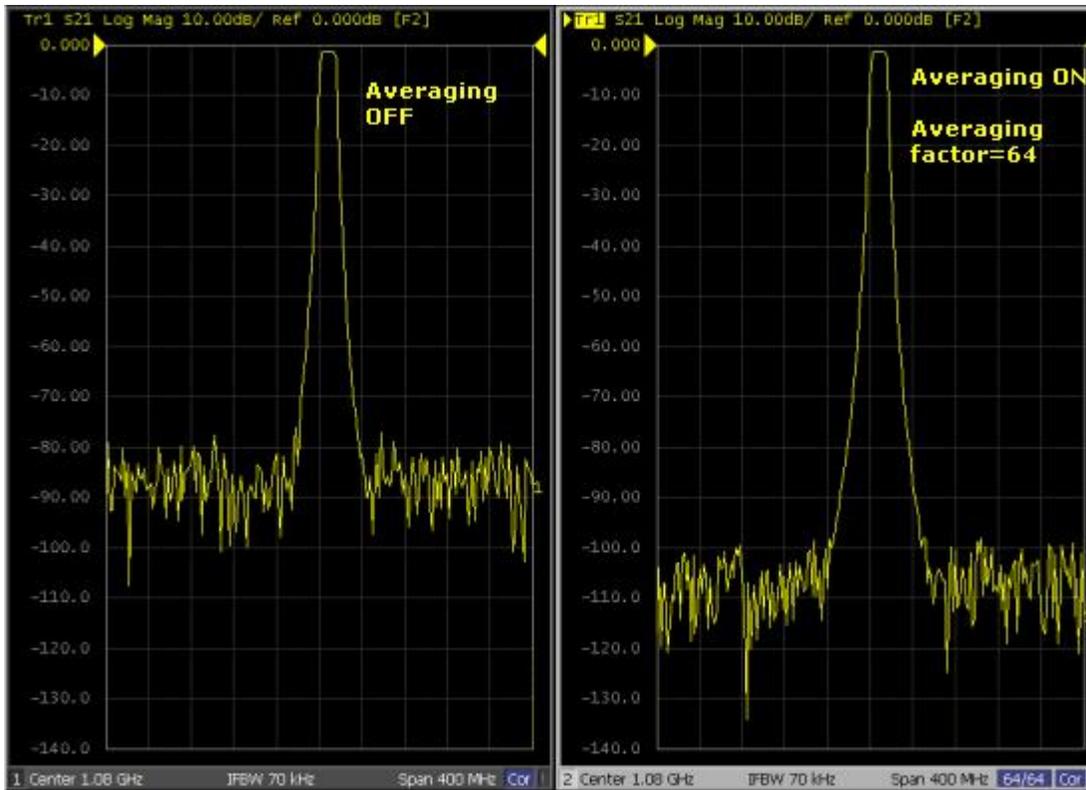
e5071c081

Define the sweep averaging by following the steps below.

1. Press **Channel Next** (or **Channel Prev**) to select the channel on which you want to define the sweep averaging.

2. Press **Avg** > **Avg Factor**, then change the averaging factor in the data entry area.

3. Click **Averaging** to turn ON the averaging.

4. 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 Turning 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.

- You can define the smoothing trace by trace.
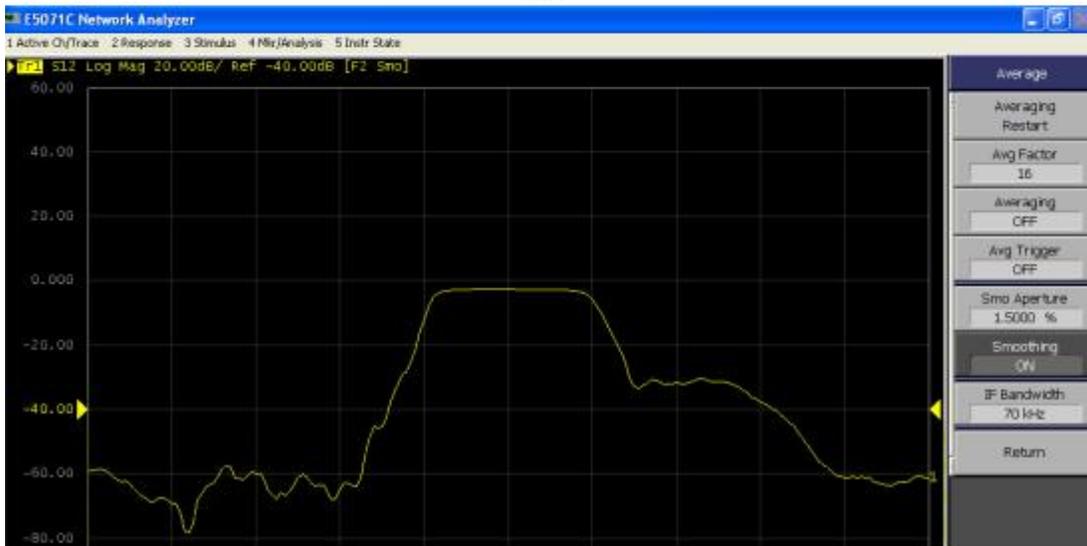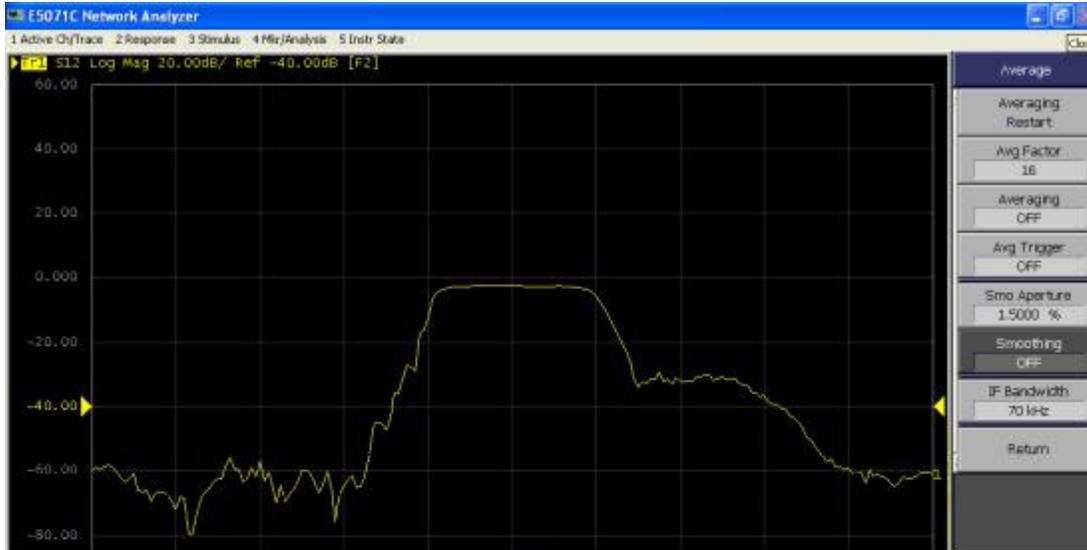
e5071c075

Setting up smoothing

Set up the smoothing operation by following the steps below:

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to activate the trace on which smoothing will be defined.

2. Press **Avg**.

3. Click **Smo Aperture**, then change the smoothing aperture (%) in the data entry area.

4. Click **Smoothing** to turn ON smoothing. **Smo** is displayed at the trace status area.

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 the deviation from the linear phase. You can specify the electrical delay trace by trace.

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to activate the phase trace of which you want to specify the electrical delay.

2. Press **Scale** > **Electrical Delay**.

3. Change the electrical delay (in seconds) in the data entry area.

4. 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.

5. If you have selected **Waveguide** as the media type, click **Cutoff Frequency**, and specify a cutoff frequency.

6. The cutoff frequency is available only when the media type is **Waveguide**.

Procedure using marker

1. Press **Channel Next** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to activate the trace of which you want to set the electrical delay.

2. Place the active marker in an appropriate position.

315

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 which occurs 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** (or **Channel Prev**) and **Trace Next** (or **Trace prev**) to activate the trace of which you want to specify the phase offset.

2. Press **Scale** > **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** (or **Channel Prev**) to activate the channel for which you want to specify the velocity factor.

2. Press **Cal** > **Velocity Factor**, then the velocity factor in the data entry area.

Port Extensions and Loss Values

Overview

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.

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.

The port extension is not available for the Gain-Phase measurement.

- You can set loss values channel by channel. Setting loss values for one particular channel does not affect other channels.

- You can define port extension channel by channel. Setting port extension for one particular channel does not affect other channels.

Enabling port extensions and loss values
1. Press **Channel Next** (or **Channel Prev**) to activate the channel for which you want to enable port extension.

2. Press **Cal** > **Port Extensions**.

3. Turn ON **Extensions**.

Setting port extension and Loss values manually.

Setting electrical delay

Follow the steps below to set the delay for coaxial cable:

1. Press **Channel Next** (or **Channel Prev**) to activate the channel for which you want to set port extension.

2. Press **Cal** > **Port Extensions.**

3. Click **Extension Port 1** or **Extension Port 2** to select the port.

4. Click **Extension** to set the extension in sec.

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.

4. Click **Extension Port n** where n can be from 1 to 2.
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:

:SENS:CORR:EXT:PORT:WAV:CUT

:SENS:CORR:EXT:PORT:WAV:TIME

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 E5063A 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 E5063A stimulus setting to proper range that the calibration kit can cover. Refer to the calibration kit manual or definition in the E5063A with **Cal** > **Modify Cal Kit** > **Define STDs** for the Maximum & Minimum frequency of the calibration kit.

Setting loss values

1. Press **Channel Next** (or **Channel Prev**) to activate the channel for which you want to set loss values.
2. Press **Cal** > **Port Extensions.**
3. Click **Extension Port 1** or **Extension Port 2** to select the port.

4. Click **Loss** to set a loss value.

5. Click **Loss1 [OFF]** to toggle to **Loss1 [ON]** (enabled), and enter a loss value (**Loss1**) and a frequency (**Freq1**).

6. 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**).

- Expression to calculate loss using Loss 1:

$$\text{Loss(f)} = \text{Loss1} \times \sqrt{\left(\frac{f}{\text{Freq1}}\right)}$$

- Expression to calculate loss using Loss 1 and Loss 2:

$$\text{Loss(f)} = \text{Loss1} \times \left(\frac{f}{\text{Freq1}}\right)^{n}$$

$$n = \frac{\log_{10}\left|\frac{\text{Loss1}}{\text{Loss2}}\right|}{\log_{10}\frac{\text{Freq1}}{\text{Freq2}}}$$

- When you specify two frequency points, set the lower frequency to Loss1, and the higher one to Loss2.

Setting a DC loss value

1. Click **Cal** > **Port Extensions** > **Extension Port 1** or **Extension Port 2** to select the port.

2. Click **Loss** > **Loss1 [OFF]** to toggle to **Loss1 [ON]** (enabled).

3. Click **Loss at DC**, then enter a DC loss value.

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 and sets them.

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.

A. Selecting a port(s)

Select the port(s) of which you want to use the auto port extension function.

1. Press **Channel Next** (or **Channel Prev**) to activate the channel of which you want to set auto port extension.

2. Press **Cal** > **Port Extensions** > **Auto Port Extension** > **Select Ports** to select the port(s) of which you want to use the auto port extension function.

B. Setting frequencies used for calculation

Set the frequency points ofh which you want to calculate a loss value.

1. Press **Cal** > **Port Extensions** > **Auto Port Extension** > **Select Ports** > **Method** to set **the frequencies used for calculation**.

2. If you have selected **User Span**, use **User Span Start** and **User Span Stop** to set a start value and a stop value.

3. 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**.

4. If the setting is not made before starting OPEN/SHORT standard measurement, it does not affect the calculation result.
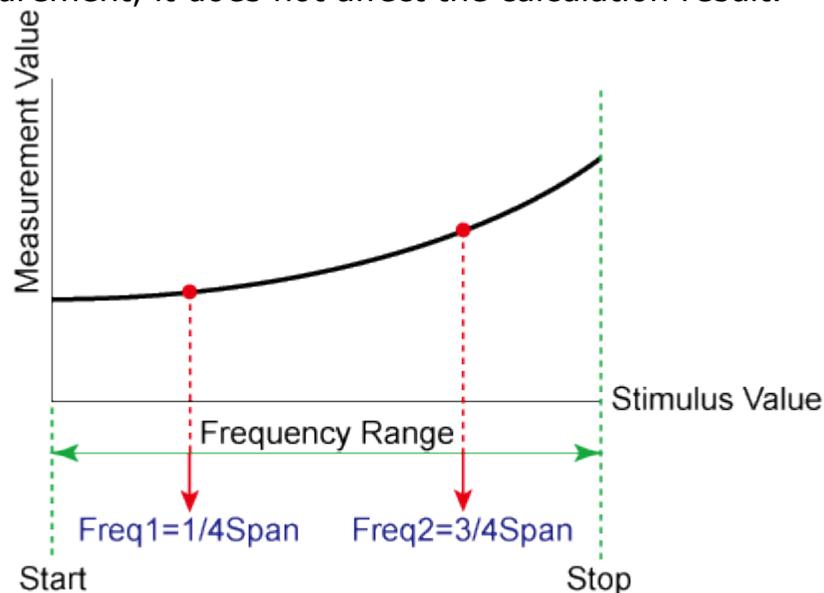
e5071c466

C. Specifying a loss value as a calculation target

Specify whether you want to include a loss value in the calculation result.

1. Press **Cal** > **Port Extensions** > **Auto Port Extension** > **Select Ports** > **Include Loss** to turn it ON.

- If the setting is not made before starting the measurement of the OPEN/SHORT standard, it does not affect the calculation result.

D. 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 **Cal** > **Port Extensions** > **Auto Port Extension** > **Adjust Mismatch** to turn it ON.

- If the setting is not made before starting the measurement of the OPEN/SHORT standard, it does not affect the calculation result.

E. 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 **Cal** > **Port Extensions** > **Auto Port Extension**.

2. If you use the OPEN standard, click **Measure OPEN**, and select the port(s) of which you want to execute measurement. Execution is restricted to ports selected in Selecting a port(s)

3. If you use the SHORT standard, click **Measure SHORT**, and select the port(s) of which you want to execute measurement. Execution is restricted to ports selected in Selecting a port(s).

- If a port extension value or loss value has been set, the value is updated to the calculated result.

- If you execute both open measurement and short measurement, the average of the calculation results is reflected to the port extension and loss value.

F. 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.

- Port extension and loss values that have been calculated are not cleared.

Improving Measurement Throughput

This section explains the methods to improve measurement throughput.

- [Turning OFF the updating of information displayed on the LCD screen](#)

Other topics about Optimizing Measurements

Turning OFF the updating of information displayed on the LCD screen

Turning OFF the updating information feature displayed on the LCD screen eliminates the processing time required to update displays within the analyzer, hence improves the 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**.

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.

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](#)
- [Frequency/Order Base Display](#)

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 delay time, and sweep time for each segment. All segments are swept sequentially as if swept in one sweep operation.

- By skipping the frequency range, which does not need to be measured, you can sweep and measure only the portions you need.

- 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 that has 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.
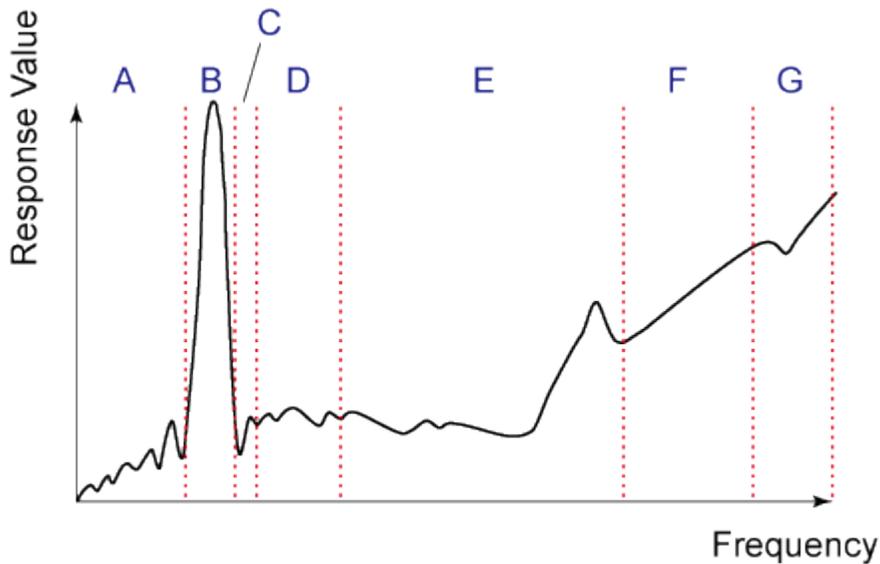
e5071c393

Frequency ranges (segments) from the figure above and their measurement conditions				
	Start frequency	Stop frequency	Number of points	IF Bandwidth
A	440 MHz	915 MHz	50	50 kHz
B	915 MHz	980 MHz	130	70 kHz
C	980 MHz	1.035 GHz	60	50 kHz
E	1.07 GHz	2 GHz	100	70 kHz
G	2.6 GHz	3 GHz	40	70 kHz

Conditions for Setting Segment Sweep

The following conditions apply in 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 100 kHz and the stop frequency of the last segment less than highest frequency, as per the range of the Network Analyzer (depending on the Option, e.g. 4.5 GHz, 8.5 GHz or 18 GHz).

- When the start frequency and stop frequency of a segment are not the same, you can define from 2 to the maximum points in a segment.

- When the start frequency and stop frequency of a segment are the same, you can define from 1 to the maximum points in a segment.
- You can set the total number of points in the segment table from 2 to the maximum points.
- You can set the number of segments in the segment table to between 1 and 201.

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, and sweep time 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 (set with **Avg** > **IF Bandwidth**) is set even if IFBW AUTO is turned ON.
Power level	For all segments, the power level for the linear/log sweep (set with **Sweep Setup** > **Power**) is set.
Sweep delay time	For all segments, 0 is set.
Sweep time	For all segments, the auto sweep time mode 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 below figures.

The definitions for sweep delay time and sweep time when trigger mode is "On Sweep".
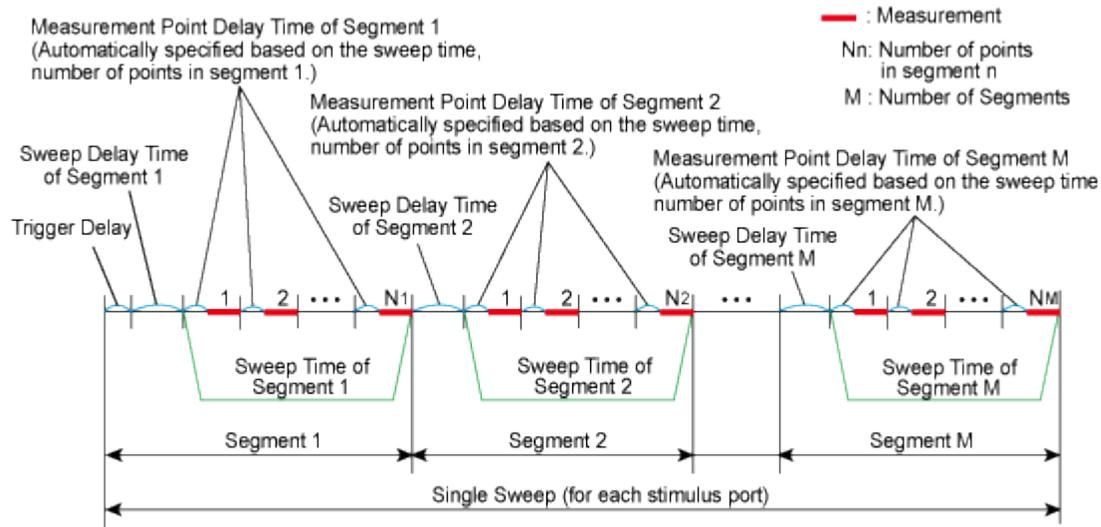
e5061b038

The definitions for sweep delay time and sweep time when trigger mode is "On Point".
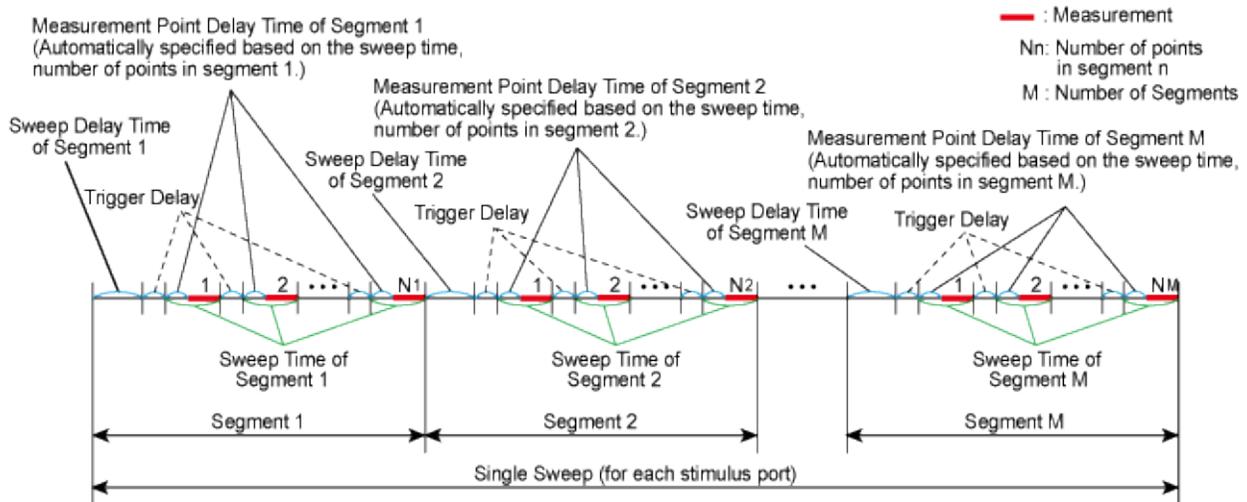
e5061b039

- The segment trigger delay is not included in the segment sweep time.

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

Display — Frequency Based / Order Based

e5071c082
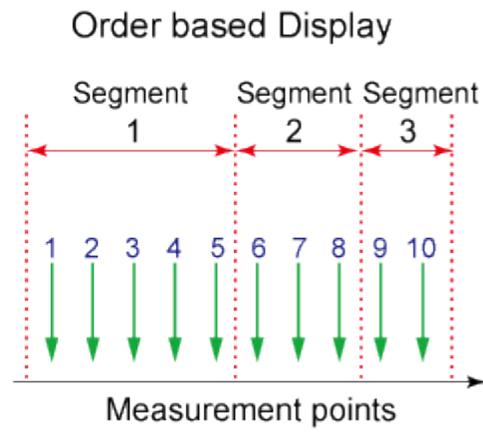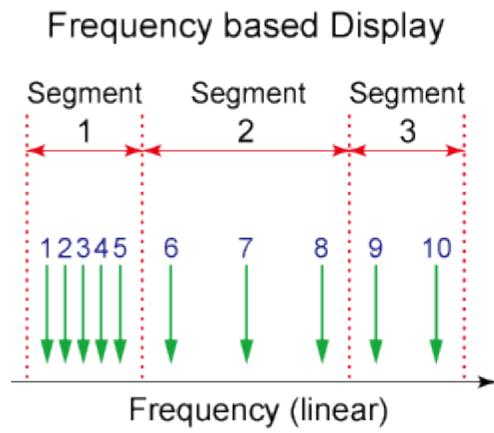
Procedure
Creating a segment table

1. Press **Channel Next** (or **Channel Prev**) to select the channel of which you want to create the segment table.

2. Press **Sweep Setup**.

3. Click **Edit Segment Table**. The segment table appears in the lower section of the screen.

4. Select the **softkey** to change the frequency range setting mode or to set the IF bandwidth, power level, sweep delay time, and sweep time for each segment.

    1. 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 **Foc** 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.

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** > **Power Range$_s$**)
Delay	Sets the sweep delay time
Time	Sets the sweep time; to specify the auto setting (**AUTO**), enter 0 as the sweep time

Useful functions when using a mouse

- You can **copy/paste/insert/delete** the cell by right-clicking on the selected cell.
- In the character-by-character edit mode, you can **undo/cut/copy/paste/delete/select all** by right-clicking 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** (or **Channel Prev**) to select the channel on which you execute the segment sweep operation.
2. Press **Sweep Setup** > **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** (or **Channel Prev**) to select the channel on which you define the segment display.
2. Press **Sweep Setup** > **Segment Display**.
3. Select the **segment 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**.
2. Click **Edit Segment Table** > **Export to CSV File** to open the Save As dialog box. Note that CSV files (***.csv**) is already 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 E5063A, you can set up the segment table.

- It is possible to recall a file from a different channel where it is saved.
1. Press **Sweep Setup**.
2. Click **Edit Segment Table** > **Import from CSV File** to open the Open dialog box. Note that CSV files (***.csv**) is already 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.

330

- You cannot import a CSV-formatted file created/edited in spreadsheet software into the E5063A.

Test Wizard (Opt.011)

Test Wizard (Opt.011)

Test wizard is available when option 011 is installed.

- [Overview](#)
  - [Introduction](#)
  - Test Wizard Screen Navigation
- [Setting Up Test Sequence](#)
- [Executing Test](#)

Overview

Overview

- [Introduction](#)
- Test Wizard Screen Navigation

[Other topics about Test Wizard](#)

Introduction

- [Overview](#)
- [Turning ON Test Wizard](#)
- [Introduction to the Screen Area](#)
- [Front Panel and Other Softkey Functionality](#)

Overview

The E5063A with the installation of Option 011 Test Wizard is designed to perform automated manufacturing go/no-go test of the characteristics impedance of Printed Circuit Boards (PCB). It is highly suitable for in circuit testing, wider bandwidth instruments and PCB antennas testing. It could perform single-ended and differential impedance measurement.

Apart from the PCB manufacturing use, the E5063A Test Wizard is also designed for other manufacturing testing such as for S-parameter and impedance measurement.

e5063a097
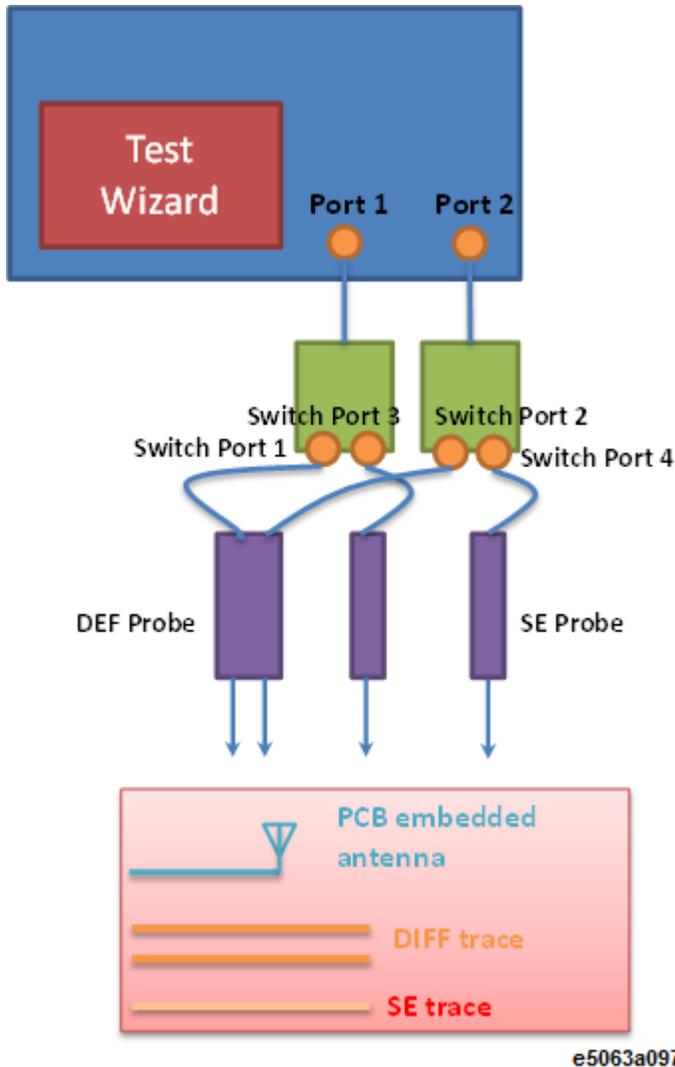
E5063A Test Wizard Manufacturing Test Procedure

Though the characteristics impedance measurement can be performed without the Test Wizard it requires complicated test process/steps. For example, instrument configuration and error correction takes several extra steps and there is no option to set and distinguish between administrator login and operator login. With the Test Wizard, users are guided through the instrument/DUT configuration, error correction and calibration and test sequence arrangement.

Moreover, the Test Wizard also allows the setting of measurement requirement and configuration to be saved with administrator login, allowing only authorized personal to perform and edit the test and measurement configuration. The configured test can be performed as either automated or manual test by any non-skilled operator.

e5063a096

Define Test Parameters/Sequence (Edit Test Mode)

By using the Test Wizard, users have the option to configure and define the test sequence or execute a test sequence. Considering a typical manufacturing environment, the engineer, by using administrator login, which is the Design Mode, could edit the instrument configuration, perform calibration and error correction, edit test sequence as well as edit test list. The Test Wizard guides easy to follow step by step procedure to configure the instrument as per the physical connection and then perform calibration according to the configuration. In Edit Test mode, the test sequence can be set as well; either manual or automatic, the type and location of the output and some advance setting that can help the non-technical operators perform the test easily. The type of measurement and format can also be set at this mode.

The Edit test mode allows users to set and perform several different test limits including absolute test, average and multi-segment in time domain measurement. In S-Parameter measurement, only multi-segment test-limit is allowed.

Production Test Mode (Execute Test Mode)

In Execute Test mode, operators can perform the test measurement according to the sequence defined in the Edit mode. With password, the users of this mode have limitation of the screen they can access to. This prevents the non-skilled operators from changing the settings by mistake. With the configuration being set in the Design Mode, the users at the

production only need to execute the test as defined. As the tests are being executed, the Test Wizard shows whether a test or series of tests have passed or failed. The users can save measurement result with the serial number of the DUT to ease future retrieval.

The structure of the Test Wizard is as shown below.
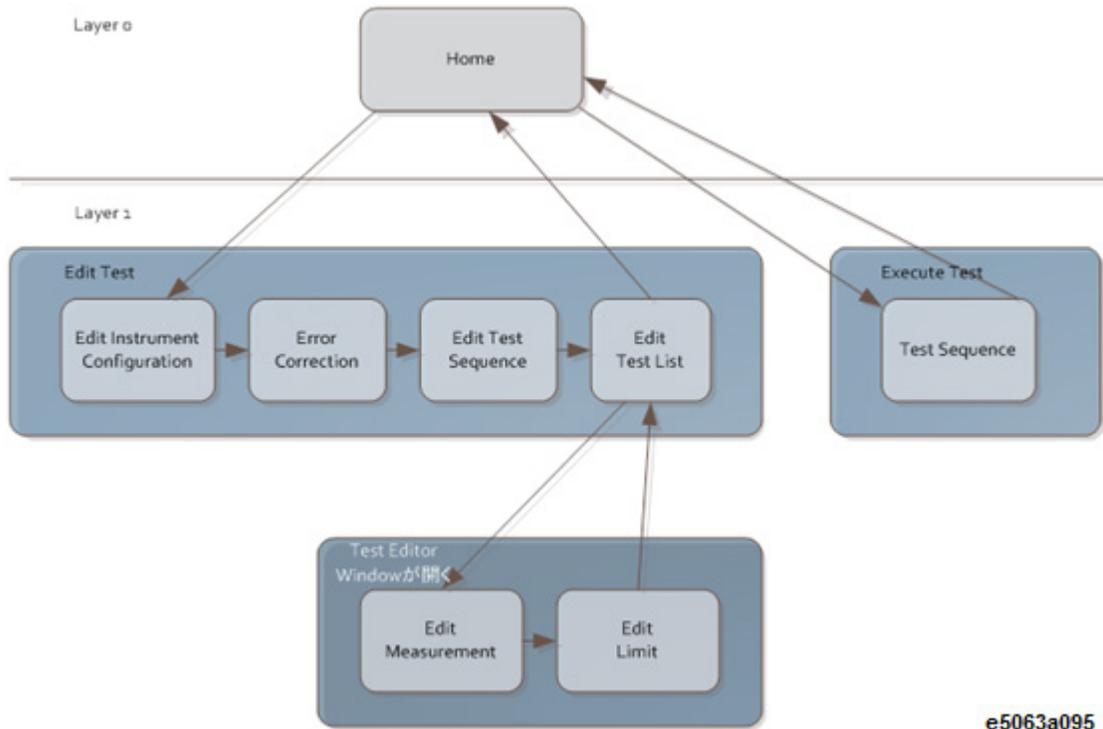
e5063a095

Turning ON Test Wizard

To turn ON the Test Wizard, select **Analysis** > **Start Test Wizard**.
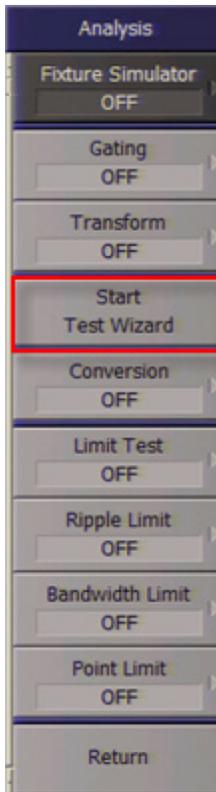
e5063a007

Introduction to the Screen Area

e5063a008

Graph Area

The test result of the Time Domain and S-Parameter is displayed here. Only one trace is displayed. Multiple trace display is not supported.

Navigation Bar

The active menu is shown here. You can toggle from one menu to the other by clicking on the arrows.

Setting Area

The setting area changes and is displayed according to the selected menu.

Function Area

Available operation key for the active menu is displayed here.

Front Panel & Other Softkey Functionality

Front Panel Lock and Softkey

e5063a034

The hardkeys highlighted above and softkeys associated with them become inactive when Test Wizard is running active. However, when the Advanced Mode is turned ON, these hardkeys and softkeys are activated.

> **NOTE** Some keys activated under the Advanced Mode may collide with some functions under the Test Wizard. Hence, use the keys activated under Advanced Mode with attention.

Entry Box

e5063a045

When numeric entry box is clicked, the Entry Box as shown above is displayed.

The units displayed on the Entry Box (right most) varies depending on the numeric entry box.

Setting Up Test Sequence

Setting Up Test Sequence

- [Selecting Language](#)
- [Setting Up New State and Recalling State](#)
- [Configuring Instrument](#)
- [Configuring Error Correction](#)
- [Configuring Test Sequence](#)
- [Configuring Test List](#)

[Other topics about Test Wizard](#)

Selecting Language

The drop-down button below **Select Language** label, provides you option between English, Traditional Chinese, Simplified Chinese, Korean or Japanese language. The language selection determines the language in which the Test Wizard will be displayed. However, the labels in the measurement screen and the softkey menu displayed in the **Advanced Mode** are in default English language.

This language selection is applied only to the Test Wizard area. The language for other area including soft keys, cannot be changed.

NOTE   Do NOT change the settings of the Standards and Formats in Regional Options and Languages of Windows OS from default setting (English).

Other topics about Setting Up Test Sequence

Setting Up New State and Recalling State

- New Test
- Recall Test
- Other Home Menu

Other topics about Setting Up Test Sequence

New Test

Begins a new test. Previous test data will be lost once New Test is selected.

1. Click on **New Test** button.
2. Below dialog box appears. Click **OK** to continue.
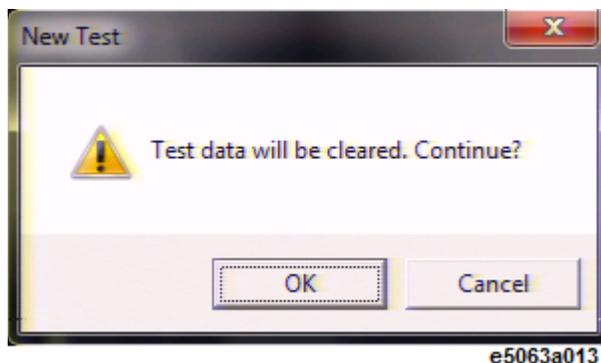
3. The earlier test data will be lost and all settings are changed to default settings.

Recall Test

Recall test setting state file saved earlier. The supported test wizard state file format is *.twz.

1. Click on the **Recall Test** button.
2. Below open file dialog box appears.
3. Select the desired state file and click **Open**.
4. The earlier test data will be lost and all settings are changed to the recalled state file settings.

Other Home Menu

Save Test

Saving the existing test setting and overwriting earlier test setting.

Save Test As

Saving the existing test setting by assigning a filename.

Home

Returns to main menu.

About Box

e5063a011

Displays About E5063A Test Wizard box, which shows the test wizard and firmware revision, serial number and build date.

Information

Displays the test setting information - test file name, topology setup, number of tests, type of error correction selected and the test option.

Configuring Instrument

- Selecting DUT Topology
- Configuring More Function

Other topics about Setting Up Test Sequence

The display of Edit Menu depends on the security mode. In Administrator Security Mode, password is required to enter the Edit Test and perform changes.

Selecting DUT Topology

To select the DUT Topology:

1. Click **Edit Test** at *Home* menu.

2. Click **Modify** at *Edit Test* page.

3. Instrument Configuration Wizard is executed and *Select DUT Topology* dialog box appears.

4. Define the external switch connection setup (U1810B switch) as well as DUT topology, click on the option matching the physical connection and click **Finish**.
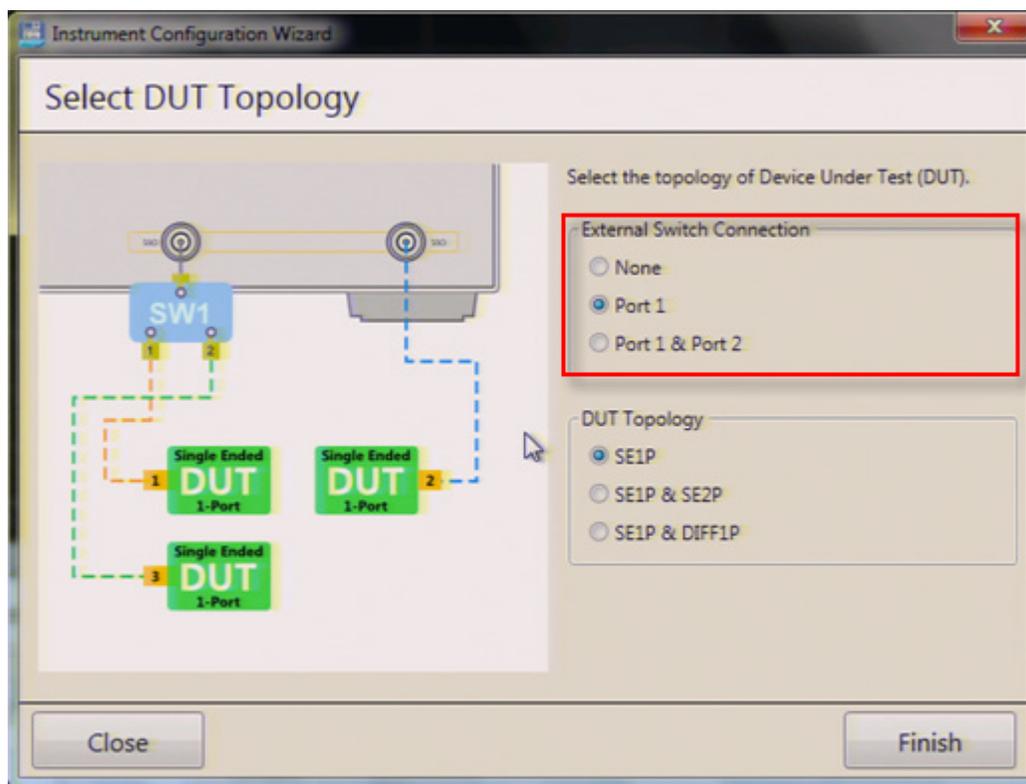
U1810B USB Coaxial Switch SPDT (Single-Pole Double-Throw)

Test Wizard can connect up to 4 ports when you use U1810B switch. However, you can make only 2 port-measurement at any one time.

You can connect up to 2 switches to E5063A,which means you may make, up to 4 DUT port measurement. When you connect only one switch, you must connect it to port 1 of the E5063A.

If the number of connected U1810B switch is set at 2 but only 1 switch is connected:

- In Edit menu: Warning message pops up.
- In Executed Test menu: Error message pops up.

To learn more about U18010B USB Coaxial Switch SPDT, visit www.agilent.com.

In the Test Wizard, DUT port number is assigned to physical DUT port number is used for control purpose.

Correlation of number of switch with the maximum number of ports is shown in the below table:

DUT port number assignment

Figure #	No. of U1810B switches	Maximum no. of DUT ports	DUT Port 1	DUT Port 2	DUT Port 3	DUT Port 4
1	0	2	E5063A Port 1	E5063A Port 2	N/A	N/A
2	1	3	U1810B Port 1	E5063A Port 2	U1810B Port 2	N/A
3	2	4	U1810B Switch 1 Port 1	U1810B Switch 2 Port 1	U1810B Switch 1 Port 2	U1810B Switch 2 Port 2
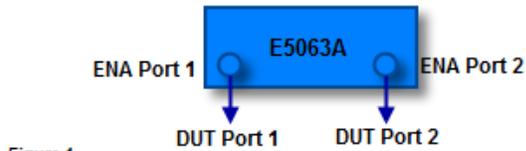
Figure 1
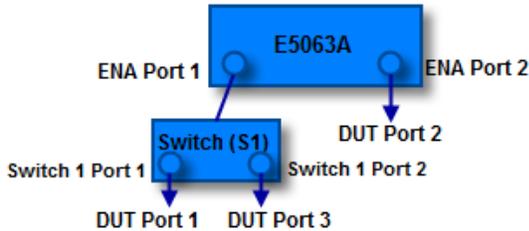
Figure 2
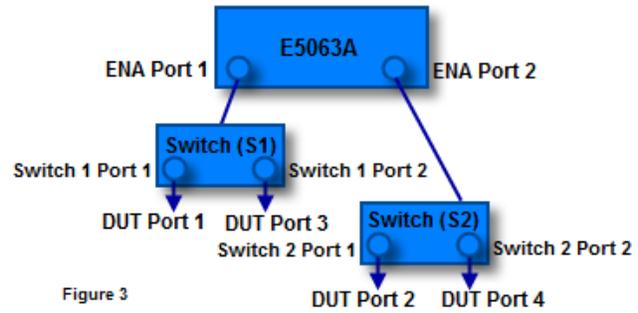
Figure 3

e5063a20

Typical Configuration and DUT Topology

Number of your DUTs			Required Number of USB Switch	Port Connection (DUT Port Number: Px)		
Single-end 1-Port DUT	Single-end 2-Port DUT	Differential 1-Port DUT (Physical 2 Port)		Single-end 1-Port DUT	Single-end 2-Port DUT	Differential 1-Port DUT
1	0	0	0	P1 or P2	-	-
1	1	0	1	P3	P1-P2	-
1	0	1	1	P3	-	P1-P2
2	0	0	0	P1,P2	-	-
2	1	0	2	P3	P1-P2	-
2	0	1	2	P3,P4	-	P1-P2
3	0	0	1	P1,P2,P3	-	-
4	0	0	2	P1,P2,P3,P4	-	-
0	1	0	0	-	P1-P2	-
0	0	1	0	-	-	P1-P2
0	2	0	2	-	P1-P2, P3-P4	-
0	0	2	2	-	-	P1-P2, P3-P4

347

Single Ended 1-Port Topology with no Switch connected
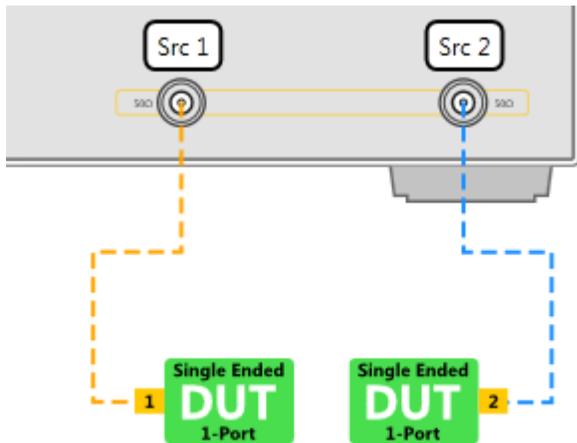
Single Ended 2-Port Topology with no Switch connected
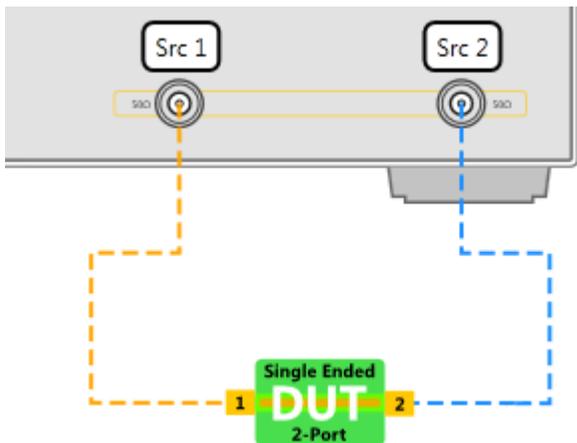
Differential 1 Port Topology with no Switch connected
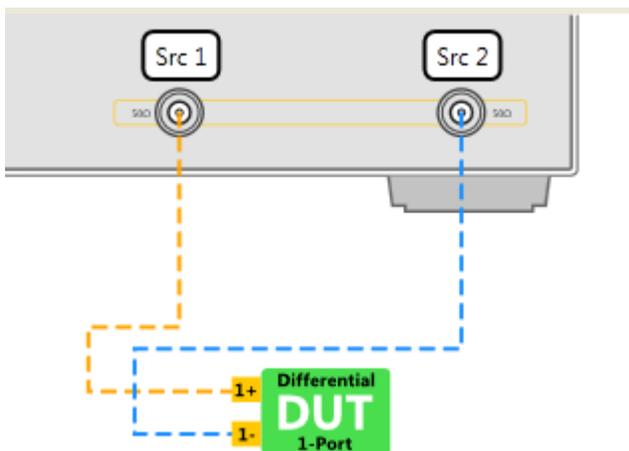
Single Ended 1-Port Topology with 1 Switch
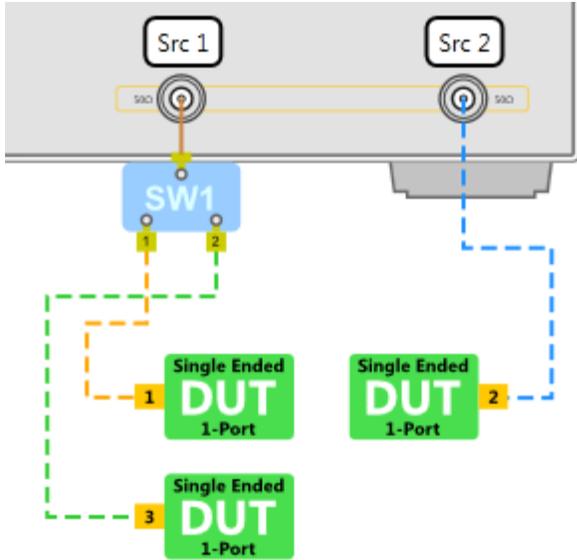
Single Ended 1-Port and Single Ended 2-Port Topology with 1 Switch
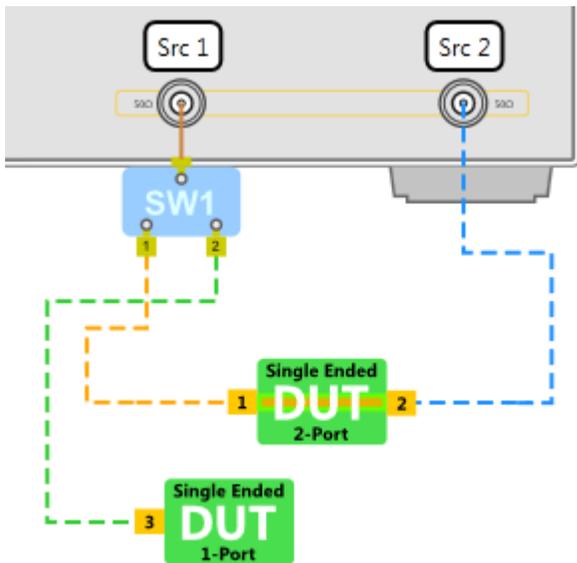
Single Ended 1-Port and Differential 1-Port Topology with 1 Switch
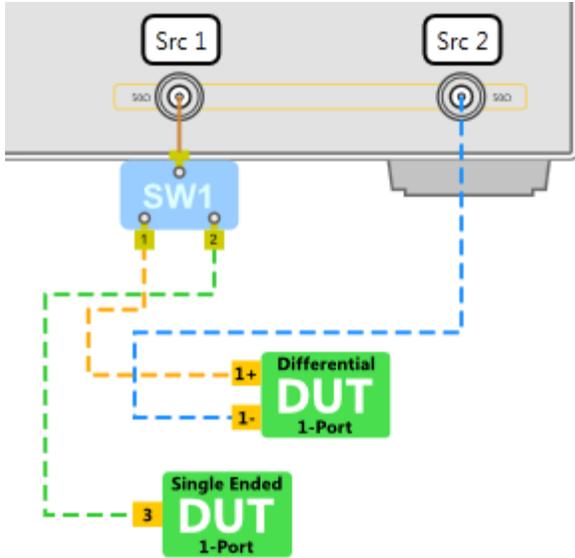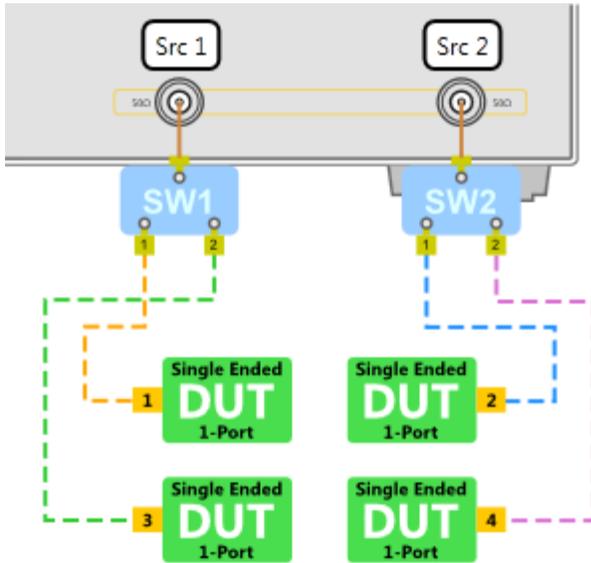
Single Ended 1-Port Topology with 2 Switches

Single Ended 2-Port Topology with 2 Switches
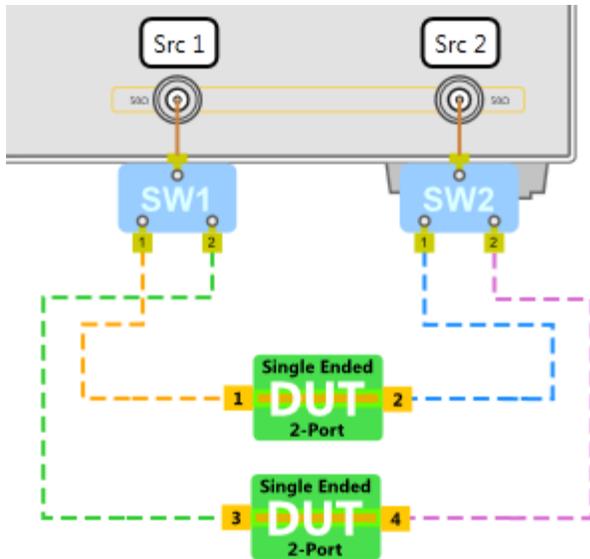
Single Ended 1-Port and Single Ended 2-Port Topology with 2 Switches
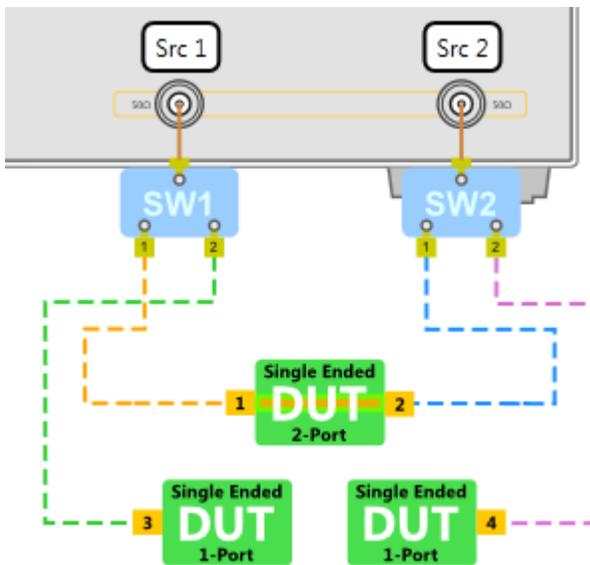
Differential Single Ended 1-Port with 2 Switches
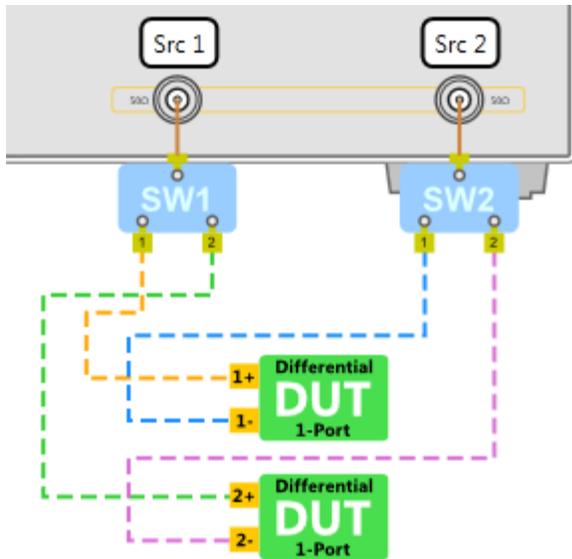
Single Ended 1-Port & Differential 1-Port Topology with 2 Switches

Configuring More Function

e5063a016

Ref.Z

Reference Impedance

Dielectric Constant

Editing dielectric constant changes the value of velocity factor automatically.

Velocity Factor

Related to dielectric constant, hence, editing dielectric constant automatically changes the value of velocity factor and vise versa.

X-Axis Unit

The unit of X-axis. Selection choices are Second, Meters or Inches.

Couple Rise Time

Couples the rise time between test items.

Advanced Mode

Activates the advanced mode, where the E5063A softkey menu are activated.

Consideration in Advanced Mode

In the Advanced Mode, you can access all setting of E5063A. The setting you changed may affect the measurement unexpectedly. The measurement may not be correct if you have such a case. Therefore, the measurement performance is not guaranteed in the Advanced Mode.

It is known that changing the following setting causes incorrect measurement.

Hard key	Do not use
Meas	Do not change the parameter
Sweep Setup	Start/Stop/Center/Span
	Points
	Sweep Type
Marker Function	Marker -> Start, Marker -> Stop

Here is another caution in the Advanced Mode.

Function		Note
Display	Increasing Num of Trace	This may cause to tilt the wave form in the time domain.
Analysis	Fixture Simulator	This may cause to tilt the wave form in the time

		domain.
	Transform > Window > Impulse width	This may cause the setting change.
Calibration	Increasing Port Extension	This may cause to exceed the DUT length limitation.
Scale	Increasing Electrical Delay	This may cause to exceed the DUT length limitation.
Ave	IF Bandwidth	Even if you have narrow IF Bandwidth, you may not have noise reduction at lower frequency.
Display	Data Math, Equation Editor	The data and memory positions in data processing are different.

To configure more functions:

1. Click **Edit Test** at *Home* menu.

2. List of functions are found under **More Functions** group. Enter or select the desired value:

    - Reference impedance - by default its set to 50 Ω.

- Dielectric constant & Velocity Factor - when you change the dielectric constant, the velocity factor is changed automatically according to the below formula. This value is used to convert from electrical length to physical length at X-Axis and Marker reading:

  - Velocity Factor = 1/sqrt(Dielectric Constant)

- X-Axis Unit - can be toggled between time and length. Length can be defined in meters or inches.

- Couple Rise Time - check this option to couple Rise Time between test items.

- Advanced Mode -

  1. Click on this button and below dialog box appears.
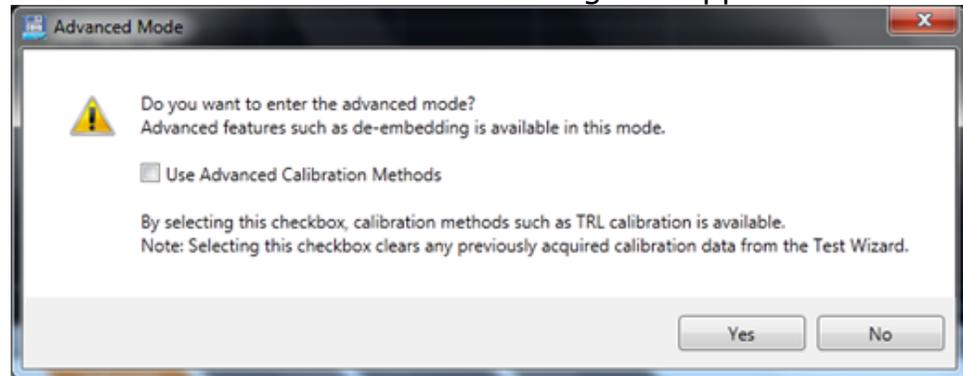
e5063a017

  2. When you want to perform any calibration such as 8 term-calibration in Advanced Mode, select the check box named **Use Advanced Calibration Methods** in dialog box.

     - In advance calibration methods:

       - Deskew and DLC can not be used because its accuracy becomes worse.

       - The calibration you did in basic mode is cleared.

  3. Click Yes to start the Advanced Mode.

  4. The following settings are changed.

     - Softkeys are displayed on the right side of screen.

     - All hard keys are unlocked.

     - **SVC** in the instrument status bar turns in blue.

  5. Once Advanced Mode is activated, the button is replaced by **Basic Mode** button as shown below. Click on the this

button to return to normal mode, which presets the firmware.
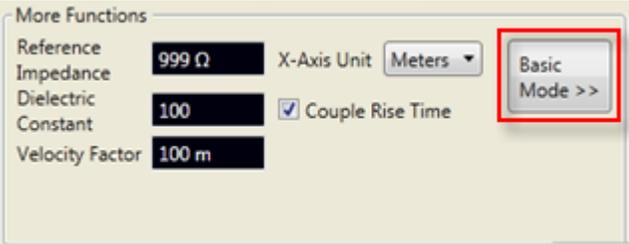
e5063a018

Configuring Error Correction

- Performing Error Correction with Wizard
- Performing Manual Error Correction

Other topics about Setting Up Test Sequence

Performing Error Correction with Wizard

Refer to Error Correction Menu for brief explanation of the Error Correction Wizard dialog box. Below is an example of performing error correction by using wizard. In this example, error correction for 2 Single Ended 1 Port (SE1P) setup is performed, where Port one with Deskew & Loss and Port 2 with ECal. :

1. Click the Error Correction Wizard button to execute the Error Correction Wizard.
2. The pages shown varies according to the error correction method selected. At the **Select Method** page, select the error correction method for the active ports, in this example, both Port 1 (Deskew & Loss) and Port 2 (ECal).
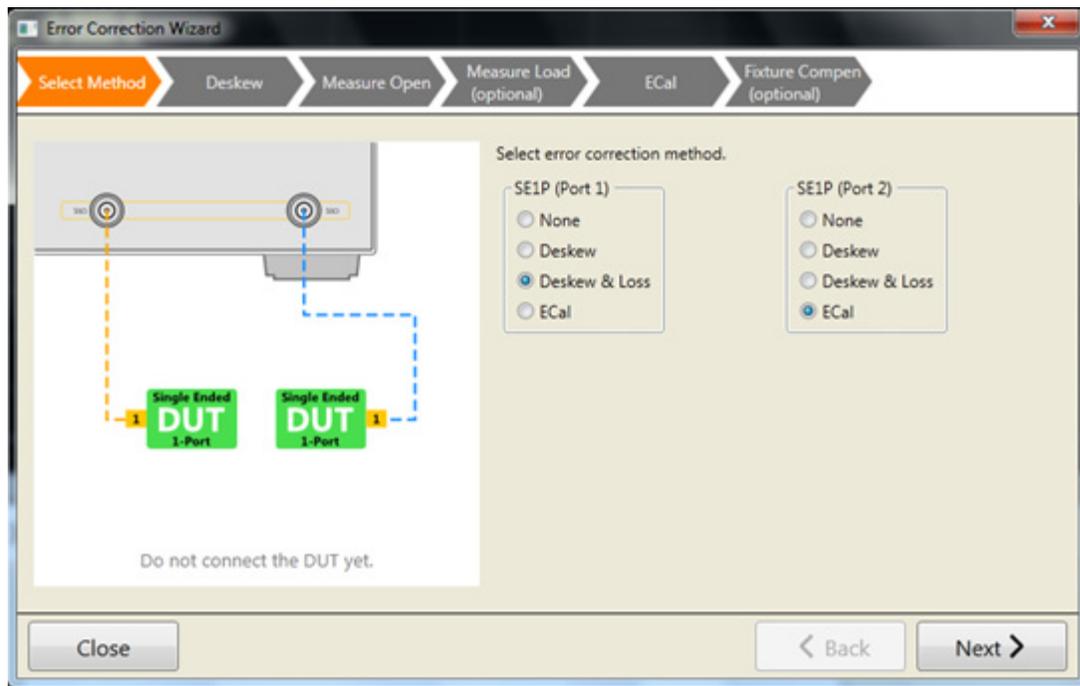
e5063a035

3. At **Deskew** page, connect all cables and probes to the instrument, however, ensure to disconnect DUT from cables and probes. Then, press the **Deskew Port [ ]** button.
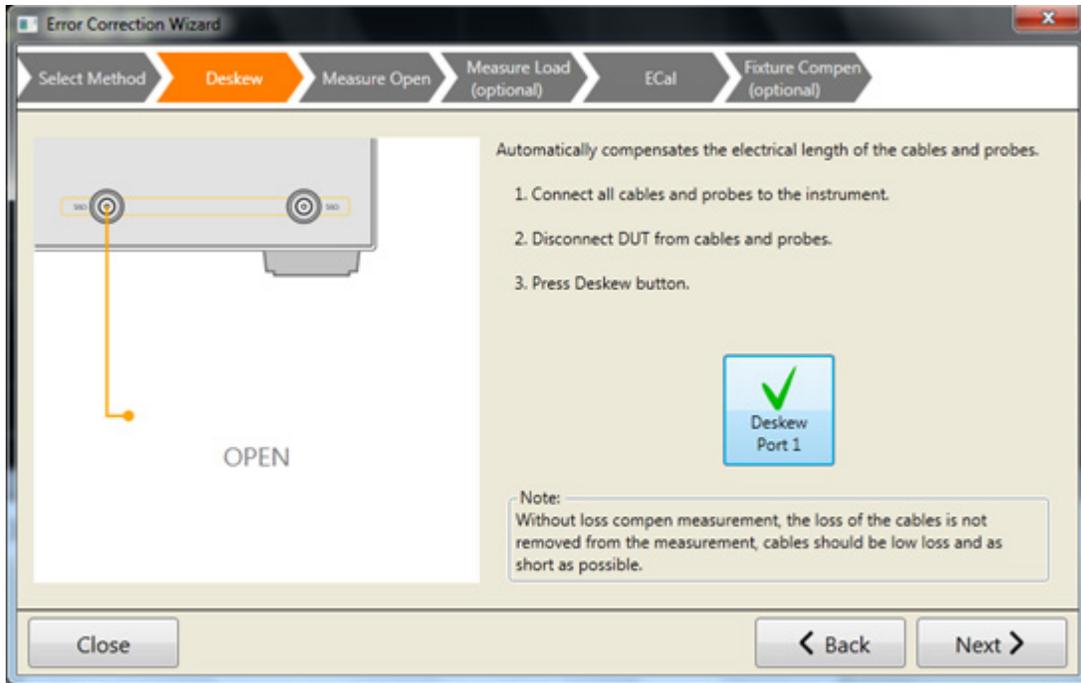
e5063a036

4. At **Measure Open** page, connect thru to the relevant port, in this example, Port 1 and click **Measure Port [ ]** button.
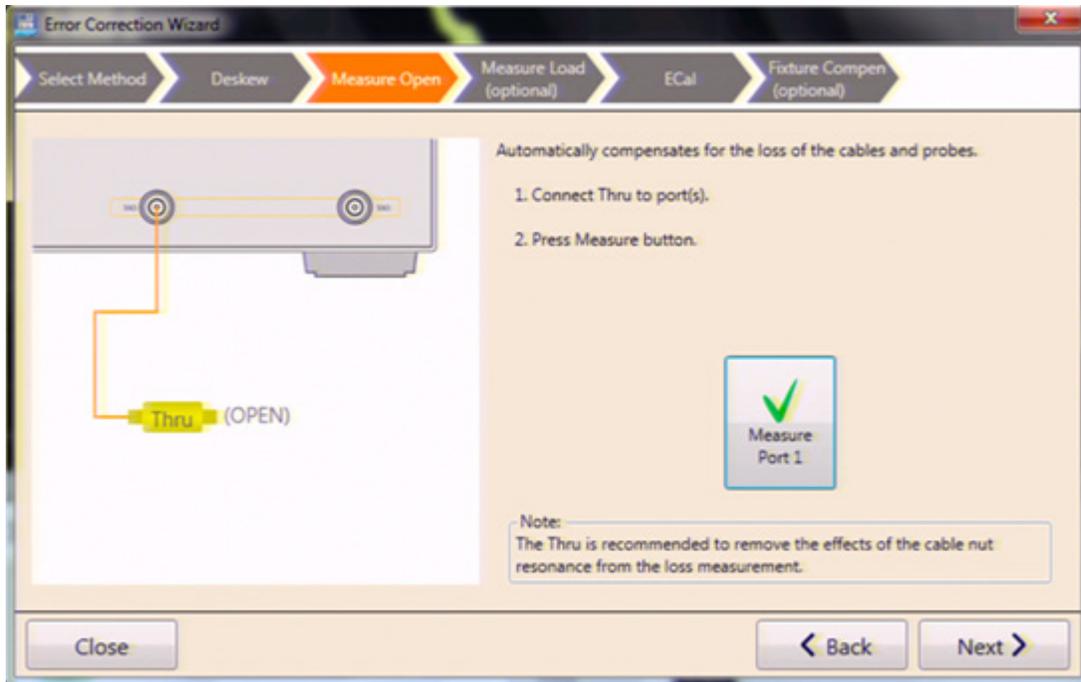
e5063a037

5. At **Measure Load** page, connect load to the relevant port, in this example, Port 1 and click **Measure Port [ ]** button.

e5063a038
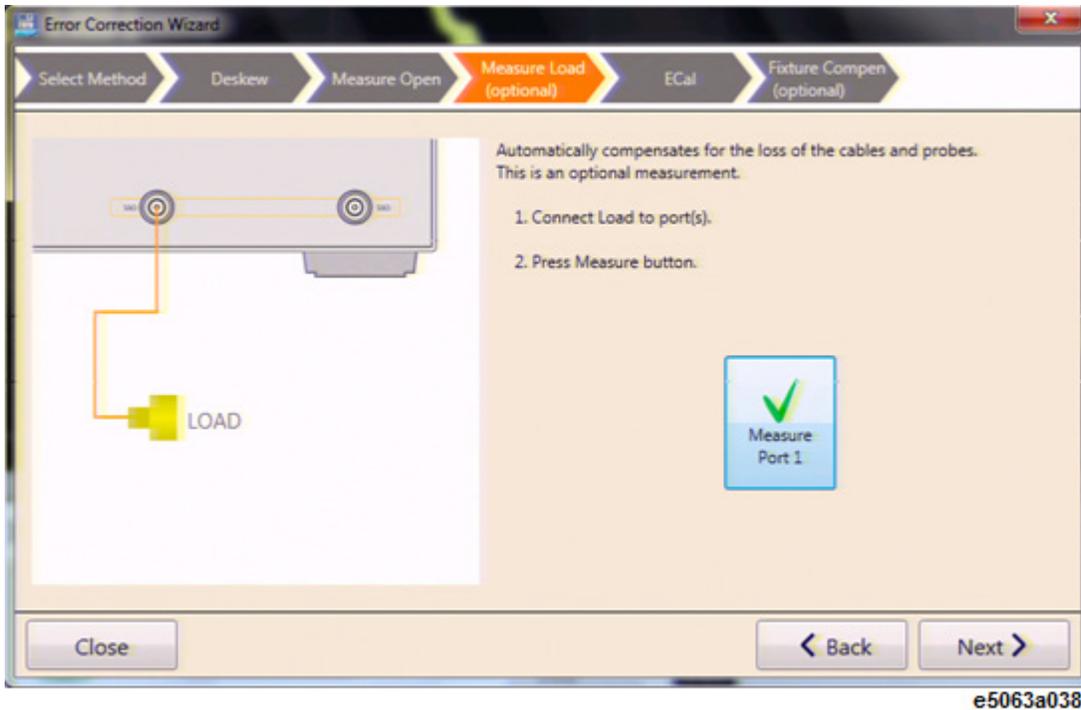
6. At **ECal** page, connect ECal module to the relevant port, in this example, Port 2 and click **Calibrate Port [ ]** button.

e5063a039
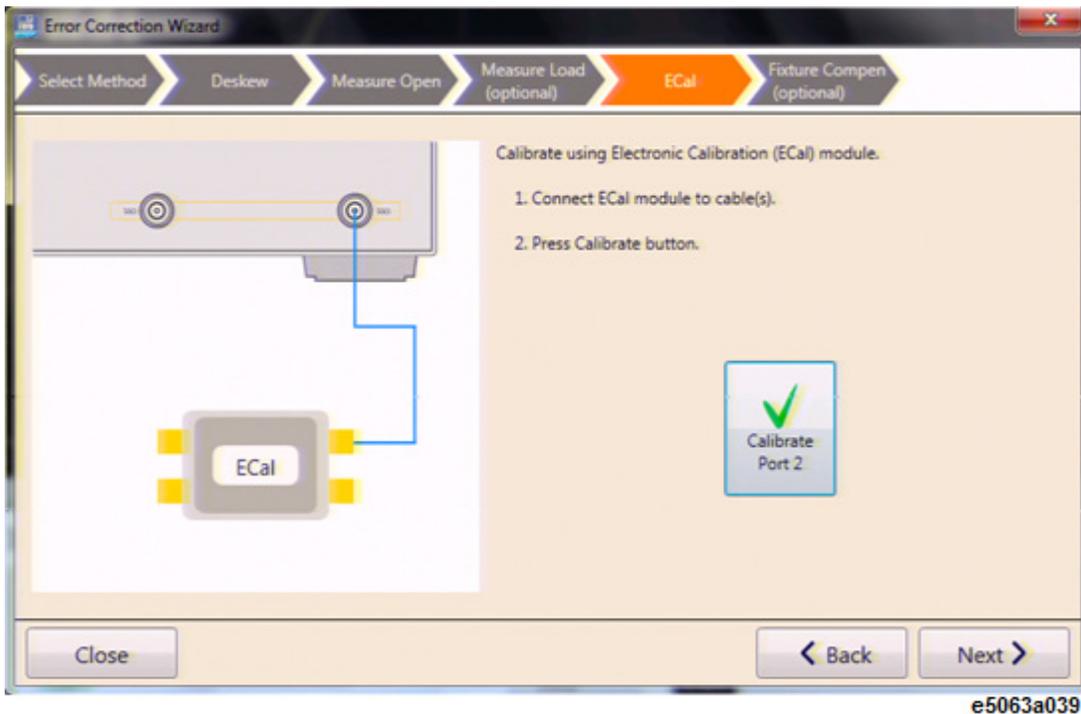
7. At **Fixture Comp** page, once again, connect all cables and probes to the instrument, however, ensure to disconnect DUT from cables and probes. Then click the **Fixture Comp Port [ ]** button.

e5063a040
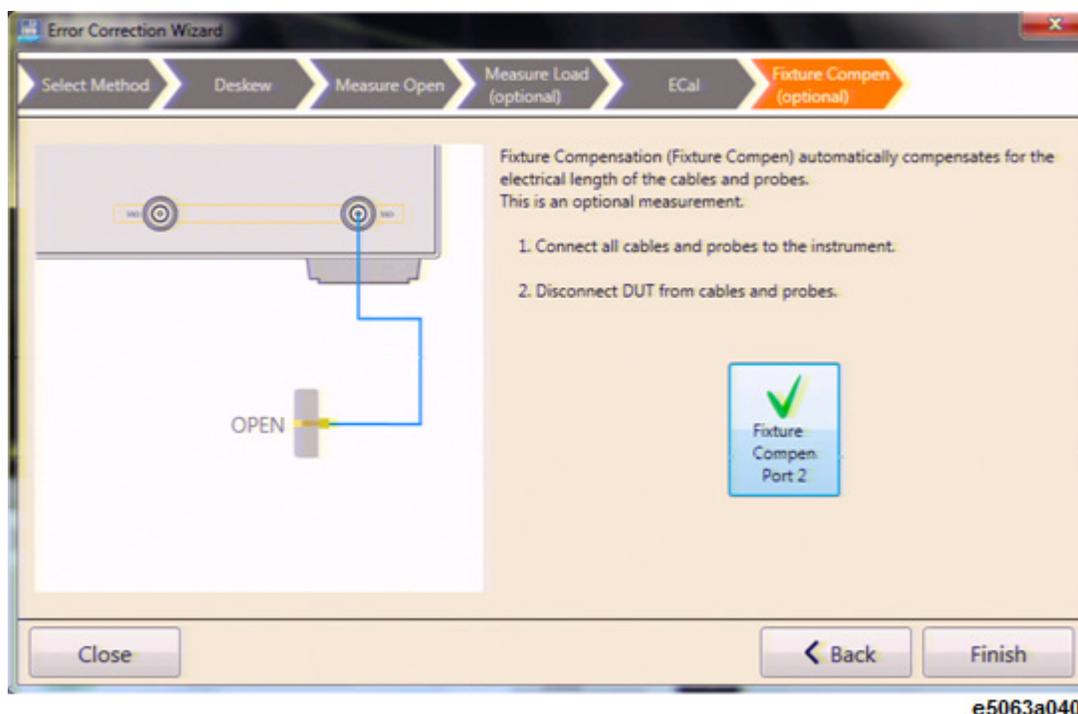
7. Upon completion, click the Finish button which completes the Error Correction performance.

Performing Manual Error Correction

Instead of using the Error Correction Wizard, you can also perform error correction manually. Refer to Error Correction Menu for brief explanation of the screen.

Below is an example of performing error correction manually. In the same example as above, error correction for 2 Single Ended 1 Port (SE1P) setup is performed, where Port one with Deskew & Loss and Port 2 with ECal. :

e5063a021
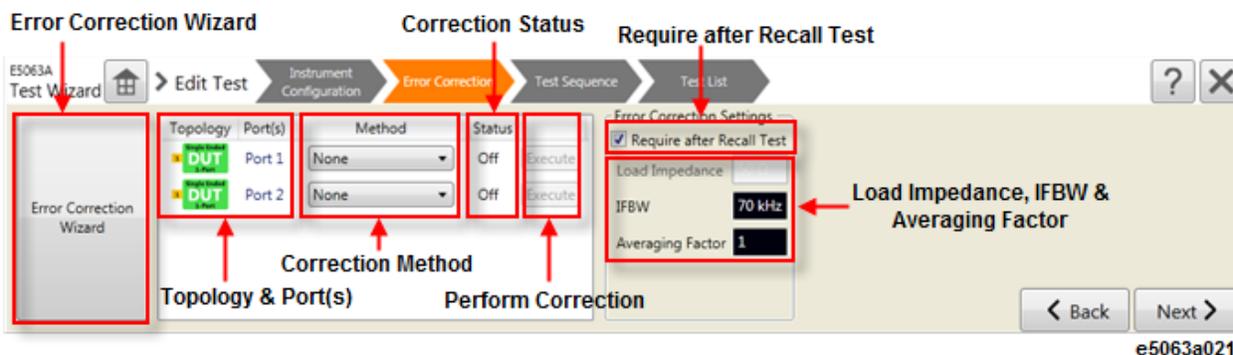
Topology & Port(s)
Topology and Port(s) are display only elements. Topology setup and Ports setup are performed at Instrument Configuration Menu.

Correction Method

Click on this button to select correction method between the given option - Deskew, Deskew & Loss and ECal. Once correction method is selected, Correction Status is highlighted in red indicating that its yet to be corrected.

Correction Status

By default, the Correction Status indicator is **Off**, in black. When the Correction Method is selected but Error Correction is not performed, the indicator change to **Off**, in red. When the Error Correction is performed with the selected Correction Method, the indicator change to **On**, in black.

Perform Correction

Performs Error Correction only for the corresponding DUT.

Require after Recall Test

Once checked, Error Correction should be performed once a test is recalled. If unchecked, Error Correction is not required.

Load Impedance, IFBW & Averaging Factor

Allows you to determine load impedance, frequency bandwidth and averaging factor.

1. For each port, click on the correction method drop-down button and select accordingly. In this example, for Port 1, select **Deskew & Loss** and for Port 2, select **Ecal**.

   > NOTE   Select correction method for all the ports prior performing correction. If you select correction method for the next port after performing correction for one port, you may lose the error correction data.

2. Once correction method is collected, correction status is highlighted in red as **Off** and **Perform Correction** buttons are activated.

3. Click on the Perform Correction button and Error Correction Wizard prompts, similar to the above.

4. Other factors that you can configure manually are:

   - If error correction is required to be performed when a test is recalled.
   - Load impedance
   - frequency bandwidth
   - averaging factor

Configuring Test Sequence

- [Setting Up Test Sequence](#)
- [Setting Up File Output](#)
- [System Security](#)

Other topics about Setting Up Test Sequence

Upon completion of error correction performance, perform test sequence configuration, at the Test Sequence Menu page.

Setting Up Test Sequence

At Test Sequence area, set the below properties:

- **Auto Advance: On | On Pass | Off** - If On is selected, next test is selected automatically and executed once the earlier test is completed. If On Pass is selected, next test is selected only when the earlier test has passed. If Off is selected, next test has to be manually selected to be executed once the earlier test is completed.

- **Beep: On | On Pass | On Fail | Off** - If On is selected, the beeper goes off once a test is completed. Other options include turning off the beeper, beeper goes off when a test pass and beeper goes off when a test fails.

- **Auto Scale: On | Off** - When turned On, performs auto scale after measurement is completed.

- **Fail Sign: On | Off** - When turned On, displays fail sign on the graph area.

- **Operator Instruction Window: On | Off** - When turned On: Enlarge/popup the [operator instruction dialog](#) when users move to the next test item or enter to Execute Test. When turned Off: Does not show the [operator instruction dialog](#).

Setting Up File Output

At the File Output area, set the below properties:

- **Output File Path** - Path where result file is saved. Select the path by clicking the **Browse** button.

- **Auto Save Results** - Prompts the save dialog box automatically after all test is done.

- **Auto Save Trace** - Saves the trace data after all the test are executed and completed.

- **Auto Save Touchstone** - Saves the SnP file after all the test are executed and completed.

System Security

At the System Security area, set the below properties:

- **Operator** - Allows you to open Edit Test menu without any password.
- **Administrator** - Password is required to open Edit Test Menu.
- **Password** - Password for Administrator mode.

Configuring Test List

- [Introduction](#)
- [Editing/Adding/Deleting Each Test Sequence](#)
- [Defining Each Test](#)
- [Defining Test Limit](#)
- [Making Measurement for Confirming Result](#)

Introduction

Test List Display

Displays the test information as below:

- **Test No** - The test index number.
- **Description** - Test description as entered by user. If description is not specified, "Test N" is displayed, where N is number.
- **DUT** - Selected DUT topology.
- **Parameter** - Measurement parameter.
- **Format** - Measurement format, e.g. Impedance, Linear, Log Mag etc.
- **Limit Type** - Absolute Limit when Time Domain is selected and Multi Segment when S-Parameter is selected.

Others

- **Add** - Adds a new test at the end of the test list.
- **Delete** - Deletes the selected test.
- **Edit** - Edits the selected test. Opens the Measurement Setup Menu where the test configuration is edited.

Editing/Adding/Deleting Each Test Sequence

Editing Test Sequence

1. Select on the test you intend to edit.
2. Click the **Edit** button.
3. Measurement Setup Menu opens where the test configuration is edited. Refer to [Defining Each Test](#) and [Defining Test Limit](#).

Adding Test Sequence

1. Click the **Add** button.
2. A new line of test is added below the last row of test.
3. Select the new test.
4. Click the **Edit** button to configure the new test.

Deleting Test Sequence

1. Select on the test you intend to delete.
2. Click the **Delete** button.
3. The test is removed from the test list.

Defining Each Test

You can define each test at Measurement Setup Menu page.

Test Description

- **Test No** displays the sequence number of the test.

- **Description** entry box allows you to enter the descriptive statement about the test. By default, "Test N" is displayed, where N is number.

Operator Instruction

- **Operator Instruction** - Click on the **Setup** button and the **Operator Instruction Wizard** shows where you can select the instruction type (default, text and picture) and edit accordingly. The operator instructions dialog box is displayed at the beginning of each test. As for picture, you can select *.png file of any size however, 512 × 320 pixel picture fits the frame well. Picture of other sizes is resized automatically if its larger or smaller than 512 × 320 pixel.

- The operator instructions dialog box can be disabled with Operator Instruction Window: On | Off.
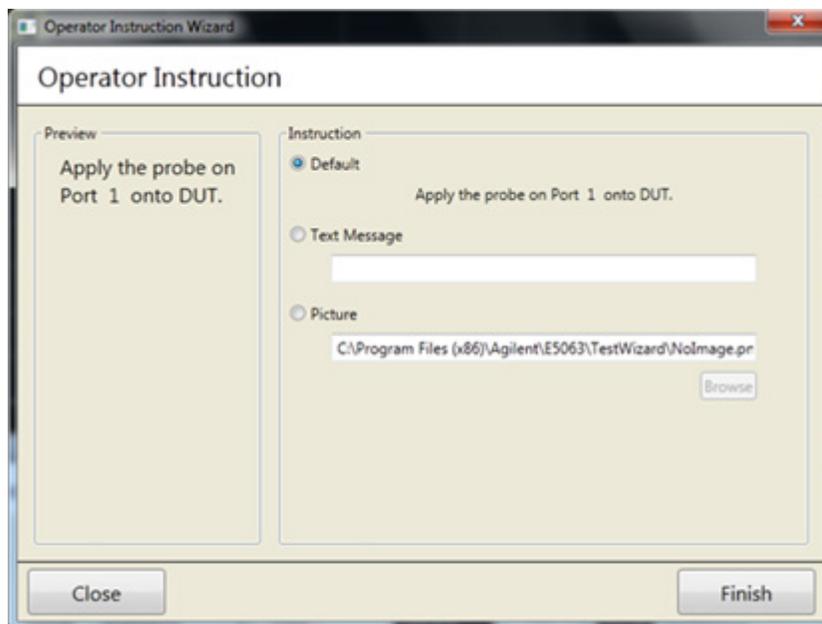
e5063a027

Measurement

- **Measure** - Allows you to select between Time Domain and S-Parameter measurement. The measurement selection determines the limit selection, located at the right side of the **Measurement Setup Menu**.

- **Parameter** - Parameter selection is also determined by the measurement selection. Parameter options for Time Domain selection are T11, T21, T12, T22, Tdd11, Tdc11, Tcd11 and Tcc11. S11, S21, S12, S22, Sdd11, Sdc11, Scd11 and Scc11 are for S-Parameter measurement. The available selection is depend on the DUT Topology.

- **Format** - Allows you to select the format type. The list of available format is determined by the measurement selection.

- **Averaging** - Check this box to trigger the averaging function, where a number of sweep is executed with a single trigger. The number of sweep is determined by the averaging factor.

- **Averaging Factor** - Determines the number of sweep executed with a single trigger when averaging function is activated.

- **IFBW** (Frequency Bandwidth) - Determine the frequency bandwidth. The default value is 70 kHz.

Time Domain/S-Parameter

This section is determined by the measurement selection:

Time Domain

- **DUT Length** - DUT length can be specified for each test independently. The DUT length is used to calculate the limit test range and auto scale for X axis. The longest DUT length in each test is used for actual measurement setting. The DUT length is displayed at the Test List Menu. When the longest DUT length is changed to above 6.96 ns(*), the memory trace in all the Time Domain tests are cleared. However, when the longest DUT length is set within the range below 6.96 ns(*), the memory trace are not cleared and remain unchanged. To automatically set the DUT Length, click on the **Auto** button to execute the **DUT Length Wizard**.

    NOTE   (*) This value is depending on the frequency option. 6.96 ns (Opt. 2H5), 14.8 ns (Opt. 285), 27.9 ns (Opt. 245).

e5063a028
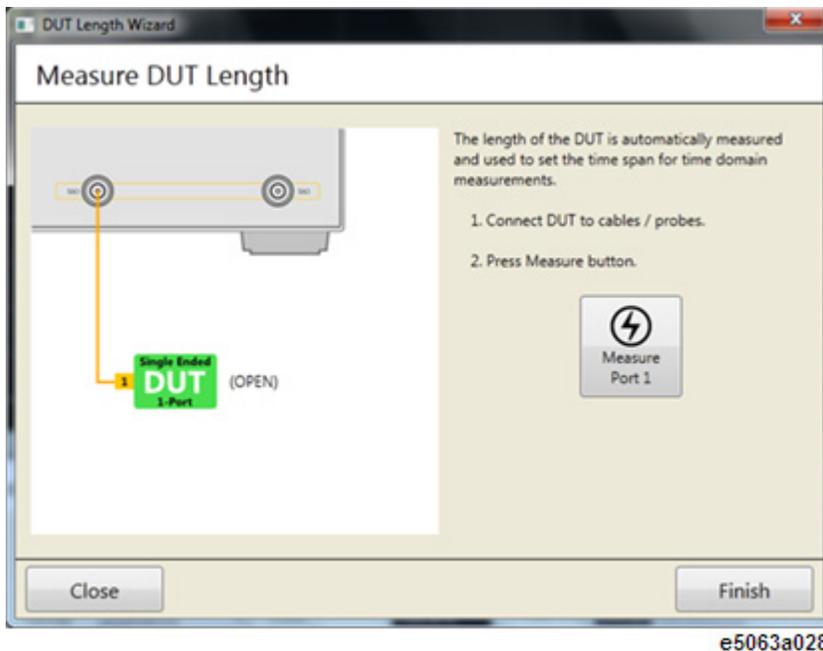
- Connect the DUT to cables and probes.
- Click the **Measure** button.
- Once done, click **Finish**.

- **Rise Time** - Allows you to set the rise time and its threshold, with the option of 10%-90% or 20%-80% of the step signal.

- **Peeling** - If checked, allows peeling analysis to be applied.

S-Parameter

- **Start** &**Stop** - Allows you to enter start and stop frequency limit in S-Parameter measurement to set the sweep range.

- **Points** - Allows you to enter the number of points. Default value is 201.

- **Center/Span** - Allows you to enter the center and span value, another way to set the sweep range, other than using the Start and Stop value.

Defining Test Limit
Limit Type 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. 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.

There are 3 types of test limit available: **Absolute Limit | Average Limit | Multi Segment**. During Time Domain measurement, all 3 test limits are available.

However, during S-Parameter measurement, only Multi Segment is available.

Test Limit can be defined at Edit Limit Menu page.

Absolute Limit Test - In Absolute Limit test, the limit test zone is defined by Begin Ratio and End Ratio, and the position of markers 1 and 2. For example, where Marker 1 = 0s, Marker 2 = 100ns, Begin Ratio is 30% and End Ratio is 70%. Hence, 30ns to 70ns is the zone for the Limit Test. Lower and Upper limit can be specified by nominal and range (abs or %). If any measured point is out of limit line, then the test has failed.

Average Limit Test - The limit line is the same as Absolute Limit Test. If averaged measured result within limit line range is out of limit line, then the test has failed.

Multi Segment Limit Test - This test is the same as E5063A limit test. You can specify Max, Min, Off, Begin Stimulus, End Stimulus, Begin Response and End Response for one segment. You can specify up to 100 segments. If any measured point is out of limit line, then the test has failed.

Absolute Limit / Average Limit

Both absolute limit and average limit have similar selection option. To set the absolute limit, set the below variables:

- **Begin Stimulus Ratio** - Set the lower limit in percentage of the full span of Marker 1 to 2.

- **End Stimulus Ratio** - Set the upper limit in percentage of the full span of Marker 1 to 2.

- **Tolerance** - Can be set either with ratio in percentage or unit depending on format such as impedance in ohm.

Multi Segment

More than one frequency range (segments) can be set. To set the multi segment limits:

- **Add** - Add a new segment where you can add the upper and lower limit of the segment.

- **Delete** - Delete the selected (highlighted) segment.

- **Export** - Export the segment data (all of segments) in *.csv format.

- **Import** - Import segment data file (*.csv file).

Making Measurement for Confirming Result

In order to define the test limit, you can perform measurement and observe the waveform of your reference DUT. To learn more, see Measurement Setup and Edit Limit at Test Wizard Screen Navigation.

Making Trigger

1. Click **Edit Test** > **Test List** and select the test you would like to perform.

e5063a049
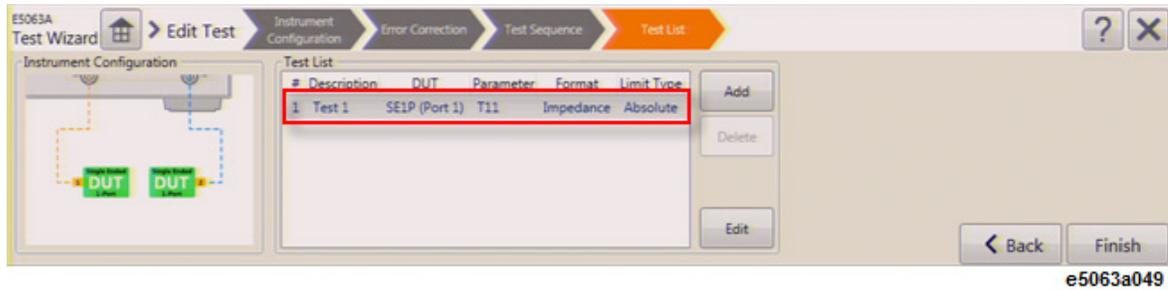
2. At **Edit Measurement** or **Edit Limit** page, click on the **Measure** button to perform the trigger.
3. You will notice changes on the display screen.

Using Auto Scale

1. Select the trace which you want to change the scale.

2. Click on the **Auto Scale** button on the tool bar or right-click in the graph plot area.

3. Select the desired scale. Refer to the table below for the details of each selection.
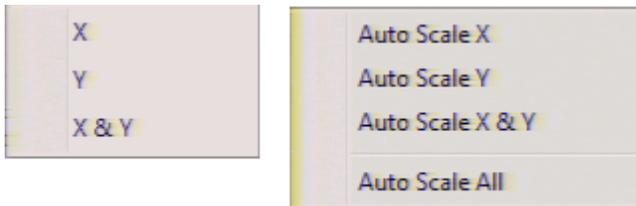
e5063a050

Scale	Details
Auto Scale X	Auto scaling X axis only, on the active trace
Auto Scale Y	Auto scaling Y axis only, on the active trace
Auto Scale X & Y	Auto scaling both X & Y axis, on the active trace
Auto Scale All	Auto scaling both X & Y axis, on all traces

Changing Scale Manually

Horizontal and vertical scale adjustment operation can be performed by using the virtual knob. Scale adjustment operation changes the display screen in real time. The scale setting is independent of each test.

e5063a051

Changing Horizontal Scale

1. Select the trace which you want to change the scale.
2. Go to **Edit Limit Menu**.
3. Using Virtual knob:
    a. Click the left knob under **Horizontal Scale**. The centre of the knob turns blue.
    b. Scroll up or down the wheel mouse button to change the scale.
4. Type in **Entry** box:

a. Click the box below the left knob under **Horizontal Scale**. An **Entry** dialog box appears.

b. Type the precise value.

5. Clicking Wave icons:

. Click the wave icons above the left knob under **Horizontal Scale**. Details are as table below:

Function Icon	Details
∿	Click to increase the scale in a 1-2-5 sequence. The width of the waveform compresses. The default value for basic time units is 1.0 ns/div.
∿	Click to decrease the scale in a 1-2-5 sequence. The width of the waveform expands.

Reference Position for Horizontal Axis

The reference point of horizontal axis can be selected from the center line or left edge of the displayed graticule.

Click the icon as shown in table below under **Horizontal Scale** to change the reference point:

370

Function Icon	Details
⇨	Reference point of the display graticule to the left edge. The position of left edge is not changed even if you change the scale.
⬌	Reference point of the display graticule to the center. The position of center is not changed even if you change the scale.

Changing Horizontal Position

1. Select the trace which you want to change the horizontal position.
2. Go to **Edit Limit Menu**.
3. Using Virtual knob:
   a. Click the right knob under **Horizontal Scale**. The centre of the knob turns blue.
   b. Scroll up or down the wheel mouse button to change the position.
4. Type in **Entry** box:
   a. Click the box below the right knob under **Horizontal Scale**. An **Entry** dialog box appears.
   b. Type the precise value.
5. Clicking Arrow icons:
   a. Click the arrow icons above the right knob under **Horizontal Scale**. Details are as table below:

Function Icon	Details
◀	Click to increase the delay from trigger; the waveform moves to the left of the display.
▶	Click to decrease the delay from trigger; the waveform moves to the right of the display.

Changing Vertical Scale

1. Select the trace which you want to change the scale.
2. Go to **Edit Limit Menu**.
3. Using Virtual knob:

    a. Click the left knob under **Vertical Scale**. The centre of the knob turns blue.

    b. Scroll up or down the wheel mouse button to change the scale.

4. Type in **Entry** box:

    a. Click the box below the left knob under **Vertical Scale**. An **Entry** dialog box appears.

    b. Type the precise value.

5. Clicking Wave icons:

a.   Click the wave icons above the left knob under **Vertical Scale**. Details are as table below:

Function Icon	Details
∿	Click to increase the scale in a 1-2-5 sequence; the waveform height compress.
∿	Click to decrease the scale in a 1-2-5 sequence; the waveform height will expand.

Changing Vertical Position

1. Select the trace which you want to change the vertical position.

2. Go to **Edit Limit Menu**.

3. Using Virtual knob:

    a. Click the right knob under **Vertical Scale**. The centre of the knob turns blue.

    b. Scroll up or down the wheel mouse button to change the position.

4. Type in **Entry** box:

    a. Click the box below the right knob under **Vertical Scale**. An **Entry** dialog box appears.

    b. Type the precise value.

5. Clicking the Arrow icons

    a. Click the arrow icons above the right knob under **Vertical Scale**. Details are as table below:

Function Icon	Details
▼	Click to increase the offset value; the waveform shift downward.

▲	Click to decrease the offset value; the waveform shift upward.

6. Drag and drop:

    a. Click and hold the y-axis of the graph plot.

    b. Drag up or down to the desired position and release the button.

Using Marker

Activating Marker

1. Click on the **Marker** button.

2. Select **3** from the list to activate Marker 3.

    `NOTE`   You can use the markers 3 to 9. The test wizard uses the markers 1 and 2 for DUT length.

    `NOTE`   Active marker has an arrow head pointing down. Inactive markers have an arrow head pointing up. Only markers 1 and 2 have vertical dotted lines. All marker values are displayed at the upper left corner of the graph view. There are a maximum of 9 markers that can be activated.
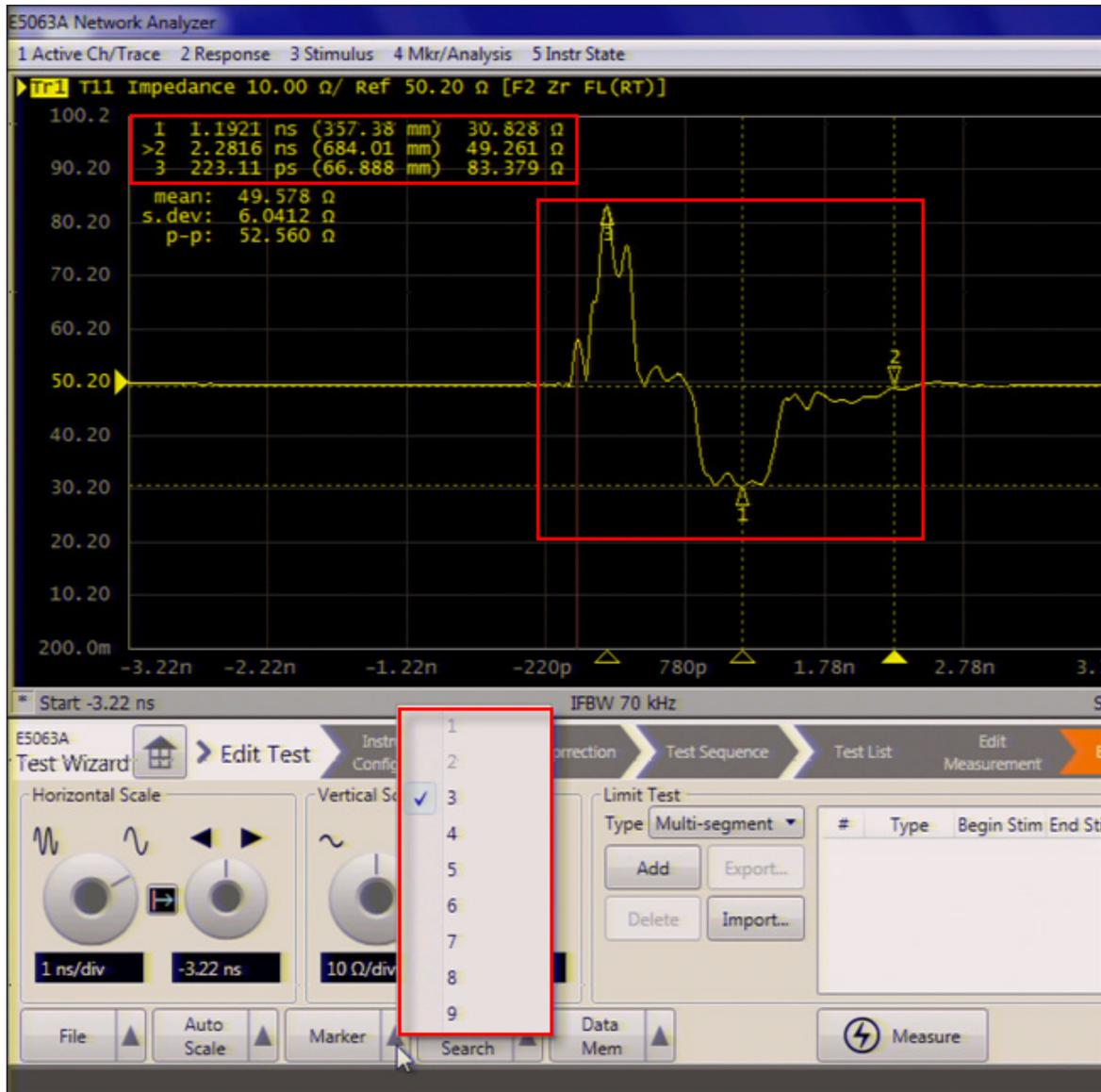
e5063052

Moving Marker

Click and hold on the marker, drag the marker to the desired point and release the mouse button.

Using Marker Search

Searching Max point on trace

1. Click on the desired trace.
2. Click on the **Marker Search** button.
3. Select **Max** from the drop-down menu.

Searching Min point on trace

1. Click on the desired trace.

2. Click on the **Marker Search** button.

3. Select **Min** from the drop-down menu.

> **NOTE**   The marker is constantly on tracking mode when Max or Min is selected. Even moving the marker with the mouse, the marker tracks the Max or Min of the trace and points to it.

Measuring Rise Time

1. Click on the desired trace.

2. Click on the **Marker Search** button.

3. Select **Rise Time (10-90%)** or **Rise Time (20-80%)**from the drop-down menu.

The data is displayed at the top left corner of the graph plot.
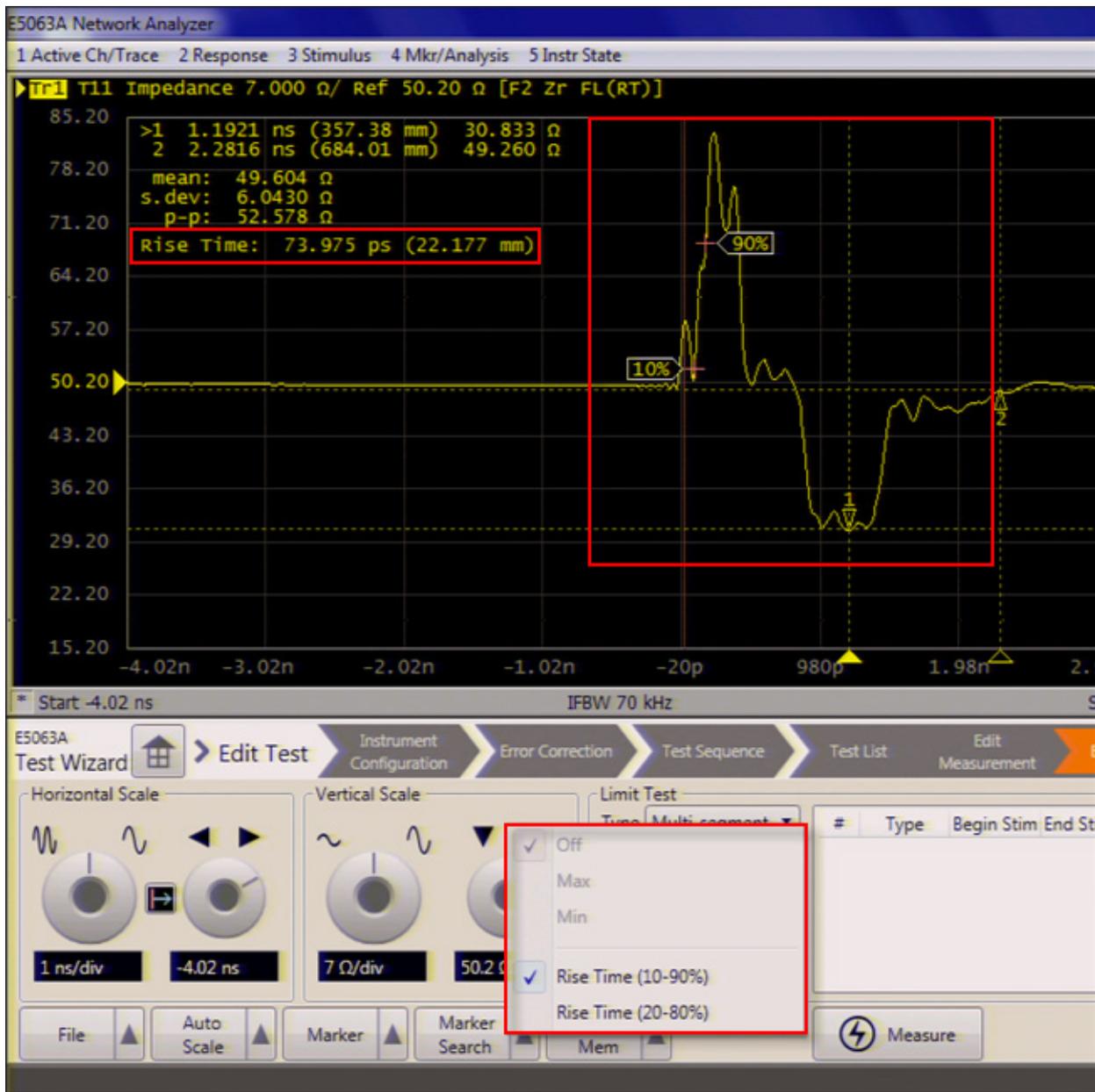
e5063a054

Using Data Memory

Storing Data Traces to Memory

1. Click on the **Data Mem** button.
2. Select **Data -> Mem** from the drop-down menu.

Display the Memory Traces

1. Click on the **Data Mem** button.
2. Select **Memory** from the drop-down menu.

Table below explains other functions in the Data Mem list:

Function	Details
Data -> Mem	Save Data traces to Memory
Off	Display off
Data	Display Data traces only
Memory	Display Memory traces only
Data & Memory	Display Data and Memory traces

Executing Test

Executing Test

- [Testing DUT](#)
- [Confirming Test Result](#)

[Other topics about Test Wizard](#)

Testing DUTs

- Instruction Window
- Save Test Result
- Auto Save Results/Trace/Touchstone

Other topics about Test Wizard

After pressing the Execute Test button, the test starts.

Instruction Window
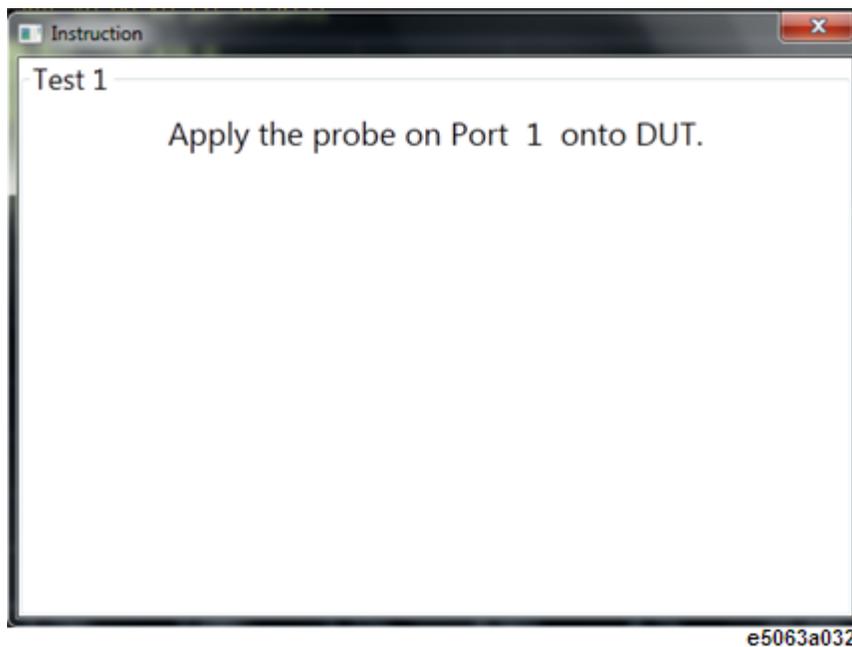
1. The instruction dialog box prompts to guide you on the DUT connection. The instruction on this box can be customized at Measurement Setup Menu. The dialog box is displayed at the beginning of each test.

2. Follow the instruction. Once done, click on **Measure** button to execute the test.

3. The test sequence options are as below:

   - **Auto Advance = On**: Next test is selected after a test is done.
   - **Auto Advance = Off**: You need to select the next test manually. Click the Finish button to move to the next DUT test.
   - **Auto Advance = On Pass**: Next test is selected after a test is done and it has passed.

- **Auto Scale = On**: Auto Scale X&Y is executed after a test is done.
- **Auto Save Results/Trace/Touchstone** - when turned ON, a folder is created under Output File Path, then result/trace/touchstone file is created(*). If the Test Description is null, Testxxx is used as the file name. If the Test Description has characters which cannot be used for the file name, the character will be replaced with "-".

> **NOTE** (*) The Test Description is used as the file name.

4. Upon measurement completion, the Test List will indicate the test result in brief, as shown below.
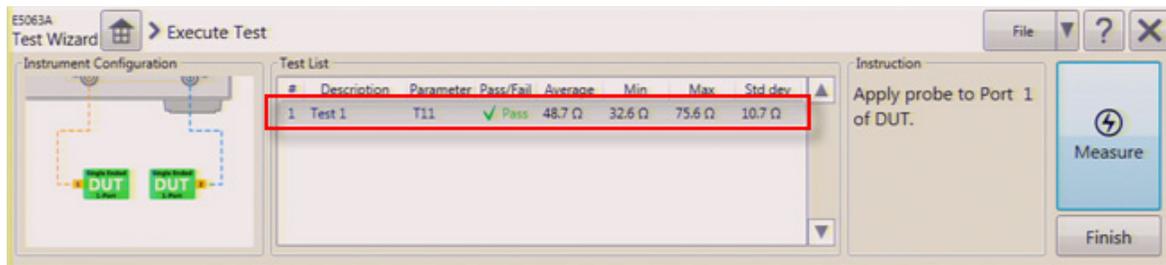
e5063055

5. Click the **Finish** button to save the test results.

Save Test Result

Upon test completion, the **Save Test Result** dialog box as shown below appears.

e5063a033
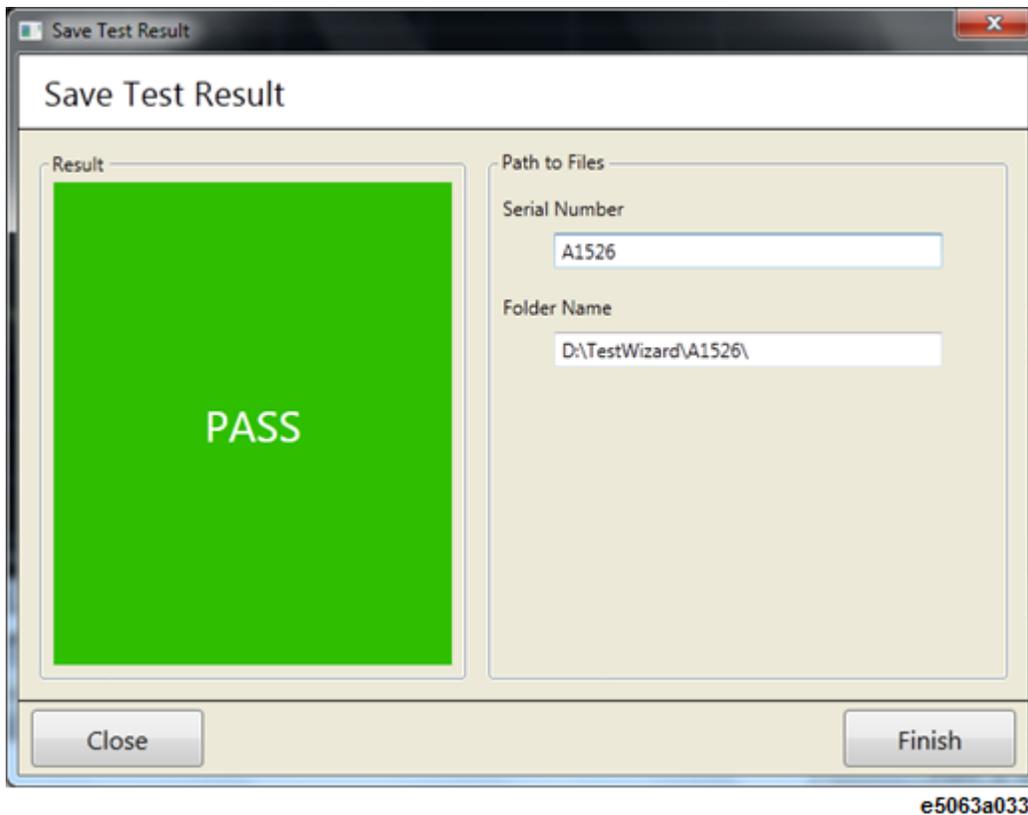
- When you input a Serial Number, the folder name is displayed. You cannot modify the file name on the dialog box. Serial number can be either numbers or letters or combination of both.

- The following items cannot be entered as Serial Number: Null, "AutoSave" or serial number which are already used/entered.

- When an unacceptable serial number is entered, warning is displayed and **Finish** button is grayed out.

Save Test Result dialog box is displayed under the following conditions:

- At the Execute Test menu, click the Finish button OR

- When all the following conditions are met:

  - Auto Save Result = On

  - Auto Advance = On OR Auto Advance = On Pass, when the test has passed

  - All the tests are measured

When there is test that is not executed, the Save Test Result dialog cannot be displayed.

Confirming Test Result

If you have set the File Output preference during test sequence configuration, the test results are saved automatically upon test completion. Alternatively, you can choose to save the test results after the test completion as below:

- Trace Data
- Touchstone
- Image

e5063a059
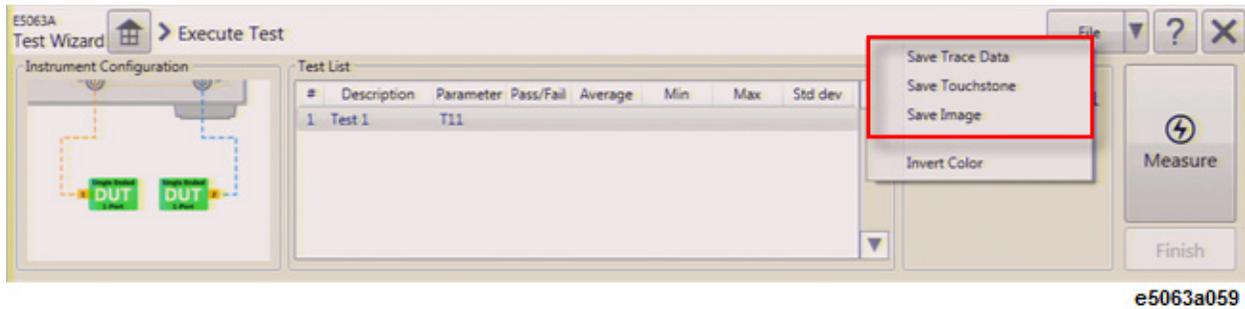
When Auto Save Trace/Touchstone is turned on, the output folder is automatically created under Output File Path and the trace file/touch stone is created and saved here as Result file.

Full Path = **[Output File Path]/[Serial No.]/[Test Description].[extension]**

When the test description is null, TestXXX is used for file name. If the Test Description has characteristics which cannot be used for file name, the character is replaced with "_".

Trace Data

When you select to save the result as Trace Data file, the result is saved in extension *.csv, as shown in the below example.

e5063a056

Touchstone

When you select to save the result as Touchstone file, the result is saved in one of the following extensions:

- *.s1p (for DUT = Single Ended 1 Port, SE1P)
- *.s2p (for DUT = Single Ended 2 Port, SE2P or Differential 1 Port, DIFF1P)

Below is an example of touchstone file.

e5063a057
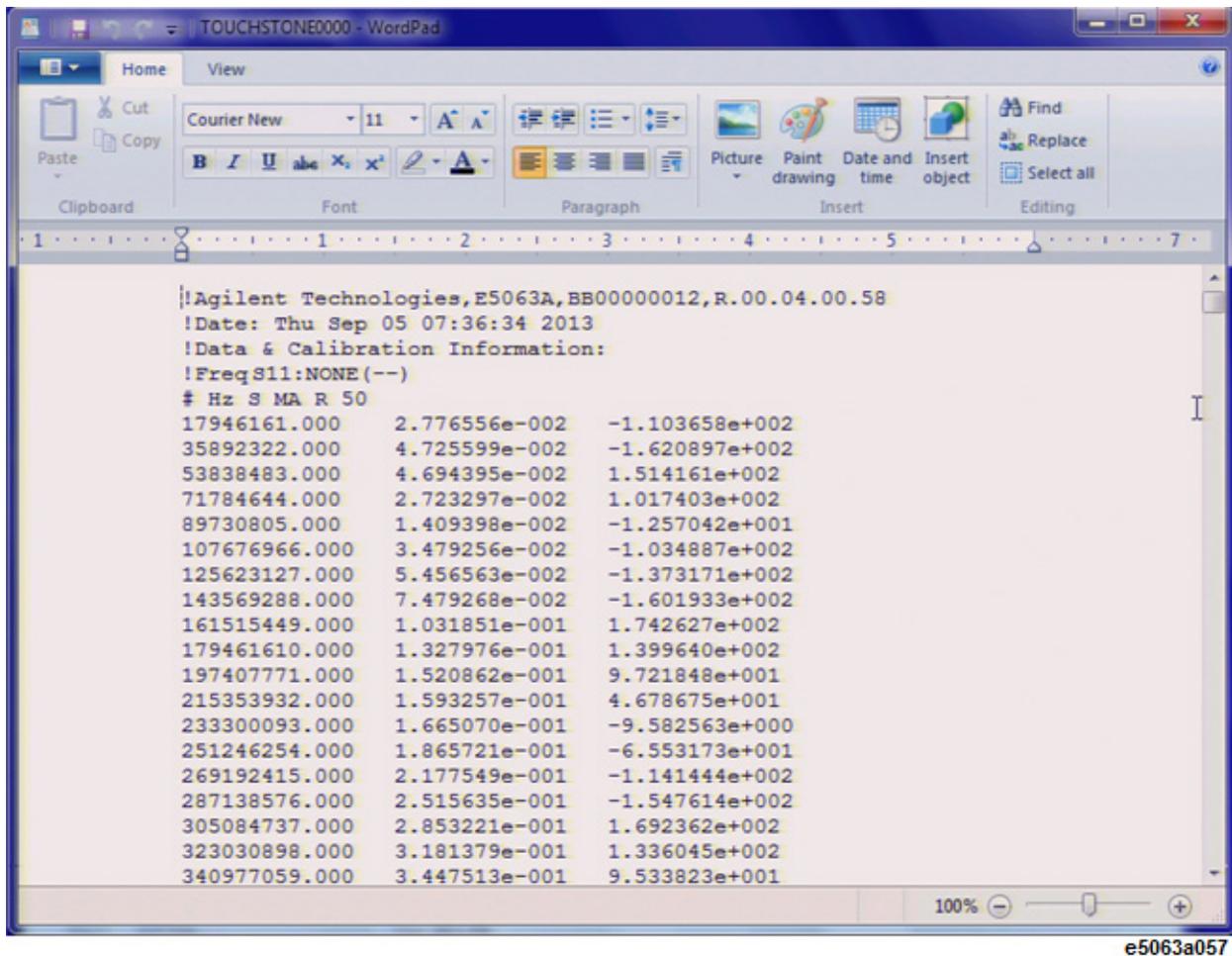
Image

When you select to save the result as Image, the screen shot of the result is saved with extension *.png. Below is an example of test result image.

e5063a060
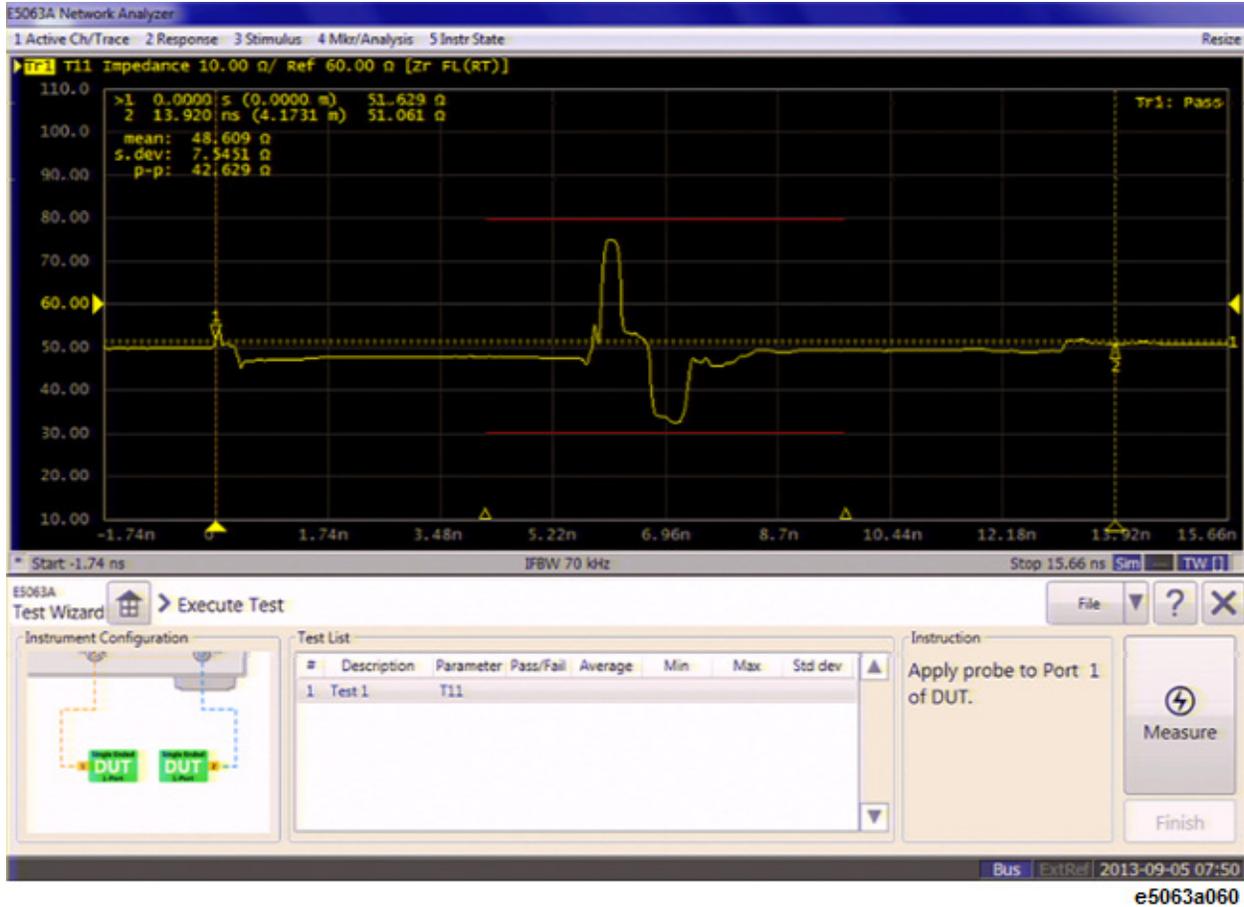
Result File

When you select <u>Save Test Result</u> and save the test result, Summary.csv file is created in the output folder, as shown below.

e5063a058
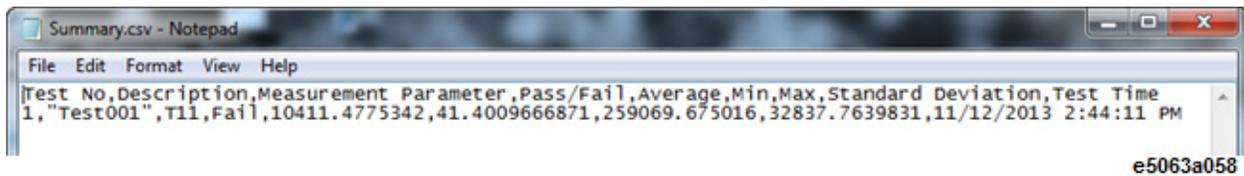
Setting Control Functions

Setting Control Functions

Remote Control
- [Setting the GPIB](#)
- [Remote Control Using HTTP (Web Browser)](#)

Display
- [Turning off the Date/Time Display](#)
- [Turning off the LCD Screen Backlight](#)
- [Calibration of the Touch Screen](#)

Others
- [Checking the Product Information](#)
- [Source ON only during measurement (Initial Source Port Control Function)](#)
- [Wait Time at Power Level Change](#)
- [Locking the Front Keys, Keyboard, and/or Mouse (Touch Screen)](#)
- [Setting USB Switch](#)
- [Setting the Beeper (Built-in Speaker)](#)
- [Setting the preset function](#)
- [Exit/Restart E5063A Measurement Application](#)
- [Overload Detection and Power Trip Function](#)

Setting the GPIB

Other topics about Setting Control Functions

This section describes how to set the interface necessary to use the GPIB (General Purpose Interface Bus) of the E5063A.

Setting talker/listener GPIB address of E5063A

When controlling the E5063A using GPIB commands from the external controller connected to the GPIB connector, you need to set the talker/listener GPIB address of the E5063A.

Follow these steps to make this setting:

1. Press **System** > **Misc Setup** > **GPIB Setup**> **Talker/Listener Address**.
2. Enter the address using the ENTRY block keys on the front panel.

Setting system controller (USB/GPIB interface)

When controlling an external device from the E5063A, connect the USB port of the E5063A and the GPIB port of the external device through the USB/GPIB interface.

> **NOTE**   Do not connect two or more USB/GPIB interfaces.

Follow these steps to set the USB/GPIB interface:

1. Connect the USB port of the E5063A to the USB/GPIB interface.
2. The driver installation and connection is performed automatically.

Changing the setting of the USB/GPIB interface

If you need to check/change the setting of the USB/GPIB interface after connecting the USB/GPIB interface, follow these steps:

1. Press **System** > **Misc Setup** > **GPIB Setup** > **System Controller Configuration**.
2. The Agilent Connection Expert appears. (You can also execute Agilent Connection Expert from Task bar or Start menu in Windows.)
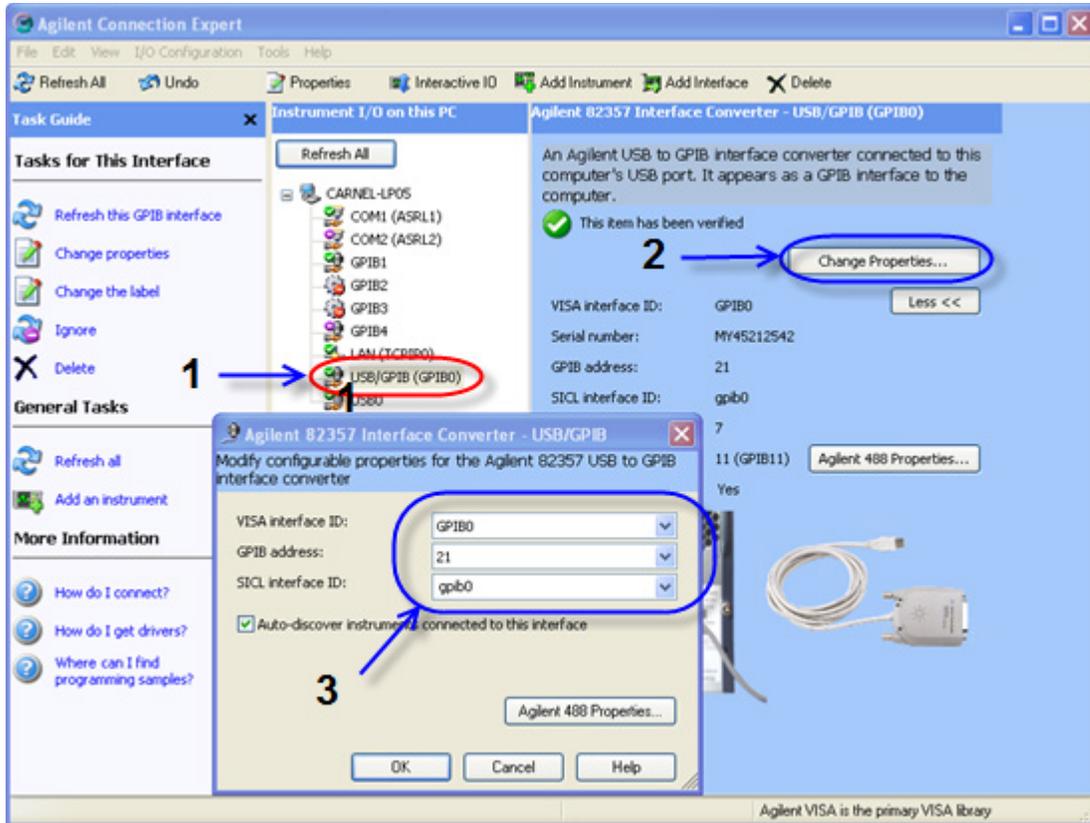
e5071c172

Need update

3. Select USB/GPIB (GPIBx)

4. Click **Change Propetries...**.

5. Change the setting of USB/GPIB interface.

Remote Control Using HTTP (Web Browser)

- Enabling Web Server
- Browser Web Control

Other topics about Setting Control Functions

You can access the web page installed in the E5063A by using the hypertext transfer protocol (http) and the E5063A's IP address from the external PC's web browser. This function is called web-enabled analyzer. Through the built-in web page, you can control the E5063A remotely and display the measurement screen on external PCs.

The following browsers are recommended:

- Internet Explorer 8.0 and later

Enabling Web Server

Network Configuration

To use web server, you have to configure the E5063A's network correctly. For detailed information on configuration and notes, see Configuring the Network.

Enabling Web Server

Enable the web server for the E5063A so that it may allow access from an external PC. Follow these steps:

1. Press **System** > **Misc Setup** > **Network Setup**.
2. Click **Web Server** to turn it ON.

Access from an external PC

1. Execute web browser on your PC.
2. Check IP address of the E5063A.
3. Enter IP address of the E5063A in the address bar and press Enter. The following screen appears:

e5063a087

To change LAN (TCP/IP) confriagtion of you ENA.

1. Click **View & Modify Configuration** (**1** in the Figure below). The following screen appears:
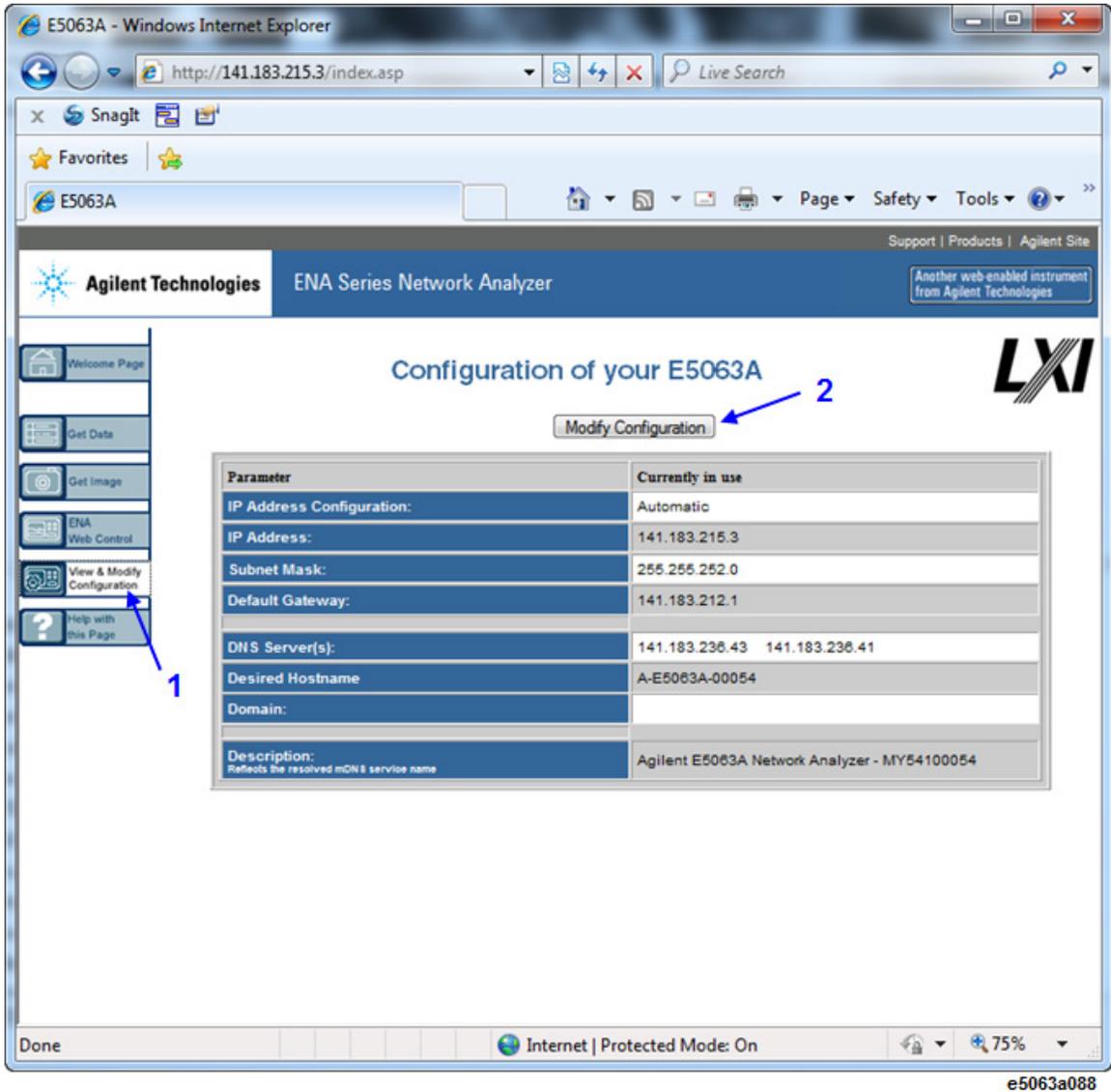
e5063a088

2. Click **Modify Configuration** (**2** in the Figure above). Modifying this setup affects the Windows Internet Protocol (TCP/IP) property. The following screen appears:

e5063a089

3. Enter the password (Default: "agilent") in the Password field (**1** in the figure above) and click **OK** (**2** in the figure above). The following screen appears:
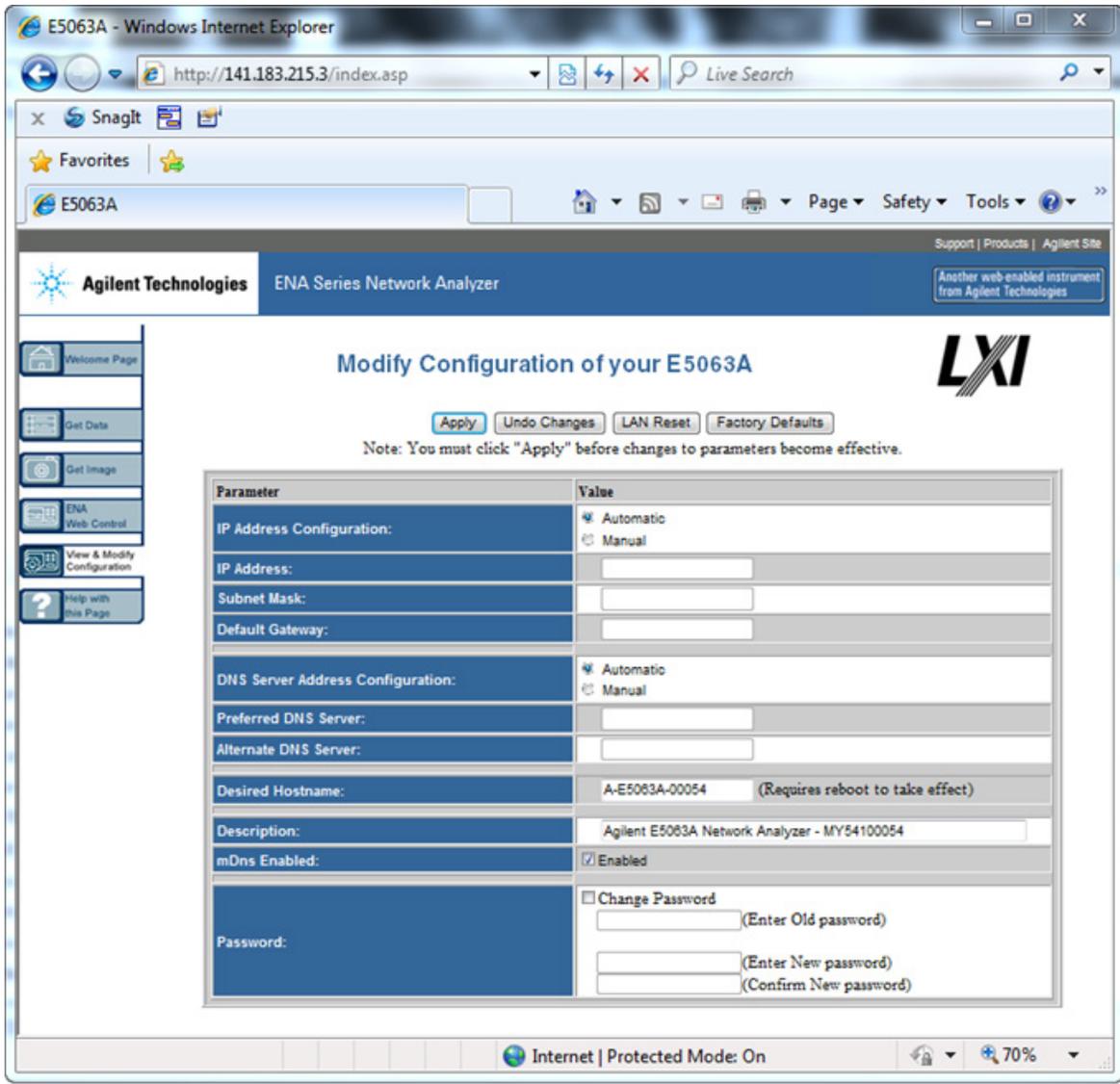
e5063a090

Browser Web Control

Browser Web Control function allows you to control your E5063A from web browser. This function is executed by the VNC server.

- The external PC must have the Java Runtime Environment installed otherwise the Browser Web Control function might not work properly. To install Java Runtime Environment, see http://www.java.com.

The following is a description of how to start the VNC server configuration. Visit the web site at http://www.realvnc.com for information on the password setting procedure and VNC server.

1. Press **System** > **Misc Setup** > **Network Setup**.

2. Click **VNC Server Configuration** to start the VNC Server Properties.

3. To restrict external access, set a password for the VNC server configuration. The default password at factory shipment is blank.

4. Redistribution of VNC is licensed under the General Public License version 2 copyright. Source code of VNC and a copy of the GPLv2 may be found in the directory of E:\OpenSource\vnc.

Turning on/off the Date/Time Display

The date/time display in the instrument status bar can be switched ON/OFF using the following procedure.

1. Press **System** > **Misc Setup** > **Clock Setup**.

2. Click **Show Clock** to toggle the date/time display ON/OFF.

Refer to Change Date/Time Settings

Other topics about Setting Control Functions

Turning off the LCD Screen Backlight

You can switch OFF the backlight (illumination) of the LCD screen of the E5063A. This extends the life of the backlight when it is used continuously over a long period.

Turning OFF the LCD Screen Backlight

1. Press **System** > **Backlight** to switch the backlight ON/OFF.

2. Switching OFF the backlight causes the indications on the LCD screen to be almost invisible.

3. The backlight that has been switched OFF can be turned ON again by pressing any key from the front panel.

Other topics about Setting Control Functions

Calibration of the Touch Screen

When you have executed system recovery on the E5063A, you have to calibrate the touch screen. Follow the procedure described below to calibrate the touch screen.

1. Press **System** > **Service Menu** > **Test Menu** > **Adjust Touch Screen**.

2. The touch panel device properties dialog appears. Press **Calibration**.

3. The touch screen calibration screen appears.

4. Touch the x mark on the upper left with your finger. The mark x also appears on the lower left, upper right, and lower right. Touch the x marks in that order with your finger.

5. Touching all the four locations described above with your finger automatically concludes the touch screen calibration.

6. With no operation on the touch screen calibration screen for a preset time, it automatically closes and the previous measurement screen reappears.

Other topics about Setting Control Functions

Checking the product information

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

Other topics about Setting Control Functions

Overview

Items	Description	When the number changes for your unit after shipment
Serial Number	Product Serial Number. The first 5 letters expresses the prefix.	Immutable number
Storage Revision	Storage Revision. The first two letters express the prefix. The last three letters express the revision number at the factory shipment.	The prefix is immutable number. The last three letters may change by a storage replacement at repair.
Generation ID	This expresses the product generation. The format is Gxx. This number is reserved for future usage. This is managed by the [license key](#).	Immutable number
Firmware Revision	Firmware revision. The formt is A.xx.xx.	The number is changed at firmware update.
Option	Installed [Options](#) Number. This is managed by the [license key](#).	The upgrade kit adds the option.
OS ID	Windows License Description	Immutable ID
IP Address	IP adress when the unit is connected with LAN	It is depending on LAN configuration.
MAC Address	MAC Address of LAN interface connection.	Immutable address unless the CPU board is replaced at repair.

USB ID	USB ID for VISA connection	Immutable ID

Procedure

Follow the procedure below to get the product information.

1. Press **System** > **Firmware Revision**.

2. The Firmware Revision dialog is displayed.

3. Pressing **PC Info** shows the PC hardware information.

Activating Software Option

- [Activating Option](#)
- [Backing Up License Key File](#)

Activating Option

The [software options](#) can be purchased separately to enhance the E5063A measurement functionality. When you purchase the software option upgrade kit, Agilent provides the software entitlement certificate.

Procedure

1. Get the license number at http://www.agilent.com/find/softwarelicense.
2. Press **System** > **Service Menu** > **Enable Options**, then select option which you want to activate.
3. Type the relevant 12 character long license key sent by Agilent in the Key Code entry dialog box, then click Enter.
4. Check the [installed option](#).

Backing Up License Key File

The following license key are kept in a text file (.lic) located at Factory(E:)\LICENSE\. The E5063A refers to the license codes in all of .lic files under the directory.

- Software option (Optional: Example, 011)
- Frequency option (Mandatory: Example, 245, 285, 2H5)
- Generation code (Mandatory: Example, G01)

If there is no license code of either Frequency option or Generation code, an error occurs and the E5063A measurement application works as simulator only.

When you change the storage, copy the .lic files as back up. You can re-create the license number at http://www.agilent.com/find/softwarelicense with your software entitlement certificate if the option is added with the upgrade kit.

Source ON only during measurement (Initial Source Control Function)

The E5063A does not output the RF signal source while the measurement is hold. (when Initial Source Control is ON, default setting). This function allows you to output the signal source even while the measurement status is in hold.

1. Press **System** > **Service Menu** > **Init Src Ctrl** to turn OFF.

Other topics about Setting Control Functions

Wait Time at Power Level Change

The E5063A have a wait time in measurement when the power level is changed in the following conditions.

- Different power level setting in channel
- Port couple off, different power level setting for each port
- In List Power of Segment Power Sweep, different power level setting for each segment

This wait time function can be turned off.

- **System** > **Service Menu** > **Pow. Chg. Wait**

The time can be adjusted. At default, it is 50 ms.

- **System** > **Service Menu** > **Pow. Chg. Wait Time**

These setting are saved/recalled by state files.

Other topics about Setting Control Functions

Locking the Front Keys, Keyboard, and/or Mouse (Touch Screen)

You can lock (disable) the front keys, keyboard, and/or mouse (touch screen). This feature prevents erroneous operation caused by inadvertently touching any of these devices.

Locking the Front Keys, Keyboard, and/or Mouse

1. Press **System** > **Misc Setup** > **Key Lock**.
2. Click the corresponding key to switch the lock ON/OFF.

Softkey	Function
Front Panel & Keyboard Lock	Switches the lock of the front panel keys and keyboard ON/OFF.
Touch Screen & Mouse Lock	Switches the lock ofr the touch screen and mouse ON/OFF.

- You cannot use a locked device to unlock that same device. To unlock the front panel keys, keyboard, touch screen and mouse that have been locked, press the Standby switch to turn OFF the power supply and then turn it ON again. When setting at power-on, the front panel keys, keyboard, touch screen and mouse are all in unlocked condition.

Other topics about Setting Control Functions

Setting USB Switch

- [Introduction](#)
- [Setting USB Switch](#)

Other topics about Setting Control Functions

Introduction

Any network analyzer with USB supply can be used with USB switch. With U1810B USB coaxial switch, the E5063A can be used for multi-port testing. At any one time, maximum of 6 USB coaxial switches can be connected to the E5063A. However, in practice, up to two switches are connected. By connecting the single-ended or differential probe to the USB switch and the DUT on the other end, depending on the desired measurement, the test can now be performed.

Setting USB Switch

1. Connect the USB switch(es) to the ports of E5063A.
2. Connect the USB cable, one end to E5063A and another to the USB switch. If you are using Agilent U1810B, its now ready to be used, as it supports the standard plug-and-play functionality of typical USB devices and does not require additional power supply.
3. Press **System** > **USB Switch Setup**.
4. Once the USB switch is connected, the model name of the USB switch is shown beneath the USB Switch number. If no USB switch is connected, None appears beneath the USB Switch number.
5. To toggle between Port 1 and 2,
   - Press the switch toggle location close to the USB port of the USB switch.

- **System** > **USB Switch Setup** > **Path** > **1|2.**

6. To disable the switch toggle, press **System** > **USB Switch Setup** > **Lock Local Control** > **ON.**

Setting the Beeper (Built-in Speaker)

- [Setting the Operation Complete Beeper](#)
- [Setting the Warning Beeper](#)

Other topics about Setting Control Functions

The E5063A has a built-in speaker that emits a beep tone. The beeper allows you to make two types of settings.

Type	Function
Operation complete beeper	Emits a beep tone to inform the user that operations have completed. • When calibration data measurements are done • When data storage has completed
Warning beeper	Emits a beep tone to prompt the user to use caution. • When an instrument error occurs (An error message appears at the same time.) • When a limit test fails

The operations complete beeper emits slightly longer than the warning beeper.

Setting the Operation Complete Beeper

1. Press **System** > **Misc Setup** > **Beeper** > **Beep Complete** to switch the operation complete beeper ON/OFF.
2. Clicking **Test Beep Complete** allows you to hear and check the beep tone of the operation complete beeper.

Setting the Warning Beeper

1. Press **System** > **Misc Setup** > **Beeper** > **Beep Warning** to switch the warning beeper ON/OFF.
2. Clicking **Test Beep Warning** allows you to hear and check the beep tone of the warning beeper.

Setting the preset function

- [Showing/hiding the confirmation buttons when presetting](#)
- [Setting the user preset function](#)
- [Saving a user-preset instrument state](#)

Other topics about Setting Control Functions

Showing/hiding the confirmation buttons when presetting

The preset function can be executed without displaying the **OK** and **Cancel** softkey buttons when pressing the preset button of the E5063A.

1. Press **System** > **Misc Setup** > **Preset Setup**.
2. **Confirm** to toggle ON (show)/OFF (hide) the confirmation buttons.

Setting the user preset function

You can save the instrument state of the E5063A into a file in the mass storage, and then recall it with the preset function to reproduce that state.

If no user preset instrument state is stored, you cannot set the user preset function.

1. Press **System** > **Misc Setup** > **Preset Setup** > **State**.
2. Use one of the following keys for the desired setting.

Softkey	Function
Factory	Specifies the normal preset function.
User	Specifies the [user-preset function](#).
Cancel	Returns to the softkey display in one upper level.

Saving a user-preset instrument state

To execute the user-preset function, you must have a preset setting file that has been saved. Follow these steps to save a preset instrument state of the E5063A.

1. Press **Save/Recall** > **Save State** > **User Pres**.

Exit/Restart E5063A Measurement Application

Pressing **System** > **Service Menu** > **Exit** shuts down the firmware of E5063A.

Pressing **System** > **Service Menu** > **Restart Firmware** restarts the firmware of the E5063A.

Other topics about Setting Control Functions

Overload Detection and Power Trip Function

An overload detection and Power Trip function is a safety feature to protect your E5063A from over-input. The overload occurs when RF input is over 20 dBm. When an overload is detected, the measurement is aborted and the RF source signal is turned OFF, then the "Ovld Protection" is displayed on the instrument status bar.

Other topics about Setting Control Functions

Using Windows

Using Windows

- [Windows Consideration](#)
- [Change Date/Time Settings](#)
- [User Account and Password](#)
- [On-Screen Keyboard](#)
- [Configuring Network](#)
- [Windows Firewall](#)
- [Enabling/Disabling USB Storage](#)
- [Connecting External Accessories](#)

Windows Consideration

- [Storage](#)
- [Using USB](#)
- [Plug & Play Stability and Security](#)
- [LAN Connections](#)
- [Single and Double Click option](#)
- [Printing](#)

Other topics about Using Windows

Storage

The E5063A storage contains several partitions. The following table explains the different partitions of the E5063A.

Drive	Description
KAxxx (C:)	The system drive is replaced with the original image when system recovery is executed. Do not change any files and folders in this.
USER (D:)	User can keep their files, in this drive. This drive is not replaced even when system recovery is executed.
Factory (E:)	This drive contains license information and back up data for syscal. The drive is not replaced even when system recovery is executed. Do not change any files in this.

Using USB

The E5063A has six USB ports for connecting devices: two in the front panel and four on the rear panel. The main advantages of USB are instant connect and disconnect, and faster data transfer speed.

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

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

NOTE   Some USB memory may not operate on the E5063A. In that case, try the other kind of USB memory. There is some USB

memories which do not work on the front panel USB port, but work on the rear panel USB port.

NOTE   Agilent Technologies shall not be responsible for, nor assume any liability for data loss in your USB memory device after using it with the E5063A.

Plug & Play Stability and Security

Plug & Play capabilities provide both a stable and secured operating environment. You may also notice that it greatly reduces the number of required reboots.

LAN Connections

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

Single and Double Click option

By default, Windows allows a double-click method of launching icons. To revert to single-clicking, see the Windows help.

Printing

Adding a printer should be done as Windows operation. See Connecting Printer.

Change Date/Time Settings

The E5063A has the built-in clock that keeps track of the date and time. This clock is used for the following functions.

- To display the current date and time in the instrument status bar at the lower part of the screen
- To write date and time information when saving internal data

Below are the options to change the **Date & Time** of E5063A:

- Double-click the clock at the bottom right of the taskbar. Click on the **Change date and time settings**. The **Date and Time** dialog box appears. Click on the **Change date and time settings** button. Perform the desired changes.

- **System** > **Misc Setup** > **Clock Setup** > **Set Date and Time….**Similar to the earlier option, the **Date and Time** dialog box appears. Click on the **Change date and time** button. Perform the desired changes.
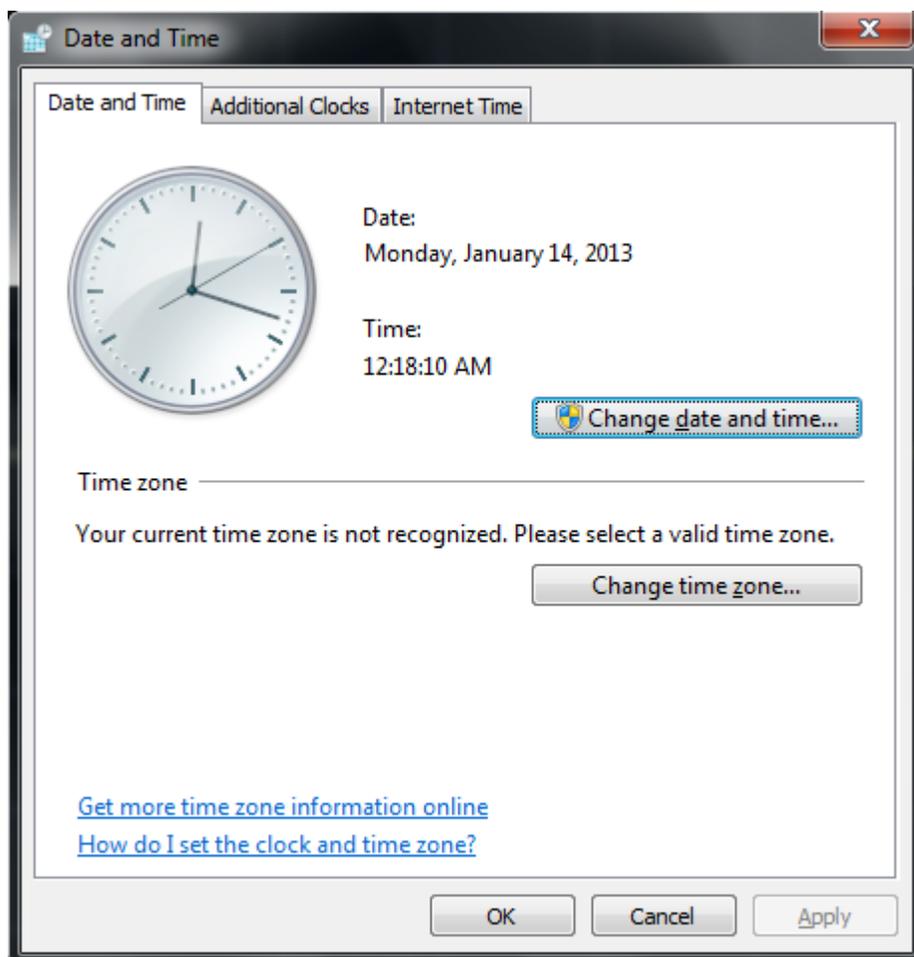
NOTE   The administrator password is required to change the time and date setting when login as standard user.

Other topics about Using Windows

User Account and Password

- [Account Settings](#)
- [To Change Password](#)
- [To Add and Remove User](#)

Other topics about Using Windows

Account Settings

The E5063A measurement application can be executed with both administrator and standard user type of account with windows.

The below account settings are available:

User Name	Password	Type	Description
Instrument	measure4u	Administrator	Auto Log On is activated by default.
Administrator	agilent4u	Built-in Administrator	For user maintenance purpose.
To be determined by user	To be determined by user	Administrator or Standard user	Not a default account setting. However, can be added by user.
Agilentonly	Not disclosed to user	Administrator	This account is used by Agilent Service in case of repair/maintenance. NOTE Do not remove this account.

NOTE Refer to [Add and Remove User](#) to learn on how to add and remove standard user and administrator.

Standard User Limitation

The standard user account has some limitation in access, especially in performing the below tasks. These are only allowed with administrator account:

- [Change date and time settings](#)
- [Firmware update](#)

- [Enable and disable USB storage](#)
- [Enable Web Server](#)
- Saving file under specified directories (such as the root directory) determined by Windows
- Change User Account Control settings

User Account Control

With administrator account, you are allowed to modify the User Account Control settings. To access the User Account Control setting, open Control Panel, click on **User Accounts** > **Change User Account Control settings**. The default setting is *Notify me only when programs try to make changes to my computer*. With this selection, you are notified each time a program tries to make any changes to your computer. Other available settings are not guaranteed in the E5063A operation.

With a standard user account, default setting (*Notify me only when programs try to make changes to my computer*) should be selected before you leave from an administrator account.

To Change Password

You are allowed to change an account's password.

Change Password

1. Open **User Accounts** in Control Panel.

2. Click **Add or remove user accounts**.

3. Click on the account you would like to change password from the **Choose the account you would like to change**.

4. Click **Change the password.**

5. Enter **new password**, **confirm new password** and **password hint**.

6. Click **Change Password** button.

To Add and Remove User

Adding new user and removing existing user can be done via Control Panel.

Add New User

1. Open **User Accounts** in Control Panel.

2. Click **Add or remove user accounts**.

3. Click **Create a new account**.

4. Enter the new account name.

5. Select either **Standard user** or **Administrator**.

6. Click **Create Account** button.

NOTE   The administrator authority is necessary for some of function in E5063A. Select computer administrator when you make the user account.

Remove Existing User

1. Open **User Accounts** in Control Panel.

2. Click **Add or remove user accounts**.

3. Click on the account you would like to remove from the **Choose the account you would like to change**.

4. Click **Delete the account**.

5. Window message confirming if you would like to keep or delete the account's files prompts.

6. Click either **Delete Files** or **Keep Files** button.

7. Next windows message prompts seeking confirmation if you are sure of deleting the selected account.

8. Click on **Delete Account** button.

On-Screen Keyboard

- [On-Screen Keyboard overview](#)
- [To open Windows On-Screen Keyboard](#)

Other topics about Using Windows

On-Screen Keyboard overview

On-Screen Keyboard is a utility provided by Windows that displays a virtual keyboard on the screen. It allows the users of the E5063A to input characters without the need of a keyboard.

- The E5063A does not requires an external keyboard for its operation. Users can input characters using an on-screen keyboard in-built with the E5063A firmware.

On-screen keyboard in-built with the E5063A firmware

To open Windows On-Screen Keyboard

You can use Windows native on-screen keyboard as well.

To open On-Screen Keyboard: click **Start** > **Accessibility On-Screen Keyboard**.

On-screen keyboard provided by windows OS

e5063a098

Configuring Network

NOTE　　　　When you use the E5063A by connecting it to your LAN, consult your network administrator and make the setting of the LAN correctly.

This section describes how to set the following basic items necessary to connect the E5063A to the LAN (Local Area Network).

- Enabling/Disabling network
- Check/Reset IP Address
- Setting IP address
- Checking computer name

If you need detail network settings, consult your network administrator and perform operation in the same way as the Windows PC.

Other topics about Using Windows

Enabling/Disabling Network

You can enable/disable the network connection function of the E5063A.

To enable/disable the network connection function

1. Use the LAN cable to connect the E5063A to the LAN.

2. Press **System** > **Misc Setup** > **Network Setup** > **Network Configuration** to open **Network Connections** window.

3. Perform either of the following:

- When switching from disable to enable:

  Double-click the Local Area Connection icon in the **Network Connections** window to enable the network connection function.

- When switching from enable to disable:

  Double-click the Local Area Connection icon in the **Network Connections** window. The **Local Area Connection Status** screen appears. Click the **Disable** button to disable the network connection function.

4. Close **Network Connections** window.

Check/Reset IP Address

The IP address of the E5063A can be checked/reset by **System** > **Misc Setup** > **Network Setup** > **LAN Dialog**.

Setting IP address

Follow these steps to set the IP address:

1. Press **System** > **Misc Setup** > **Network Setup** > **Network Configuration**.

2. Double-click the Local Area Connection icon in the **Network Connections** window. The **Local Area Connection Status** screen appears.

3. Click **Properties**. The **Local Area Connection Properties** screen appears.

4. Select (highlight) **Internet Protocol (TCP/IP),** and then click **Properties**.

5. The **Internet Protocol (TCP/IP) Properties** appears. Click (select) **Use the following IP address** and then enter the **IP address**, **subnet mask** and **gateway address**.

6. If the IP address can be obtained automatically (if the DHCP server can be used), click (select) **Obtain an IP address automatically**.

7. In **Internet Protocol (TCP/IP) Properties**, click **OK**.

8. In **Local Area Connection Properties**, click **OK**.

9. In **Local Area Connection Status**, click **Close**.

10. Close **Network Connections** window.

Checking Computer Name

Follow these steps to check the computer name:

1. Press **System** > **Misc Setup** > **Network Setup** > **Network Identification**.

2. See the desired computer name in Computer description in **Computer Name** tab.

Windows Firewall

- [To Turn Windows Firewall ON or OFF](#)

To Turn Windows Firewall ON or OFF

1. Open Windows Firewall. To open Windows Firewall, open Control Panel, and then double-click **Windows Firewall**.

2. From the right column, select either **Change notification settings** or **Turn Windows Firewall on or off** option.
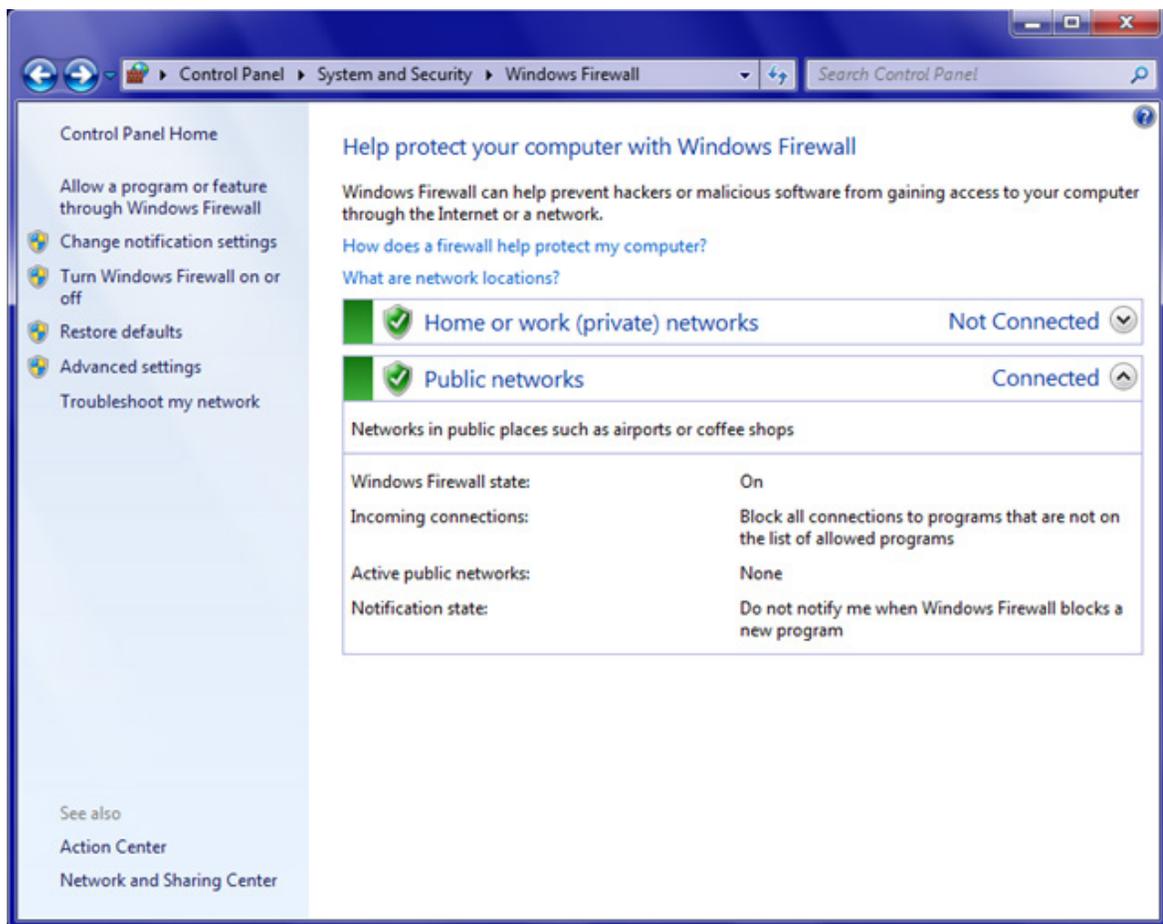
3. You may select one of the below options:

   - **Turn on Windows Firewall** - this is the default and recommended option.

     - **Block all incoming connections, including those in the list of allowed programs** - this is not a preferred option.

- ▪ **Notify me when Windows Firewall blocks new program** - this option blocks new program and notifies you, so that you can choose to block or unblock the program after reviewing it.

- **Turn off Windows Firewall (not recommended)** - this is not a recommended option. Turning OFF Windows Firewall might make your computer (and your network, if you have one) more vulnerable to damage from viruses.
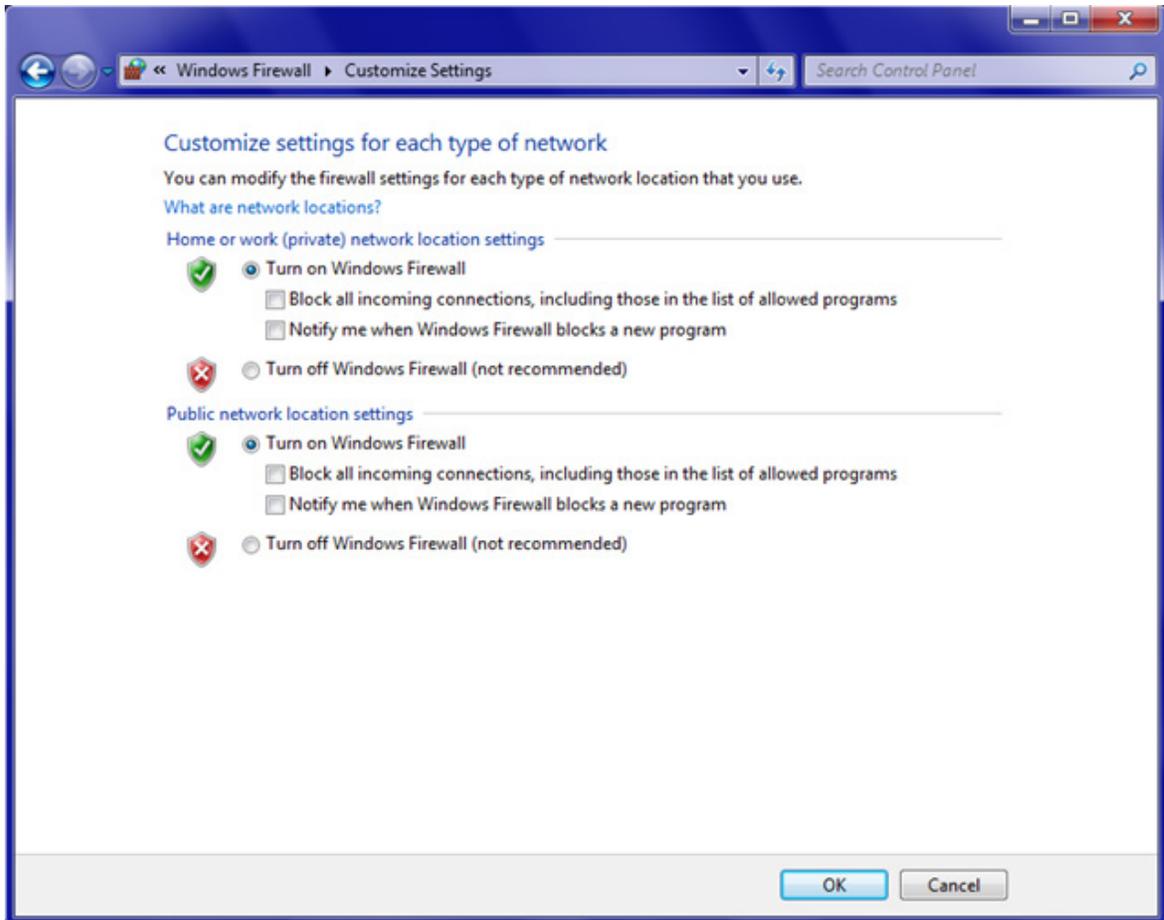
4. To restore to default settings, select **Restore defaults** from the fight field and click on the **Restore defaults** button.

Enabling/Disabling USB Storage

- [Local Group Policy Editor](#)
- [Enabling & Disabling USB Storage](#)

Local Group Policy Editor

Group Policy Editor is a part of Windows operating system that allows you to control your machine. One of its feature is the ability to turn ON and OFF access to USB storage.

Group Policy Editor is a Microsoft Management Console snap-in that provides a single user interface through which all the Computer Configuration and User Configuration settings of Local Group Policy objects can be managed.

- **Computer Configuration**
  Administrators can use Computer Configuration to set policies that are applied to computer, regardless of who logs on to the computers. Computer Configuration typically contains sub-items for software settings, Windows settings, and administrative templates.

- **User Configuration**
  Administrators can use User Configuration to set policies that apply to users, regardless of which computer they log on to. User Configuration typically contains sub-items for software settings, Windows settings, and administrative templates.

To launch the Local Group Policy Editor:

1. Click on the **Start** menu.
2. In the **Search programs and files** entry, type GPEDIT.MSC.
3. The below screen appears, with as per below
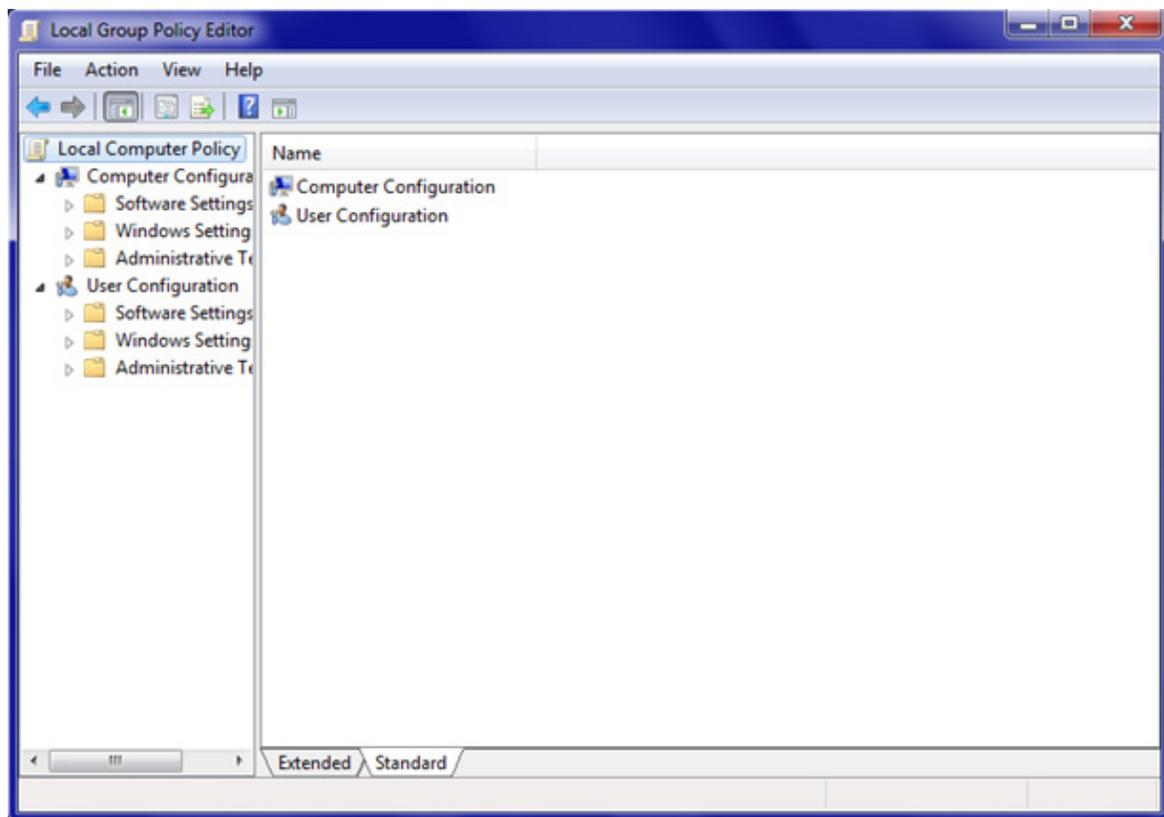
Enabling & Disabling USB Storage

1. In the left field, open **Computer Configuration > Administrator Templates > System > Removable Storage Access**.
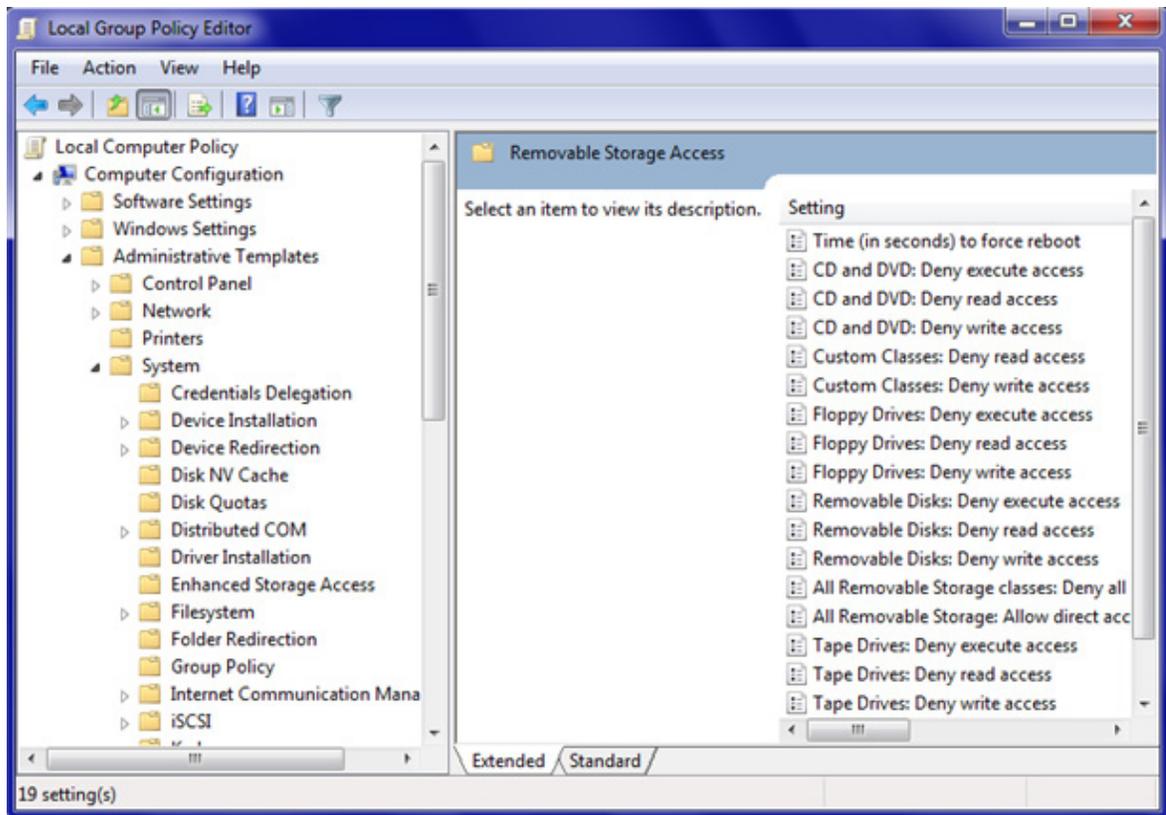
2. Table below shows the possible settings for the USB storage access:

Setting Option	Enable USB Storage (Default)	Read Only Storage	Disable USB Storage
Removable Disk: Deny execute access	Not configured	Not configured	Enable
Removable Disk: Deny read access	Not configured	Not configured	Enable
Removable Disk: Deny write access	Not configured	Enable	Enable

3. Double click on the desired setting option and the below dialog appears. As you select the desired option between Not Configured, Enabled and Disabled, the Help column at the bottom right corner displays the details of the option.
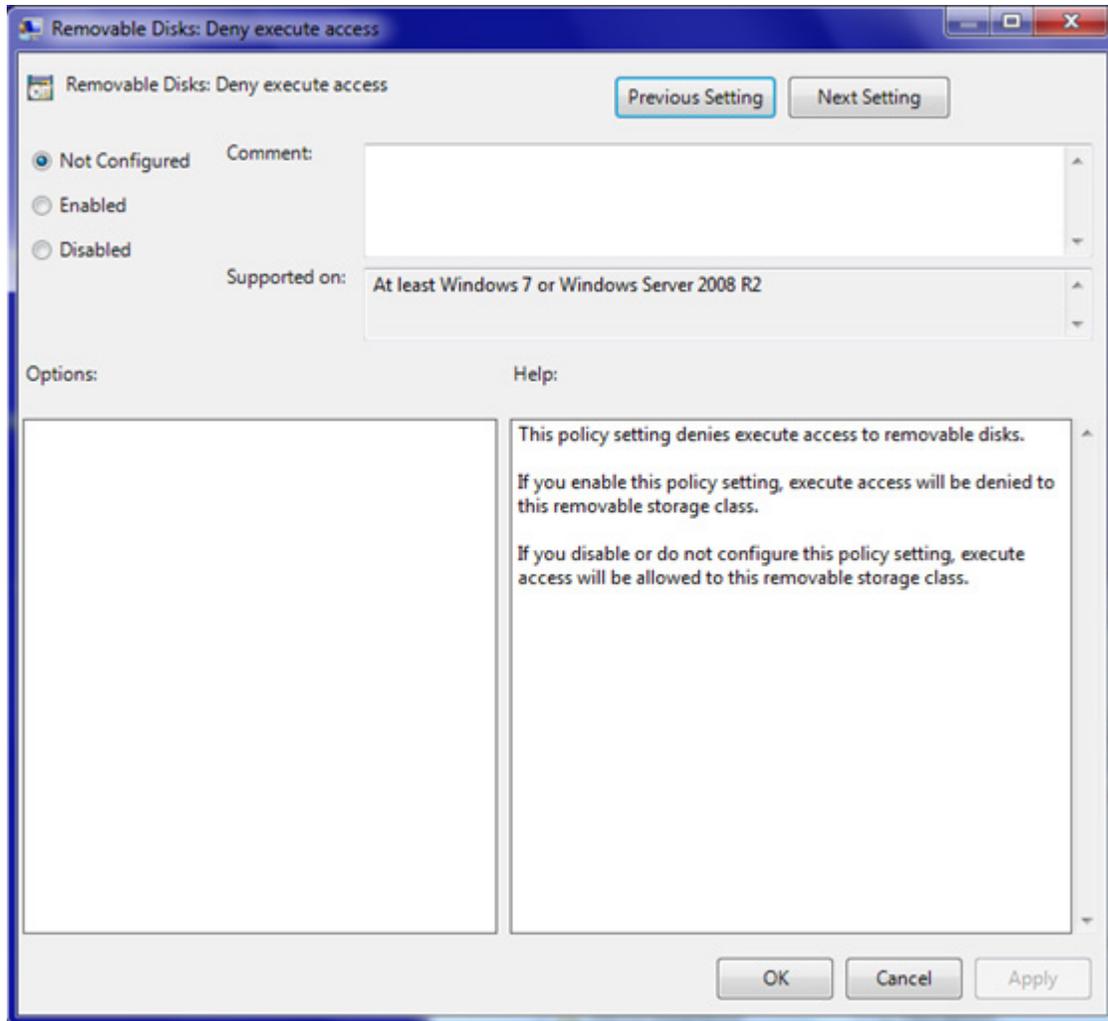
4. Select the desired option and click **Apply > OK** or just **OK**.

5. For example if your setting selection is **Removable Disk: Deny read access** and:

- *Not Configured* option is selected (default selection) - allows read access to removable storage disks.

- *Disable* option is selected - allows read access to removable storage disks.

- *Enabled* option is selected - read access to removable disks is denied.

Connecting External Accessories

The E5063A can be connected to external input/output devices such as printer, monitor, keyboard and mouse.

- [Connecting a Printer](#)
- [Connecting a Mouse/Keyboard](#)
- [Connecting a Monitor](#)

[Other topics about Using Windows](#)

Connecting a Printer

A printer can be connected to the E5063A through any USB ports on the front or rear panels. Its driver may be required to install.

Connecting a Mouse/Keyboard

A USB mouse and USB keyboard can be connected to the E5063A through any USB ports on the front or rear panels.

Connecting a Monitor

A monitor can be connected to the E5063A using the External Monitor Output Terminal (Video) located in the rear panel of the E5063A.

> **NOTE** An external monitor needs to be connected to the analyzer and turned ON before the analyzer is turned ON so that the analyzer recognizes the monitor properly.

Product Information

Product Information

- [Options](#)
- [Documentations](#)
- [Specifications](#)
- [Customer Contacts](#)
- [Error Messages](#) ([Warning Message](#))
- Default Conditions
- [Troubleshooting](#)
- [Maintenance](#)
- [Measurement Accessories](#)
- [General Principles of Operation](#)

Options for E5063A

The following list shows available options. Some options can be retrofitted on your E5063A. For upgrade (retrofit) kits, refer to http://www.agilent.com/find/ena_support and configuration guide.

- Test Set Options
- Software Options
- Accessary Options
- Calibration Option
- System Rack Options

Other topics about Product Information

Test Set Options

Option Number	Description
245	Network Analyzer 100 kHz to 4.5 GHz, S-parameter test set, 50 Ω system impedance
285	Network Analyzer 100 kHz to 8.5 GHz, S-parameter test set, 50 Ω system impedance
2H5	Network Analyzer 100 kHz to 18 GHz, S-parameter test set, 50 Ω system impedance

Software Options

Option Number	Description
011	Time Domain/Test wizard

Accessary Options

Option Number	Description
810	Add Keyboard
820	Add Mouse

These options are not displayed at the option information.

Calibration Option

Option Number	Description
A6J	ANSI Z540 Compliant Calibration

1A7	ISO 17025 Compliant Calibration

These options are not displayed at the option information.

System Rack Options

Option Number	Description	Equivalent Agilent Part Number
1CM	Rack Mount Kit	5063-9216
1CN	Front Handle Kit	5063-9229
1CP	Rack Mount and Front Handle Kit	5188-4430

These options are not displayed at the option information.

Documentations for E5063A

- Manuals
- Sales Literature

Other topics about Product Information

Manuals

The following documentations are provided with the E5063A.

Name	Description
Help (This file)	Provides the information about the measurement operation, programming, I/O interface.
Installation Guide	Provides information about start up setup and system recovery information when the Windows cannot be boot up. This is furnished with the E5063A as a hardcopy manual.
Service Guide	Provides information about the parts, troubleshooting, performance test, adjustment and service menu.

Both Installation Guide and Service Guide can be downloaded from http://www.agilent.com/find/e5063a-manual. The latest revision of Help System, Help in PDF and WebHelp formats are also available at the site.

Sales Literature

The following sales literatures are available on http://www.agilent.com/find/e5063a.

- Brochure, Brochure for PCB test
- Data sheet
- Configuration Guide (Ordering information)
- Application notes

Specifications

The Data Sheet which shows the E5063A specification is available at
http://cp.literature.agilent.com/litweb/pdf/5991-3615EN.pdf

The reader is required to see the PDF format.

Other topics about Product Information

Customer Contacts

For assistance on E5063A, refer to http://www.agilent.com/find/assist for your regional customer contacts.

Other topics about Product Information

Error Messages

Error Messages

An error message is displayed against a red background in the instrument message/warning area in the lower left part of the screen. Pushing a front panel key or executing :DISP:CCL command clears the error message. Errors caused by the operation of a front panel key simply appear on the display. They are not stored in the error queue with some exceptions.

An error with a positive error number is uniquely defined for this instrument. On the other hand, an error with a negative error number is basically defined for common GPIB devices in IEEE488.2.

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

A

No.	Message	Description
20	Additional standard needed	The GPIB command that turns ON the calibration function has been sent before to all of the data measurements needed to calculate the calibration factor that has been completed. In 1-port calibration, for example, when measurements are completed for OPEN and SHORT standards but not yet for LOAD standard:
		Try to turn ON the calibration function by calculating 1-port calibration coefficient using SENS:CORR:COLL:SAVE. Be sure to measure all necessary calibration data before sending the commands. This error is not generated by front key operations.

B

No.	Message	Description
-168	Block data not allowed	An block-data element has been received at a position where this instrument does not accept one.

C

No.	Message	Description
240	Calibration data lost	This error occurs when a file containing the system calibration data is not found or in a damaged state at time of the startup of this

No.	Message	Description

instrument, indicating a failure of this instrument. Contact the Agilent Technology sales office or the company from which you bought the instrument.

22	Calibration method not selected	This error occurs when the command for validating the calibration data, SENS:CORR:COLL:SAVE is executed before the command for selecting a calibration type, SENS:CORR:COLL:METH:xxxx, is executed. This error is not generated by front key operations.
-148	Character data not allowed	A character data element (not violating the standard) has been received at a position where this instrument does not accept one. Double quotes (") are omitted where it is necessary to place a parameter in double quotes ("), for example.
-100	Command error	A comprehensive syntax error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a command error defined in 11.5.1.1.4, IEEE488.2 has occurred.

D

No.	Message	Description
-222	Data out of range	A data element (not violating the standard) outside the range defined by this instrument has been received. This error occurs when an integer-based command for which the parameter can be rounded exceeds the range of -65536 to +65536 or when a real-number-based command for which the parameter can be rounded exceeds the range of -9.9e37 to +9.9e37, for example. This error occurs also when a numeric value other than a specified one is entered into a command in which the "port number" and "CalKit number" are specified as parameters and hence the parameters are not rounded. Such commands are, for example, SENS:CORR:COLL:ACQ:OPEN, SENS:CORR:COLL:ECAL:SOLT2, SENS:CORR:COLL:CKIT:ORD:LOAD, etc.
-	Data type	The parser has recognized a data element that

104	error	must not exist. Block data is sent instead of numeric value data or character string data that had been expected, for example.

E

No.	Message	Description
32	ECal module not in appropriate RF path	This error occurs when an ECal command, SENS:CORR:COLL:ECAL:SOLTn, is executed with the port on the ECal module not connected correctly to the instrument. When the auto-detect function is turned OFF, (SENS:CORR:COLL:ECAL:ORI OFF), however, this error does not occur even when the port on the ECal module is not connected correctly to the instrument.
502	Equation runtime error	This error occurs under the following conditions: • The trace number in data(tr)/mem(tr)/xAxis(tr) is out of range • The port number in Advanced math function is out of range # of port available depends on the model option
-200	Execution error	An error associated with execution has been generated for which this instrument cannot specify the error message. This code shows that an error associated with execution defined in 11.5.1.1.5, IEEE488.2 has occurred. This error also occurs when a calibration measurement is aborted.
-123	Exponent too large	The absolute value of the exponent exceeds 32,000 (see 7.7.2.4.1, IEEE488.2).
-178	Expression data not allowed	An expression-data element has been received at a position where this instrument does not accept one.
-170	Expression error	When the expression data is put to syntactic analysis, an error not corresponding to one of Error Numbers -171 through -179 occurs.

F

No.	Message	Description

31	Failed to configure ECal module	This error occurs when the control of the ECal module fails at the time of executing an ECal command, SENS:CORR:COLL:ECAL:SOLTn. The failure results are from the failure to connect the ECal module to the USB port, failure of the ECal module, etc.
102	Failed to copy file	This error occurs when copying a file (MMEM:COPY command) fails.
104	Failed to create directory	This error occurs when creating a directory (MMEM:MDIR command) fails.
103	Failed to delete file	This error occurs when deleting a file (MMEM:DEL command) fails.
100	Failed to read file	This error occurs when the formatted data array (MMEM:LOAD:FDAT command) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:LOAD:SEGM command) for the active channel, etc. cannot be read normally.
101	Failed to write file	This error occurs when the formatted data array (MMEM:STOR:FDATcommand) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:STOR:SEGM command) for the active channel, display image (MMEM:STOR:IMAG command) for the LCD screen, etc. cannot be written normally.
-257	File name error	A file name error. This message appears when an error exists in the file name and hence a command is not executed correctly. This error occurs when you try to copy to an unsuitable file name, for example.
-256	File name not found	The specified file name is not found and hence the command is not executed correctly. This error occurs when you try to read a file that does not exist in a disk or a disk is not correctly inserted into the drive to read or write a file, for example.
107	File transfer failed	This error occurs when writing data into or reading data from a file (MMEM:TRAN command)

		fails.
201	Firmware license not found	Frequency option or/and Generation code license codes do not exist in .lic file. See the [Backing Up License Key File](#).

G

No.	Message	Description
-105	GET not allowed	A group execution trigger (GET) has been received in the program message (see 7.7, IEEE488.2).

H

No.	Message	Description
-240	Hardware error	When hardware error occurs, this error is displayed.
-114	Header suffix out of range	The unit of the header is outside the range. The header is invalid in the unit for numeric parameters following a SCPI command.

I

No.	Message	Description
-224	Illegal parameter value	The parameter value is not suitable. This error occurs when the CALC:PAR:DEF command is used to specify an S-parameter that does not exist in the model (specify S44), for example.
-213	Init ignored	Because another measurement is in progress, the request for initiating a measurement ("INIT" command) is ignored.
35	Insufficient ECal module memory	This error occurs when the embedded memory is insufficient to save the user property in ECal module during the user definition ECal processing.
243	Insufficient privileges	Logged-in Windows account does not have administrator privileges. Re-Log-in with administrator account.
-161	Invalid block data	Block data has been expected, but the block data that appears is invalid for some reason (see 7.7.6.2, IEEE488.2). The END message is received before the length of block data has been filled, for example.

28	Invalid calibration method	If the type of calibration is not specified or not correct when partial overwrite is executed with the GPIB command, this error occurs. This error does not occur for operation with front keys.
-101	Invalid character	An invalid character exists in the program message character string.
-141	Invalid character data	An invalid character is found in the character data element, or the parameter received is not valid.
-121	Invalid character in number	A character that is invalid for the data type subject to syntactic analysis has been received. For example, a letter is found in a decimal numeric value or a numeric character "9" in octal data.
500	Invalid equation expression	The equation expression used in Equation Editor is not valid.
501	Invalid equation label	The equation label used in Equation Editor is not valid.
-171	Invalid expression	The expression-data element is invalid (see 7.7.7.2, IEEE488.2). For example, parentheses are not paired, or illegal characters are used.
-232	Invalid Format	A legal program data element was parsed but could not be executed because the data format or structure is inappropriate. For example when loading memory tables or when sending a :SYST:SET parameter from an unknown instrument.
-103	Invalid separator	The parser (a syntactic analysis program) had been expecting a delimiter, but a character that is not a delimiter has been sent.
-151	Invalid string data	Character string data has been expected, but the character string data that appears is invalid for some reason (see 7.7.5.2, IEEE488.2). The END message is received before the ending quotation mark character appears, for example.
-	Invalid suffix	The suffix does not comply with the syntax defined in 7.7.3.2, IEEE488.2. Or it does not suit

131		E5063A.

L

No.	Message	Description
53	Log sweep requires 2 octave minimum span	The span of sweep range did not satisfy the requirement for logarithmic sweep. The sweep type is automatically changed to linear sweep when this error occurs.
		For example, this error occurs when, with the start and stop frequency are set 1 MHz and 2 MHz respectively, the sweep type is changed to logarithmic sweep.
		Set the stop frequency to more than four times as many as the start frequency. And then select logarithmic sweep.

M

No.	Message	Description
-109	Missing parameter	The number of parameters is less than that required for the command, or the parameter has not been entered. For example, the :SENS:SWE:POIN command requires one more parameter. Therefore, when a message "SENS:SWE:POIN" is sent, this error is returned.
-270	Macro error	An error occurring when macro is executed.

N

No.	Message	Description
70	No GPIB system controller	The GPIB system controller is not set properly.
		This error occurs when USB/GPIB interface is not connected or the interface setting is improper, during the measurement for power calibration data or for that in scalar-mixer calibration.
		It also occurs when USB/GPIB interface is not connected or the interface setting is improper, though the control of the external signal source is enabled.
		Ensure the connection of USB/GPIB interface and its setting.

52	No valid memory trace	This error occurs when you have executed either DISP:WIND:TRAC:MEM ON command to display memory trace, or any other command to enable data calculation using memory trace (CALC:MATH:FUNC command with other than NORM is specified), though no valid data exists in memory trace. This error is not generated by front key operations.
-120	Numeric data error	An error resulting from the numeric value data (including numeric value data having no decimal point representation) has occurred. A numeric value error other than Errors -121 through -129 has occurred.
-128	Numeric data not allowed	An numeric-value-data element (not violating the standard) has been received at a position where this instrument does not accept one.

O

No.	Message	Description
200	Option not installed	The received command has been ignored because of the mismatch between the contents of an option for this instrument and the command.
		It also occurs when you try to enable the fault location function in a model not having the fault location option. This holds true for the SRL (Structural Return Loss) option.
		This error is not generated by front key operations.
-225	Out of memory	Insufficient memory is available in this instrument to perform the required operation.

P

No.	Message	Description
-220	Parameter error	When a parameter-related error other than Errors -221 through -229 occurs, that error is displayed.
-108	Parameter not allowed	The number of parameters exceeds the one required for the command.
		For instance, when a program message ":SENS1:SWE:TYPE LIN, SEGM" is sent instead of a correct program message with a command

		":SENS1:SWE:TYPE LIN" which requires a parameter, the instrument receives the message as the number of parameters is invalid. See the command reference to confirm the required number of parameters.
41	Peak not found	This error occurs when, after specifying a peak and executing the CALC:MARK:FUNC:EXEC and CALC:FUNC:EXEC commands, the specified peak is not found in the marker search analysis.
220	Phase lock loop unlocked	This error occurs when the PLL circuit of this instrument becomes unlocked while the measurement is in progress. The measurement value is not correct. This error may occur when an external reference out of specification is connected to this instrument. Should an error occur with an external reference not connected, this instrument is faulty. Contact an Agilent Technology sales office or the company from which you bought the instrument.
221	Port 1 receiver overload	The excessive signal has been input on port 1. When this error occurs, signal source is turned OFF. Check the input signal level on port 1.
222	Port 2 receiver overload	The excessive signal has been input on port 2. When this error occurs, signal source is turned OFF. Check the input signal level on port 2.
241	Power on test failed	This error occurs when the power-on test fails, indicating a failure of this instrument. Contact an Agilent Technology sales office or the company from which you bought the instrument.
61	Power unleveled	The out power level exceeds available range. For example, if the level after correction exceeds the power level that can be outputted when correcting the power level with the power slope feature, this error occurs. Check that the power level is set correctly and the correction value of the power slope is set correctly.
121	Print failed	This error occurs when printing fails for reasons other than Error 120, Printer error.

120	Printer error	This error occurs when the previous printing is still in progress or the printer fails (offline, short of paper, etc.) at the time of outputting the display image on the LCD screen to the printer (:HCOP command).
-112	Program mnemonic too long	The length of the header exceeds 12 characters (see 7.6.1.4.1, IEEE488.2).

Q

No.	Message	Description
-430	Query DEADLOCKED	The state that generates a "DEADLOCKED" Query error (see 6.3.1.7, IEEE488.2). This error occurs when both input and output buffers have become full, preventing the instrument from continuing processing, for example.
-400	Query error	A comprehensive query error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a query error defined in 11.5.1.1.7 and 6.3, IEEE488.2 has occurred.
-410	Query INTERRUPTED	The state that generates a "INTERRUPTED" Query error (see 6.3.2.3, IEEE488.2). This error occurs when data bytes (DAB) or GET are received before the transmission of the response when a query is not completed, for example.
-420	Query UNTERMINATED	The state that generates an "UNTERMINATED" Query error (see 6.3.2, IEEE488.2). This error occurs when this instrument is designated as the talker and an incomplete program message is received, for example.
-440	Query UNTERMINATED after indefinite response	After a query asking for an indefinite response has been run, another query is received in the same program message (See 6.5.7.5.7, IEEE488.2).
-350	Queue overflow	The queue contains a specific code in place of the code which caused this error. The code indicates that the error occurred because of no space is available in the queue, but the error is

		not recorded.

R

No.	Message	Description
105	Recall failed	This error occurs when reading an instrument status file (State01.sta, etc.) (MMEM:LOAD:STAT command) fails.

S

No.	Message	Description
106	Save failed	This error occurs when writing an instrument status file (State01.sta, etc.) (MMEM:STOR:STAT command) fails.
33	Selected parameter not valid for confidence check	This error occurs when the parameter except S-parameter has been selected, while using the confidence check function for calibration coefficient. Select S-parameter (S11, S21, S12 or S22).
57	SnP request not valid	This error occurs when you try to save data to a Touchstone file but no measurement has been executed.
50	Specified channel hidden	This error occurs when an attempt is made to activate a channel not on display using the DISP:WIND:ACT command. This error is not generated by front key operations.
23	Specified error term does not exist	The error occurs when the calibration coefficient type of the parameter specified in the read/write command of the calibration coefficient (SENS:CORR:COEF) is invalid for the calibration type selected by the calibration type selection command (SENS:CORR:COEF:METH:xxxx.) This error is not generated by front key operations.
21	Specified ports overlapped	This error occurs when a port number is duplicated in a command requiring two or more port numbers as parameters. Such commands are, for example, SENS:CORR:COLL:THRU 2,2. Specify port setup correctly to avoid duplication of ports. This error is not generated by front key operations.
51	Specified trace dose	This error occurs when CALC:PAR:SEL command is executed to activate more traces than specified by

	not exist	CALC:PAR:COUN command. This error is not generated by front key operations.
26	Standard not selected	This error occurs when you execute the measurement under the condition that the standard is not selected for the subclass (the standard number is set to 0).
-150	String data error	When a character-string-data element is put to syntactic analysis, an error not corresponding to one of Error Numbers -151 through -159 occurs.
-158	String data not allowed	A character-string-data element has been received at a position where this instrument does not accept one.
-138	Suffix not allowed	A suffix is attached to a numeric value element to which a suffix is not allowed to be attached.
-134	Suffix too long	The unit is too long. The unit is expressed in 12 or more characters (see 7.7.3.4, IEEE488.2).
-102	Syntax error	A command or data type that is not recognized exists.
-310	System error	One of the errors designated as "system errors" in this instrument has occurred.

T

No.	Message	Description
40	Target value not found	This error occurs when the target is not found during the marker search analysis after specifying the target and executing the CALC:MARK:FUNC:EXEC and CALC:FUNC:EXEC commands. This error occurs also when the bandwidth is not found after executing the bandwidth marker command, CALC:MARK:BWID:DATA?
-124	Too many digits	The number of digits of the argument of the decimal numeric-value-data element exceeds 255 with the preceding 0 removed (see 7.7.2.4.1, IEEE488.2).
-223	Too much data	The block-, expression-, or character-string-type program data that has been received conforms with the standard. But it exceeds the amount that

		can be processed under the condition of the memory or conditions specific to memory-related devices. In this instrument, this error occurs when the number of characters exceeds 254 in a character-string parameter.
54	Transform, Gate not allowed	This message appears when you turn ON the transform/gating function of the time domain feature, while measurement points are set to two, or sweep type is set to other than linear sweep
		Set the measurement points to three or more and the sweep type to linear sweep, and then turn ON the transform and/or gating conversion function.
-211	Trigger ignored	This instrument receives and detects a trigger command (":TRIG") or an external trigger signal. But it is ignored due to the timing condition (This instrument is not in the wait-for-trigger state, for example). Change the setup so that a trigger command or an external trigger signal can be sent after the instrument has entered the wait-for-trigger state.

U

No.	Message	Description
-113	Undefined header	A command not defined in this instrument, though not illegal in the syntactic structure, has been received. For example, when a message ":DISP:WIND1:TABL:MEM ON" is sent to a correct program message ":DISP:WIND1:TRAC1:MEM ON," the message sent is received as an undefined command by this instrument. See the command reference and use correct commands.
		This error occurs also when a port that does not exist in this model is specified in a command specifying a port number as an index. Such commands are SENS:CORR:EXT:AUTO:PORTn, SENS:CORR:COLL:ADAPn:xxxx, CALC:SRL:CONNn:xxxx, and SENS:CORR:EXT:PORTn:xxxx; they include "n" as a part.
34	User	This error occurs when the selected user profile

	characterization not found in module	is not detected in the ECal memory, while reading it from the ECal module, written by the user definition ECal.

V

No.	Message	Description
30	Valid Ecal module not found	This error occurs when a required ECal module is not connected on the E5063A. This error occurs, for example, when the Ecal characterization is executed with 4 port Ecal. Use 2 port Ecal in this case.
81	Valid USB switch not found	This error occurs when a required USB switch is not connected on the E5063A.

Error Messages

An error message is displayed against a red background in the instrument message/warning area in the lower left part of the screen. Pushing a front panel key or executing :DISP:CCL command clears the error message. Errors caused by the operation of a front panel key simply appear on the display. They are not stored in the error queue with some exceptions.

An error with a positive error number is uniquely defined for this instrument. On the other hand, an error with a negative error number is basically defined for common GPIB devices in IEEE488.2.

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

A

No.	Message	Description
20	Additional standard needed	The GPIB command that turns ON the calibration function has been sent before to all of the data measurements needed to calculate the calibration factor that has been completed. In 1-port calibration, for example, when measurements are completed for OPEN and SHORT standards but not yet for LOAD standard: Try to turn ON the calibration function by calculating 1-port calibration coefficient using SENS:CORR:COLL:SAVE. Be sure to measure all necessary calibration data before sending the commands. This error is not generated by front key operations.

B

No.	Message	Description
-168	Block data not allowed	An block-data element has been received at a position where this instrument does not accept one.

C

No.	Message	Description
240	Calibration data lost	This error occurs when a file containing the system calibration data is not found or in a damaged state at time of the startup of this instrument, indicating a failure of this

22	Calibration method not selected	This error occurs when the command for validating the calibration data, SENS:CORR:COLL:SAVE is executed before the command for selecting a calibration type, SENS:CORR:COLL:METH:xxxx, is executed. This error is not generated by front key operations.
- 148	Character data not allowed	A character data element (not violating the standard) has been received at a position where this instrument does not accept one. Double quotes (") are omitted where it is necessary to place a parameter in double quotes ("), for example.
- 100	Command error	A comprehensive syntax error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a command error defined in 11.5.1.1.4, IEEE488.2 has occurred.

D

No.	Message	Description
- 222	Data out of range	A data element (not violating the standard) outside the range defined by this instrument has been received. This error occurs when an integer-based command for which the parameter can be rounded exceeds the range of -65536 to +65536 or when a real-number-based command for which the parameter can be rounded exceeds the range of -9.9e37 to +9.9e37, for example.
		This error occurs also when a numeric value other than a specified one is entered into a command in which the "port number" and "CalKit number" are specified as parameters and hence the parameters are not rounded. Such commands are, for example, SENS:CORR:COLL:ACQ:OPEN, SENS:CORR:COLL:ECAL:SOLT2, SENS:CORR:COLL:CKIT:ORD:LOAD, etc.
- 104	Data type error	The parser has recognized a data element that must not exist. Block data is sent instead of

		numeric value data or character string data that had been expected, for example.

E

No.	Message	Description
32	ECal module not in appropriate RF path	This error occurs when an ECal command, SENS:CORR:COLL:ECAL:SOLTn, is executed with the port on the ECal module not connected correctly to the instrument. When the auto-detect function is turned OFF, (SENS:CORR:COLL:ECAL:ORI OFF), however, this error does not occur even when the port on the ECal module is not connected correctly to the instrument.
502	Equation runtime error	This error occurs under the following conditions: • The trace number in data(tr)/mem(tr)/xAxis(tr) is out of range • The port number in Advanced math function is out of range # of port available depends on the model option
-200	Execution error	An error associated with execution has been generated for which this instrument cannot specify the error message. This code shows that an error associated with execution defined in 11.5.1.1.5, IEEE488.2 has occurred. This error also occurs when a calibration measurement is aborted.
-123	Exponent too large	The absolute value of the exponent exceeds 32,000 (see 7.7.2.4.1, IEEE488.2).
-178	Expression data not allowed	An expression-data element has been received at a position where this instrument does not accept one.
-170	Expression error	When the expression data is put to syntactic analysis, an error not corresponding to one of Error Numbers -171 through -179 occurs.

F

No.	Message	Description
31	Failed to	This error occurs when the control of the ECal

	configure ECal module	module fails at the time of executing an ECal command, SENS:CORR:COLL:ECAL:SOLTn. The failure results are from the failure to connect the ECal module to the USB port, failure of the ECal module, etc.
102	Failed to copy file	This error occurs when copying a file (MMEM:COPY command) fails.
104	Failed to create directory	This error occurs when creating a directory (MMEM:MDIR command) fails.
103	Failed to delete file	This error occurs when deleting a file (MMEM:DEL command) fails.
100	Failed to read file	This error occurs when the formatted data array (MMEM:LOAD:FDAT command) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:LOAD:SEGM command) for the active channel, etc. cannot be read normally.
101	Failed to write file	This error occurs when the formatted data array (MMEM:STOR:FDATcommand) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:STOR:SEGM command) for the active channel, display image (MMEM:STOR:IMAG command) for the LCD screen, etc. cannot be written normally.
- 257	File name error	A file name error. This message appears when an error exists in the file name and hence a command is not executed correctly. This error occurs when you try to copy to an unsuitable file name, for example.
- 256	File name not found	The specified file name is not found and hence the command is not executed correctly. This error occurs when you try to read a file that does not exist in a disk or a disk is not correctly inserted into the drive to read or write a file, for example.
107	File transfer failed	This error occurs when writing data into or reading data from a file (MMEM:TRAN command) fails.

201	Firmware license not found	Frequency option or/and Generation code license codes do not exist in .lic file. See the [Backing Up License Key File](#).

G

No.	Message	Description
-105	GET not allowed	A group execution trigger (GET) has been received in the program message (see 7.7, IEEE488.2).

H

No.	Message	Description
-240	Hardware error	When hardware error occurs, this error is displayed.
-114	Header suffix out of range	The unit of the header is outside the range. The header is invalid in the unit for numeric parameters following a SCPI command.

I

No.	Message	Description
-224	Illegal parameter value	The parameter value is not suitable. This error occurs when the CALC:PAR:DEF command is used to specify an S-parameter that does not exist in the model (specify S44), for example.
-213	Init ignored	Because another measurement is in progress, the request for initiating a measurement ("INIT" command) is ignored.
35	Insufficient ECal module memory	This error occurs when the embedded memory is insufficient to save the user property in ECal module during the user definition ECal processing.
243	Insufficient privileges	Logged-in Windows account does not have administrator privileges. Re-Log-in with administrator account.
-161	Invalid block data	Block data has been expected, but the block data that appears is invalid for some reason (see 7.7.6.2, IEEE488.2). The END message is received before the length of block data has been filled, for example.
28	Invalid calibration	If the type of calibration is not specified or not correct when partial overwrite is executed with the

	method	GPIB command, this error occurs. This error does not occur for operation with front keys.
-101	Invalid character	An invalid character exists in the program message character string.
-141	Invalid character data	An invalid character is found in the character data element, or the parameter received is not valid.
-121	Invalid character in number	A character that is invalid for the data type subject to syntactic analysis has been received. For example, a letter is found in a decimal numeric value or a numeric character "9" in octal data.
500	Invalid equation expression	The equation expression used in Equation Editor is not valid.
501	Invalid equation label	The equation label used in Equation Editor is not valid.
-171	Invalid expression	The expression-data element is invalid (see 7.7.7.2, IEEE488.2). For example, parentheses are not paired, or illegal characters are used.
-232	Invalid Format	A legal program data element was parsed but could not be executed because the data format or structure is inappropriate. For example when loading memory tables or when sending a :SYST:SET parameter from an unknown instrument.
-103	Invalid separator	The parser (a syntactic analysis program) had been expecting a delimiter, but a character that is not a delimiter has been sent.
-151	Invalid string data	Character string data has been expected, but the character string data that appears is invalid for some reason (see 7.7.5.2, IEEE488.2). The END message is received before the ending quotation mark character appears, for example.
-131	Invalid suffix	The suffix does not comply with the syntax defined in 7.7.3.2, IEEE488.2. Or it does not suit E5063A.

L

No.	Message	Description
53	Log sweep requires 2 octave minimum span	The span of sweep range did not satisfy the requirement for logarithmic sweep. The sweep type is automatically changed to linear sweep when this error occurs.
		For example, this error occurs when, with the start and stop frequency are set 1 MHz and 2 MHz respectively, the sweep type is changed to logarithmic sweep.
		Set the stop frequency to more than four times as many as the start frequency. And then select logarithmic sweep.

M

No.	Message	Description
-109	Missing parameter	The number of parameters is less than that required for the command, or the parameter has not been entered. For example, the :SENS:SWE:POIN command requires one more parameter. Therefore, when a message "SENS:SWE:POIN" is sent, this error is returned.
-270	Macro error	An error occurring when macro is executed.

N

No.	Message	Description
70	No GPIB system controller	The GPIB system controller is not set properly.
		This error occurs when USB/GPIB interface is not connected or the interface setting is improper, during the measurement for power calibration data or for that in scalar-mixer calibration.
		It also occurs when USB/GPIB interface is not connected or the interface setting is improper, though the control of the external signal source is enabled.
		Ensure the connection of USB/GPIB interface and its setting.
52	No valid memory	This error occurs when you have executed either DISP:WIND:TRAC:MEM ON command to display

	trace	memory trace, or any other command to enable data calculation using memory trace (CALC:MATH:FUNC command with other than NORM is specified), though no valid data exists in memory trace. This error is not generated by front key operations.
-120	Numeric data error	An error resulting from the numeric value data (including numeric value data having no decimal point representation) has occurred. A numeric value error other than Errors -121 through -129 has occurred.
-128	Numeric data not allowed	An numeric-value-data element (not violating the standard) has been received at a position where this instrument does not accept one.

O

No.	Message	Description
200	Option not installed	The received command has been ignored because of the mismatch between the contents of an option for this instrument and the command. It also occurs when you try to enable the fault location function in a model not having the fault location option. This holds true for the SRL (Structural Return Loss) option. This error is not generated by front key operations.
-225	Out of memory	Insufficient memory is available in this instrument to perform the required operation.

P

No.	Message	Description
-220	Parameter error	When a parameter-related error other than Errors -221 through -229 occurs, that error is displayed.
-108	Parameter not allowed	The number of parameters exceeds the one required for the command. For instance, when a program message ":SENS1:SWE:TYPE LIN, SEGM" is sent instead of a correct program message with a command ":SENS1:SWE:TYPE LIN" which requires a parameter, the instrument receives the message

		as the number of parameters is invalid. See the command reference to confirm the required number of parameters.
41	Peak not found	This error occurs when, after specifying a peak and executing the CALC:MARK:FUNC:EXEC and CALC:FUNC:EXEC commands, the specified peak is not found in the marker search analysis.
220	Phase lock loop unlocked	This error occurs when the PLL circuit of this instrument becomes unlocked while the measurement is in progress. The measurement value is not correct. This error may occur when an external reference out of specification is connected to this instrument. Should an error occur with an external reference not connected, this instrument is faulty. Contact an Agilent Technology sales office or the company from which you bought the instrument.
221	Port 1 receiver overload	The excessive signal has been input on port 1. When this error occurs, signal source is turned OFF. Check the input signal level on port 1.
222	Port 2 receiver overload	The excessive signal has been input on port 2. When this error occurs, signal source is turned OFF. Check the input signal level on port 2.
241	Power on test failed	This error occurs when the power-on test fails, indicating a failure of this instrument. Contact an Agilent Technology sales office or the company from which you bought the instrument.
61	Power unleveled	The out power level exceeds available range. For example, if the level after correction exceeds the power level that can be outputted when correcting the power level with the power slope feature, this error occurs. Check that the power level is set correctly and the correction value of the power slope is set correctly.
121	Print failed	This error occurs when printing fails for reasons other than Error 120, Printer error.

120	Printer error	This error occurs when the previous printing is still in progress or the printer fails (offline, short of paper, etc.) at the time of outputting the display image on the LCD screen to the printer (:HCOP command).
-112	Program mnemonic too long	The length of the header exceeds 12 characters (see 7.6.1.4.1, IEEE488.2).

Q

No.	Message	Description
-430	Query DEADLOCKED	The state that generates a "DEADLOCKED" Query error (see 6.3.1.7, IEEE488.2). This error occurs when both input and output buffers have become full, preventing the instrument from continuing processing, for example.
-400	Query error	A comprehensive query error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a query error defined in 11.5.1.1.7 and 6.3, IEEE488.2 has occurred.
-410	Query INTERRUPTED	The state that generates a "INTERRUPTED" Query error (see 6.3.2.3, IEEE488.2). This error occurs when data bytes (DAB) or GET are received before the transmission of the response when a query is not completed, for example.
-420	Query UNTERMINATED	The state that generates an "UNTERMINATED" Query error (see 6.3.2, IEEE488.2). This error occurs when this instrument is designated as the talker and an incomplete program message is received, for example.
-440	Query UNTERMINATED after indefinite response	After a query asking for an indefinite response has been run, another query is received in the same program message (See 6.5.7.5.7, IEEE488.2).
-350	Queue overflow	The queue contains a specific code in place of the code which caused this error. The code indicates that the error occurred because of no space is available in the queue, but the error is

		not recorded.

R

No.	Message	Description
105	Recall failed	This error occurs when reading an instrument status file (State01.sta, etc.) (MMEM:LOAD:STAT command) fails.

S

No.	Message	Description
106	Save failed	This error occurs when writing an instrument status file (State01.sta, etc.) (MMEM:STOR:STAT command) fails.
33	Selected parameter not valid for confidence check	This error occurs when the parameter except S-parameter has been selected, while using the confidence check function for calibration coefficient. Select S-parameter (S11, S21, S12 or S22).
57	SnP request not valid	This error occurs when you try to save data to a Touchstone file but no measurement has been executed.
50	Specified channel hidden	This error occurs when an attempt is made to activate a channel not on display using the DISP:WIND:ACT command. This error is not generated by front key operations.
23	Specified error term does not exist	The error occurs when the calibration coefficient type of the parameter specified in the read/write command of the calibration coefficient (SENS:CORR:COEF) is invalid for the calibration type selected by the calibration type selection command (SENS:CORR:COEF:METH:xxxx.) This error is not generated by front key operations.
21	Specified ports overlapped	This error occurs when a port number is duplicated in a command requiring two or more port numbers as parameters. Such commands are, for example, SENS:CORR:COLL:THRU 2,2. Specify port setup correctly to avoid duplication of ports. This error is not generated by front key operations.
51	Specified trace dose	This error occurs when CALC:PAR:SEL command is executed to activate more traces than specified by

	not exist	CALC:PAR:COUN command. This error is not generated by front key operations.
26	Standard not selected	This error occurs when you execute the measurement under the condition that the standard is not selected for the subclass (the standard number is set to 0).
-150	String data error	When a character-string-data element is put to syntactic analysis, an error not corresponding to one of Error Numbers -151 through -159 occurs.
-158	String data not allowed	A character-string-data element has been received at a position where this instrument does not accept one.
-138	Suffix not allowed	A suffix is attached to a numeric value element to which a suffix is not allowed to be attached.
-134	Suffix too long	The unit is too long. The unit is expressed in 12 or more characters (see 7.7.3.4, IEEE488.2).
-102	Syntax error	A command or data type that is not recognized exists.
-310	System error	One of the errors designated as "system errors" in this instrument has occurred.

T

No.	Message	Description
40	Target value not found	This error occurs when the target is not found during the marker search analysis after specifying the target and executing the CALC:MARK:FUNC:EXEC and CALC:FUNC:EXEC commands. This error occurs also when the bandwidth is not found after executing the bandwidth marker command, CALC:MARK:BWID:DATA?
-124	Too many digits	The number of digits of the argument of the decimal numeric-value-data element exceeds 255 with the preceding 0 removed (see 7.7.2.4.1, IEEE488.2).
-223	Too much data	The block-, expression-, or character-string-type program data that has been received conforms with the standard. But it exceeds the amount that

		can be processed under the condition of the memory or conditions specific to memory-related devices. In this instrument, this error occurs when the number of characters exceeds 254 in a character-string parameter.
54	Transform, Gate not allowed	This message appears when you turn ON the transform/gating function of the time domain feature, while measurement points are set to two, or sweep type is set to other than linear sweep Set the measurement points to three or more and the sweep type to linear sweep, and then turn ON the transform and/or gating conversion function.
-211	Trigger ignored	This instrument receives and detects a trigger command (":TRIG") or an external trigger signal. But it is ignored due to the timing condition (This instrument is not in the wait-for-trigger state, for example). Change the setup so that a trigger command or an external trigger signal can be sent after the instrument has entered the wait-for-trigger state.

U

No.	Message	Description
-113	Undefined header	A command not defined in this instrument, though not illegal in the syntactic structure, has been received. For example, when a message ":DISP:WIND1:TABL:MEM ON" is sent to a correct program message ":DISP:WIND1:TRAC1:MEM ON," the message sent is received as an undefined command by this instrument. See the command reference and use correct commands. This error occurs also when a port that does not exist in this model is specified in a command specifying a port number as an index. Such commands are SENS:CORR:EXT:AUTO:PORTn, SENS:CORR:COLL:ADAPn:xxxx, CALC:SRL:CONNn:xxxx, and SENS:CORR:EXT:PORTn:xxxx; they include "n" as a part.
34	User	This error occurs when the selected user profile

		characterization not found in module	is not detected in the ECal memory, while reading it from the ECal module, written by the user definition ECal.

V

No.	Message	Description
30	Valid Ecal module not found	This error occurs when a required ECal module is not connected on the E5063A. This error occurs, for example, when the Ecal characterization is executed with 4 port Ecal. Use 2 port Ecal in this case.
81	Valid USB switch not found	This error occurs when a required USB switch is not connected on the E5063A.

Warning Message

A warning message is displayed in the <u>instrument message/Warning area</u> in the lower left part of the display against a gray background. Pushing a front panel key or executing :DISP:CCL command clears the message.

This message simply appears on the display, being not known to a remote environment such as a GPIB. This message is not displayed when another error (against a red background) is already displayed in the instrument message/Warning area.

Warning Messages during Measurement

Messages	Description
Calibration data lost ([parameter])/	One or more of the below calibration data is lost. The lost parameter is indicated in the message, for example, calibration data lost (band information):
	(nominal value), (band information), (local gain), (source output power), (virtual bridge), (receiver if range), (port characteristics), (absolute gain), (mixer current), (constants for sweep controller), (reference frequency),(receiver if flatness), (PLL cpc).
Calibration extrapolated	The frequency range that can be calibrated with the ECal module is exceeded. In this case, the calibration data for the minimum frequency or maximum frequency are used instead of the data exceeding the allowable frequency range for calibration.
Fixture Simulator failed to read user file	This warning occurs when the recalling files is not proper format.
Incompatible recall file	The recall state file is stored by the unit with upper frequency model. Some setting may be lost.
Invalid key code	The key code entered to enable or disable an option is incorrect. Enter the correct key code to proceed.
Peak not found	This warning message is displayed when, with the tracking turned ON, the peak specified by the marker search has not been found by the time the sweep is finished (with the tracking executed).

Segment table changed	This warning message is displayed when the setting specified segment by segment in the segment table is automatically changed by a change in the other setting.
	For example, this warning message is displayed when, with the power specified segment by segment in the segment table, the power setting for a segment is adjusted by a change in the power range setting.
Target value not found	This warning message is displayed when, with the tracking turned ON, the target specified by the marker search has not been found by the time the sweep is finished (with the tracking executed).
	This warning message is displayed also when, with the bandwidth marker displayed, the setting for the bandwidth marker is changed at the end of the sweep, or when, with the active marker changed or moved, the bandwidth is not found.
Transform, Gate not allowed	This message appears when you have set the measurement points to two, sweep type to other than linear sweep, or frequency offset to enabled, while the gating/conversion function of the time domain feature is turned on.
	If this message is displayed, the gating/conversion function is automatically turned off.
Unknown SSD image	This message appears when wrong SSD is installed.
Unknown system revision	This message appears when wrong SSD is installed.
Unable to find help file	This warning message appears when Help file could not be executed when you press the **Help** key. The file is either corrupted or unavailable.
	Re-install the firmware.
Unable to find help id file	This warning message appears when Help file could not be executed when you click a button at the menu bar and press the **Help** key to execute a help topic related to the button. The help id file is either corrupted or unavailable.
User Preset File	This warning message is displayed if no user

| not found | preset file (D:\UserPreset.sta) exists when executing the SYST:UPR command. |
| | If this message is displayed, a normal preset (:SYST:PRES) will be performed |

Test Wizard Error Messages

An error message is displayed on a dialog box. In the Test Wizard application on PC (remote control), the error message is displayed at the Error Message area of the application.

An error with a positive error number is uniquely defined for this instrument. On the other hand, an error with a negative error number is basically defined for common GPIB devices in IEEE488.2.

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

A

No.	Message	Description
20	Additional standard needed	The GPIB command that turns ON the calibration function has been sent before to all of the data measurements needed to calculate the calibration factor that has been completed. In 1-port calibration, for example, when measurements are completed for OPEN and SHORT standards but not yet for LOAD standard:
		Try to turn ON the calibration function by calculating 1-port calibration coefficient using SENS:CORR:COLL:SAVE. Be sure to measure all necessary calibration data before sending the commands. This error is not generated by front key operations.

B

No.	Message	Description
-168	Block data not allowed	An block-data element has been received at a position where this instrument does not accept one.

C

No.	Message	Description
-148	Character data not allowed	A character data element (not violating the standard) has been received at a position where this instrument does not accept one. Double quotes (") are omitted where it is necessary to place a parameter in double quotes ("), for example.

- 100	Command error	A comprehensive syntax error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a command error defined in 11.5.1.1.4, IEEE488.2 has occurred.

D

No.	Message	Description
- 222	Data out of range	A data element (not violating the standard) outside the range defined by this instrument has been received. This error occurs when an integer-based command for which the parameter can be rounded exceeds the range of -65536 to +65536 or when a real-number-based command for which the parameter can be rounded exceeds the range of -9.9e37 to +9.9e37, for example.
		This error occurs also when a numeric value other than a specified one is entered into a command in which the "port number" and "CalKit number" are specified as parameters and hence the parameters are not rounded. Such commands are, for example, SENS:CORR:COLL:ACQ:OPEN, SENS:CORR:COLL:ECAL:SOLT2, SENS:CORR:COLL:CKIT:ORD:LOAD, etc.
- 104	Data type error	The parser has recognized a data element that must not exist. Block data is sent instead of numeric value data or character string data that had been expected, for example.

E

No.	Message	Description
- 200	Execution error	An error associated with execution has been generated for which this instrument cannot specify the error message. This code shows that an error associated with execution defined in 11.5.1.1.5, IEEE488.2 has occurred. This error also occurs when a calibration measurement is aborted.
- 123	Exponent too large	The absolute value of the exponent exceeds 32,000 (see 7.7.2.4.1, IEEE488.2).
- 178	Expression data not	An expression-data element has been received at a position where this instrument does not

	allowed	accept one.
-170	Expression error	When the expression data is put to syntactic analysis, an error not corresponding to one of Error Numbers -171 through -179 occurs.

F

No.	Message	Description
102	Failed to copy file	This error occurs when copying a file (MMEM:COPY command) fails.
104	Failed to create directory	This error occurs when creating a directory (MMEM:MDIR command) fails.
103	Failed to delete file	This error occurs when deleting a file (MMEM:DEL command) fails.
100	Failed to read file	This error occurs when the formatted data array (MMEM:LOAD:FDAT command) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:LOAD:SEGM command) for the active channel, etc. cannot be read normally.
101	Failed to write file	This error occurs when the formatted data array (MMEM:STOR:FDAT command) and limit table (MMEM:STOR:LIM command) for the active trace on the active channel, segment sweep table (MMEM:STOR:SEGM command) for the active channel, display image (MMEM:STOR:IMAG command) for the LCD screen, etc. cannot be written normally.
-257	File name error	A file name error. This message appears when an error exists in the file name and hence a command is not executed correctly. This error occurs when you try to copy to an unsuitable file name, for example.
-256	File name not found	The specified file name is not found and hence the command is not executed correctly. This error occurs when you try to read a file that does not exist in a disk or a disk is not correctly inserted into the drive to read or write a file, for example.
107	File transfer	This error occurs when writing data into or

	failed	reading data from a file (MMEM:TRAN command) fails.

H

No.	Message	Description
-114	Header suffix out of range	The unit of the header is outside the range. The header is invalid in the unit for numeric parameters following a SCPI command.

I

No.	Message	Description
-224	Illegal parameter value	The parameter value is not suitable. This error occurs when the CALC:PAR:DEF command is used to specify an S-parameter that does not exist in the model (specify S44), for example.
-161	Invalid block data	Block data has been expected, but the block data that appears is invalid for some reason (see 7.7.6.2, IEEE488.2). The END message is received before the length of block data has been filled, for example.
28	Invalid calibration method	If the type of calibration is not specified or not correct when partial overwrite is executed with the GPIB command, this error occurs. This error does not occur for operation with front keys.
-101	Invalid character	An invalid character exists in the program message character string.
-141	Invalid character data	An invalid character is found in the character data element, or the parameter received is not valid.
-121	Invalid character in number	A character that is invalid for the data type subject to syntactic analysis has been received. For example, a letter is found in a decimal numeric value or a numeric character "9" in octal data.
-171	Invalid expression	The expression-data element is invalid (see 7.7.7.2, IEEE488.2). For example, parentheses are not paired, or illegal characters are used.
-103	Invalid separator	The parser (a syntactic analysis program) had been expecting a delimiter, but a character that is not a delimiter has been sent.
-	Invalid string	Character string data has been expected, but the

151	data	character string data that appears is invalid for some reason (see 7.7.5.2, IEEE488.2). The END message is received before the ending quotation mark character appears, for example.
-131	Invalid suffix	The suffix does not comply with the syntax defined in 7.7.3.2, IEEE488.2. Or it does not suit E5063A.

M

No.	Message	Description
-109	Missing parameter	The number of parameters is less than that required for the command, or the parameter has not been entered. For example, the :SENS:SWE:POIN command requires one more parameter. Therefore, when a message "SENS:SWE:POIN" is sent, this error is returned.

N

No.	Message	Description
-120	Numeric data error	An error resulting from the numeric value data (including numeric value data having no decimal point representation) has occurred. A numeric value error other than Errors -121 through -129 has occurred.
-128	Numeric data not allowed	An numeric-value-data element (not violating the standard) has been received at a position where this instrument does not accept one.

O

No.	Message	Description
-225	Out of memory	Insufficient memory is available in this instrument to perform the required operation.

P

No.	Message	Description
-220	Parameter error	When a parameter-related error other than Errors -221 through -229 occurs, that error is displayed.
-108	Parameter not allowed	The number of parameters exceeds the one required for the command. For instance, when a program message ":SENS1:SWE:TYPE LIN, SEGM" is sent instead of a correct program message with a command ":SENS1:SWE:TYPE LIN" which requires a

		parameter, the instrument receives the message as the number of parameters is invalid. See the command reference to confirm the required number of parameters.
121	Print failed	This error occurs when printing fails for reasons other than Error 120, Printer error.
120	Printer error	This error occurs when the previous printing is still in progress or the printer fails (offline, short of paper, etc.) at the time of outputting the display image on the LCD screen to the printer (:HCOP command).
-112	Program mnemonic too long	The length of the header exceeds 12 characters (see 7.6.1.4.1, IEEE488.2).

Q

No.	Message	Description
-430	Query DEADLOCKED	The state that generates a "DEADLOCKED" Query error (see 6.3.1.7, IEEE488.2). This error occurs when both input and output buffers have become full, preventing the instrument from continuing processing, for example.
-400	Query error	A comprehensive query error has occurred showing that this instrument cannot detect a more detailed error. This code simply shows that a query error defined in 11.5.1.1.7 and 6.3, IEEE488.2 has occurred.
-410	Query INTERRUPTED	The state that generates a "INTERRUPTED" Query error (see 6.3.2.3, IEEE488.2). This error occurs when data bytes (DAB) or GET are received before the transmission of the response when a query is not completed, for example.
-420	Query UNTERMINATED	The state that generates an "UNTERMINATED" Query error (see 6.3.2, IEEE488.2). This error occurs when this instrument is designated as the talker and an incomplete program message is received, for example.
-	Query UNTERMINATED	After a query asking for an indefinite response has been run, another query is received in the

440	after indefinite response	same program message (See 6.5.7.5.7, IEEE488.2).
-350	Queue overflow	The queue contains a specific code in place of the code which caused this error. The code indicates that the error occurred because of no space is available in the queue, but the error is not recorded.

R

No.	Message	Description
105	Recall failed	This error occurs when reading an instrument status file (State01.sta, etc.) (MMEM:LOAD:STAT command) fails.

S

No.	Message	Description
106	Save failed	This error occurs when writing an instrument status file (State01.sta, etc.) (MMEM:STOR:STAT command) fails.
57	SnP request not valid	This error occurs when you try to save data to a Touchstone file but no measurement has been executed.
-150	String data error	When a character-string-data element is put to syntactic analysis, an error not corresponding to one of Error Numbers -151 through -159 occurs.
-158	String data not allowed	A character-string-data element has been received at a position where this instrument does not accept one.
-138	Suffix not allowed	A suffix is attached to a numeric value element to which a suffix is not allowed to be attached.
-134	Suffix too long	The unit is too long. The unit is expressed in 12 or more characters (see 7.7.3.4, IEEE488.2).
-102	Syntax error	A command or data type that is not recognized exists.
-310	System error	One of the errors designated as "system errors" in this instrument has occurred.

T

No.	Message	Description

- 124	Too many digits	The number of digits of the argument of the decimal numeric-value-data element exceeds 255 with the preceding 0 removed (see 7.7.2.4.1, IEEE488.2).
- 223	Too much data	The block-, expression-, or character-string-type program data that has been received conforms with the standard. But it exceeds the amount that can be processed under the condition of the memory or conditions specific to memory-related devices. In this instrument, this error occurs when the number of characters exceeds 254 in a character-string parameter.
650	Test not specified	When the executed test is not specified properly.
651	Test not measured	When the executed test is not performed

U

No.	Message	Description
-113	Undefined header	A command not defined in this instrument, though not illegal in the syntactic structure, has been received. For example, when a message ":DISP:WIND1:TABL:MEM ON" is sent to a correct program message ":DISP:WIND1:TRAC1:MEM ON," the message sent is received as an undefined command by this instrument. See the command reference and use correct commands.
		This error occurs also when a port that does not exist in this model is specified in a command specifying a port number as an index. Such commands are SENS:CORR:EXT:AUTO:PORTn, SENS:CORR:COLL:ADAPn:xxxx, CALC:SRL:CONNn:xxxx, and SENS:CORR:EXT:PORTn:xxxx; they include "n" as a part.

Default Conditions

Analysis

Key Operation			Default Value	Preset	Backup	Save/Recall
Fixture Simulator	Fixture Simulator		OFF	←		√
	Topology	Device	Bal	←		√
		Port 1 (Bal)	1-2 or 2-1	←		√
		Property	OFF	←		√
	BalUn		OFF	←		√
	Port Matching	Port Matching	OFF	←		√
		Select Port	1	←		√
		Select Circuit	None	←		√
		C (C1)	0.000e-12F	←		√
		G (G1)	0.000S	←		√
		L (L1)	0.000e-9H	←		√
		R (R1)	0.0000 ohm	←		√
		C2	0.000e-12F	←		√
		G2	0.000S	←		√
		L2	0.000e-9H	←		√
		R2	0.0000 ohm	←		√
	Port Z Conversion	Port Z Conversion	OFF	←		√
		Port 1 Z0	50.000	←		√

	Real	ohm			
	Port 1 Z0 Imag	0.000 ohm	←		√
	Port 2 Z0 Real	50.000 ohm	←		√
	Port 2 Z0 Imag	0.000 ohm	←		√
De-Embedding	**De-Embedding**	OFF	←		√
	Select Port	1	←		√
	Select Type	None	←		√
Diff Matching	**Diff Matching**	OFF	←		√
	Select Bal Port	2	←		√
	Select Circuit	None	←		√
	C	0.0000 F	←		√
	G	0.0000 S	←		√
	L	0.0000 H	←		√
	R	0.0000 ohm	←		√
Diff Z Conversion	**Diff Z Conversion**	OFF	←		√
	Port1 (bal) Real	100.00 ohm	←		√
	Port1 (bal) Imag	0.00 ohm	←		√
	Port2 (bal) Real	100.00 ohm	←		√
	Port2 (bal) Imag	0.00 ohm	←		√

	Cmn Z Conversion	**Cmn Z Conversion**	OFF	←		√
		Port1 (bal) Real	25.00 ohm	←		√
		Port1 (bal) Imag	0.000 ohm	←		√
		Port2 (bal) Real	25.00 ohm	←		√
		Port2 (bal) Imag	0.000 ohm	←		√
Gating	**Gating**		OFF	←		√
	Start		-10 ns	←		√
	Stop		10 ns	←		√
	Center		0 s	←		√
	Span		20 ns	←		√
	Type		Bandpass	←		√
	Shape		Normal	←		√
	Unit		Seconds	←		√
Transform	**Transform**		OFF	←		√
	Start		0 m	←		√
	Stop		0 F	←		√
	Center		ON	←		√
	Span		210 MHz	←		√
	Type		Bandpass	←		√
	Window		Normal	←		√
		Impulse Width	230.69 ps	←		√

		Kaiser Beta	6.0000	←		√
	Unit	Seconds	Seconds	←		√
Conversion	Conversion		OFF	←		√
	Function		Z:Reflection	←		√
Limit Test	Limit Test		OFF	←		√
	Limit Line		OFF	←		√
	Limit Line Offsets	Stimulus Offset	0 Hz	←		√
		Amplitude Offset	0 dB	←		√
	Fail Sign		ON	←		√
Ripple Limit	Ripple Limit Test		OFF	←		√
	Ripple Limit		OFF	←		√
	Ripple Value		Off	←		√
	Ripple Band		1	←		√
	Fail Sign		ON	←		√
Bandwidth Limit	BW Test		OFF	←		√
	BW Display		OFF	←		√
	BW Marker		OFF	←		√
	N dB Points		3 dB	←		√
	Min Bandwidth		10 kHz	←		√
	Max Bandwidth		300 kHz	←		√

	Fail Sign		ON	←		√
Point Limit	**Point Limit Test**		A	←		√
	Point Limit		OFF	←		√
	Edit Point Limit	**Delete**				
		Add				
		Clear Point Limit Table				
		Export to CSV File...				
		Import from CSV File...				
	Fail Sign		ON	←		√

Avg

Key Operation	Default Value	Preset	Backup	Save/Recall
Avg Factor	16	←		√
Averaging	OFF	←		√
Avg Trigger	OFF	←		√
SMO Aperture	1.5 %	←		√
Smoothing	OFF	←		√
IF Bandwidth	30 kHz	←		√
IFBW Auto	OFF	←		√
IFBW Auto Limit	30 kHz	←		√

Cal

Key Operation				Default Value	Preset	Backup	Save/Recall
Correction				OFF	←		√
Calibrate	Response (Open)	Select Port		1	←		
	Response (Short)	Select Port		1	←		
	Response (Thru)	Select Ports		2-1 (S21)	←		
	Enhanced Response	Select Ports		2-1 (S21)	←		
	1-Port Cal	Select Port		1	←		
	Adapter Removal	Port 1/2	Length	Auto (0 s)	←		
		Cal Kit		85032B/E	←		
ECal	Enhanced Response			2-1 (S21 S11)	←		√
	Isolation			OFF	←		√
	Characterization			Factory	←		√
	Orientation			Auto	←		√
Property				OFF	←		√
Cal Kit				85032B/E	←		√
Modify Cal Kit	Define STDs			Define STDs	←		√
	Label Kit			85032B/E			
Port Extensions	Extensions			OFF	←		√
	Auto Port Extension	Select Ports (Port 1/Port 2)		ON	←		√
		Method		Current Span	←		√
		Method	User Span Start	5 Hz	←		√

		Method	**User Span Stop**	3 GHz	←		√
		Include Loss		OFF	←		√
		Adjust Mismatch		OFF	←		√
	Extension Port 1/Port 2	**Extension**		0 s	←		√
		Loss1		OFF	←		√
		Loss1		0 dB	←		√
		Freq1		1 GHz	←		√
		Loss2		OFF	←		√
		Loss2		0 dB	←		√
		Freq2		1 GHz	←		√
		Loss at DC		0 dB	←		√
Velocity Factor				1	←		√
Set Z0				50 Ω	←		√

Center

Key Operation	Default Value	Preset	Backup	Save/Recall
	1.50005 GHz	←		√

Display

Key Operation	Default Value	Preset	Backup	Save/Recall
Allocate Channels	x1	←		√
Number of Traces	1	←		√
Allocate Traces	x1	←		√
Display	Data	←		√
Data -> Mem	-	-	-	-
Data Math	OFF	←		√
Equation Editor	-	-	-	-
Equation	OFF	←		√
Edit Title Label	""	←		√
Title Label	""	←		√
Graticule Label	ON	←		√
Invert Color	OFF	←		√
Frequency	ON	←		√
Update	ON	←		√
E5063A Title bar	ON	←		√

Format

Key Operation	Default value	Preset	Backup	Save/Recall
	Log Mag	←		√

Macro Setup

Key Operation	Default Value	Preset	Backup	Save/Recall
Echo Window	OFF	←		√

Marker Fctn

Key Operation		Default Value	Preset	Backup	Save/Recall
Discrete		OFF	←		√
Couple		ON	←		√
Marker Table		OFF	←		√
Statistics		OFF	←		√
Annotation Options	**Marker Info X Pos**	1 %	←		√
	Marker Info Y Pos	1 %	←		√
	Align	ON	←		√
	Active Only	ON	←		√

Marker Search

Key Operation		Default Value	Preset	Backup	Save/Recall
Peak	Peak Excursion	3 dB	←		√
	Peak Polarity	Positive	←		√
Target	Target Value	0 dB	←		√
	Target Transition	Both	←		√
Multi Target	Target Value	0 dB	←		√
	Target Transition	Both	←		√
Tracking		OFF	←		√
Search Range	Search Range	OFF	←		√
	Start	0 Hz	←		√
	Stop	0 Hz	←		√
	Couple	ON	←		√
Bandwidth		OFF	←		√
Bandwidth Value		-3 dB	←		√
Notch		OFF	←		√
Notch Value		-3 dB	←		√

Marker

Key Operation	Default Value	Preset	Backup	Save/Recall
Marker1	OFF (Marker 1 is turned ON immediately after the marker softkey menu is displayed)	←		√
Ref Marker Mode	OFF	←		√

Meas

Key Operation	Default value	Preset	Backup	Save/Recall
	S11	←		√

Save/Recall

Key Operation		Default Value	Preset	Backup	Save/Recall
Save Type		State & Cal	←		√
Channel/Trace		Disp Only	←		√
Save SnP	SnP Format	Auto	←		√

Scale

Key Operation		Default value	Preset	Backup	Save/Recall
Divisions		10	←		√
Scale/Div		10.000 dB/div	←		√
Reference Position		5 Div	←		√
Reference Value		0 dB	←		√
Electrical Delay		0 s	←		√
Phase Offset		0 °	←		√
Y-Axis		Linear	←		√
Log Y-Axis	Top Value	1000 dB	←		√
	Bottom Value	0.001 dB	←		√

Span

Key Operation	Default Value	Preset	Backup	Save/Recall
	(Maximum Frequency - 10E5)	←		√
	Option 245 - 100 kHz to 4.5 GHz			
	Option 285 - 100 kHz to 8.5 GHz			
	Option 2H5 - 100 kHz to 18 GHz			

Start

Key Operation	Default Value	Preset	Backup	Save/Recall
	100 kHz	←		√

Stop

Key Operation	Default Value	Preset	Backup	Save/Recall
	(Maximum value in frequency range)	←		√
	Option 245 - 100 kHz to 4.5 GHz			
	Option 285 - 100 kHz to 8.5 GHz			
	Option 2H5 - 100 kHz to 18 GHz			

Sweep Setup

Key Operation			Default Value	Preset	Backup	Save/Recall
Power	**Power**		0 dBm	←		√
	Port Couple		ON	←		√
	Port Power	**Port 1 Power**	0 dBm	←		√
		Port 2 Power	0 dBm	←		√
	RF Out		ON	←		√
CW Freq			100 kHz	←		√
Sweep Time AUTO			ON	←		√
Sweep Delay			0 s	←		√
Points			201	←		√
Sweep Type			Lin Freq	←		√
Edit Segment Table	**Freq Mode**		Start/Stop	←		√
	List IFBW		OFF	←		√
	List Power		OFF	←		√
	List Delay		OFF	←		√
	List Time		OFF	←		√
Segment Display			Order Base	←		√

System

Key Operation			Default Value	Preset	Backup	
Invert Image			ON	←		
USB Switch Setup	USB Switch 1	USB Switch 1	None	←		
		Path	1	←		
		Lock Local Control	OFF	←		
		Switching Cycle	0	←		
	USB Switch 2	USB Switch 2	None	←		
		Path	1	←		
		Lock Local Control	OFF	←		
		Switching Cycle	0	←		
Misc Setup	Beeper	Beep Complete	ON	←		
		Beep Warning	ON	←		
	GPIB Setup	Talker/Listener Address	17	Non-changing	√	
	Network Setup	Telnet Server	OFF	Non-changing	√	
		Web Server	ON	Non-changing	√	
	Clock Setup	Show Clock	ON	←		
	Key Lock	Front Panel & Keyboard Lock	OFF	←		
		Touch Screen & Mouse Lock	OFF	←		

	Display Setup	Color Setup	Normal (**Invert** is the same)	Data Trace 1	Red:5 Green:5 Blue:0	←	
				Data Trace 2	Red:0 Green:5 Blue:5	←	
				Data Trace 3	Red:5 Green:0 Blue:5	←	
				Data Trace 4	Red:0 Green:5 Blue:0	←	
				Mem Trace 1	Red:3 Green:3 Blue:0	←	
				Mem Trace 2	Red:0 Green:3 Blue:3	←	
				Mem Trace 3	Red:3 Green:0 Blue:3	←	
				Mem Trace 4	Red:0 Green:3 Blue:0	←	
				Graticule Main	Red:3 Green:3 Blue:3	←	
				Graticule Sub	Red:1 Green:1 Blue:1	←	
				Limit Fail	Red:5 Green:0 Blue:0	←	
				Limit Line	Red:3 Green:0 Blue:0	←	
				Background	Red:0 Green:0 Blue:0	←	

		Magnification	Normal		√
	Preset Setup	**State**	Factory	Non-changing	√
		Confirm	ON	Non-changing	√
	Control Panel		OFF	←	
Backlight			ON	←	
Service Menu	**Test Menu**	**Diagnostic Test**	OFF	←	
	System Correction		ON	←	
	Init Src Ctrl		ON	←	
	Init Src Port		1	←	
	Pow. Chg. Wait		ON	←	
	Pow. Chg. Wait Time		50 ms	←	
	Overload Detect		ON	←	
	Security Level		None	←	

Trigger

Key Operation	Default Value	Preset	Backup	Save/Recall
Continuous	Continuous (Ch1) Hold (Other Ch)	←		√
Trigger Source	Internal	←		√
Trigger Event	On Sweep	←		√
Trigger Scope	All Channel	←		√
Ext Trig Input	Negative Edge	←		√
Trigger Delay	0 sec	←		√
Ext Trig Output	OFF	←		√
Polarity	Positive Pulse	←		√
Position	After Point	←		√
Pulse Width	1 µs	←		√

Troubleshooting

This section describes the steps you should take when you believe that the Agilent E5063A is operating improperly. The results of these simple investigative procedures may help you avoid the down-time and inconvenience of repair service. The troubleshooting instructions are divided into three categories.

When all troubleshooting measures are taken but it does not work. Contact Agilent Technology's Customer Contact.

- Troubleshooting during Startup
- Troubleshooting during Operation
- Troubleshooting for External Devices

Troubleshooting during Startup

Symptom	Solution
Turning ON (|) the standby switch does not start up the system.	- Confirm that the power cable is properly plugged in. - Confirm that the line switch on the rear panel is turned ON.
Standby switch color is red and does not start up the system.	There is some trouble on the fan. Turn OFF and ON the line switch once, then turn ON this standby switch.
The system starts up, but it automatically shuts down immediately.	Execute the system recovery.
The system starts up, but it enters the service mode (The instrument status bar in the lower right part of the screen displays SVC in red).	Execute the system recovery.
The measurement screen appears after startup, but the date and time displayed on the instrument status bar in the lower right part of the screen differ greatly from the previous settings.	Execute the system recovery.
The measurement screen appears after	Execute the system

startup, but the power-on test fails, with Error Message 241 appearing against the red background in the instrument message/warning area in the lower left part of the screen.

recovery.

Troubleshooting during Operation

Symptom	Solution
The Error Message "Port N receiver overload" (N denotes a port number) is Displayed. During the measurement of an amplifier, Error Messages 720 through 721 "Port N receiver overload" (N denotes a port number) are displayed. This error occurs when the input to a test port exceeds the maximum input level in the measurement of an amplifier. The measurement value obtained in such cases is not correct. In the worst case, a failure (damage to the receiver) may occur.	• Disconnect any cable and adapter in the test ports. • Change the measurement condition so that the input to the test port does not exceed the maximum input level.
A clearly abnormal measurement value The measurement value is not reproducible, or clearly abnormal.	• Confirm that the DUT, connection cables, and other parts are connected correctly. • Confirm that the connectors and cables used to connect the DUT are free from damage and poor contact. • Confirm that the calibration has been executed correctly. If you have not acquired a correct error correction factor, you cannot obtain a correct measurement value. • Confirm that the calibration kit is selected correctly. • Confirm that the calibration kit is defined correctly.

	• Confirm that if the stimulus signal output is turned ON.
The system cannot operate manually (Front Panel Keys, Keyboard, Touch Screen and Mouse) The keyboard or mouse becomes inoperable.	• Confirm that the keyboard or mouse is connected correctly. When it is connected correctly, turn OFF the power once, and restart the system.
The front panel key or keyboard becomes inoperable.	• Using the mouse, turn **System > Key Lock > Front Panel & Keyboard Lock OFF**.
The touch screen becomes inoperable.	• Using the front panel keys, press **System** > **Key Lock** > **Touch Screen & Mouse Lock** OFF to turn OFF the lock. • Execute the calibration of the touch screen.
The mouse becomes inoperable.	• Using the front panel keys, press **System** > **Key Lock** > **Touch Screen & Mouse Lock** to turn OFF the lock.
All of the front panel keys, keyboard, and mouse become inoperable.	• Confirm that the keyboard or mouse is connected correctly. When it is connected correctly, turn OFF the power once, and restart the system.
The keyboard and mouse have been connected after power-on.	• Turn OFF the power once, and restart the system. When taking all these measures does not recover operability, there is a possibility of a failure.
The screen freezes and all operations become impossible. The measurement in progress or screen update is stalled and all of the front panel keys, keyboard, mouse, and touch screen are inoperable.	• Press the standby switch to turn OFF the power once, and restart the system.

The system freezes while in operation.	• Press the standby switch to turn OFF the power once, and restart the system.
The rear cooling fan does not operate.	• There is a possibility of a failure.
The sweep action stops during measurement or is not executed. An error or warning message appears.	• There is a possibility of a failure.
An error or warning message is displayed on the instrument message/warning area in the lower part of the screen	• Refer to Error Messages and Warning Messages.

Troubleshooting for External Devices

Symptom	Solution
Cannot output to a printer Cannot output a measurement screen or data to a printer. Attempting to output to a printer causes Error Messages 120 and 121 to appear.	• Confirm that the power to the printer is turned ON and that the line cable is connected correctly. • Confirm that the connector cable of the printer is connected correctly. • Confirm that the printer is online. • Confirm that the printer has not run out of paper. • Confirm that the printer has not run out of ink.
Does not respond to an external controller/fails to function normally A GPIB device does not respond to the external controller, or fails to function normally.	• Confirm that the GPIB address is defined correctly. • Confirm that the GPIB cable is connected. • Confirm that another instrument connected by the GPIB cable has the same

| | GPIB address. |
| | • Confirm that the GPIB cable connection forms a loop. |

Maintenance

Maintenance

- [Backing Up the Data](#)
- [Cautions Applicable to Requesting Repair, Replacement, Regular Calibration, etc.](#)
- [Cleaning this Instrument](#)
- [Replacement of Parts with Limited Service Life](#)
- [System Recovery](#)
- [Updating Firmware](#)
- [Service Functions](#)
- [Removing Log Data](#)

Removing Log Data

The E5063A creates automatic log of data for troubleshooting purpose. For security reasons, if this data needs to be deleted, then :SERV:LOGG:CLE command can be used to clear the log recorded by the E5063A.

The log file stores data related to:

- Power ON time
- Number of times of power ON
- Result of power ON test
- Number of times of overload
- Event Log
- Hardware driver installation/error Log
- Connected USB device Log
- User calibration Log
- Firmware error Log
- Temperature of analog boards
- Internal test result

Backing Up the Data

Be sure to regularly back up your important data (including program) files in this instrument to a CD-R or other backup medium. Agilent Technologies shall not be liable for any data damages caused by improper function of this instrument.

Making Backup Files

You can make backup files on the storage of an external PC using following methods.

You can access to drive D: of the E5063A from an external PC via LAN, and copy your important data files on the drive D: to the external PC.

You can transfer your important data files on the drive D: of the E5063A to the external PC using :MMEM:TRAN command via GPIB.

CAUTION   Do not modify any files and folders in drives other than drive D:. Doing so will cause malfunction. Even in drive D, do not change the files and folders other than you created.

Other topics about Maintenance

Cautions Applicable to Requesting Repair, Replacement, Regular Calibration, etc.

- [Backing Up Data in the storage](#)
- [Devices to be Sent Back for Repair or Regular Calibration](#)

Other topics about Maintenance

Backing Up Data in the storage

The user is requested to back up the stored programs and data into external media by using the instrument's storing function before requesting the Company's Service Center to repair the instrument or replace storages.

See [Making Backup Files](#) for how to make backup files.

Please take note that the Company will not be held liable to any extent for potential erasure or change of stored programs or data due to the repair or replacement of storages performed by the Company. When a storage itself fails, the programs and data stored in it cannot be recovered.

Devices to be Sent Back for Repair or Regular Calibration

If it is necessary to send the unit to the Service Center of Agilent Technologies for repair or regular calibration, please follow the instructions below.

Equipment to be Sent

When requesting repair or regular calibration of the unit by our Service Center, send only the E5063A main unit without any installed option you may have ordered. Unless specifically instructed, it is not necessary to send accessories and calibration kits.

Packing

Use the original package and shock absorbers, or equivalent anti-static packing materials, when sending the unit.

Shipping Address

For the location of the nearest Agilent Technologies Service Center, contact the [Customer Contact](#).

Recommended Calibration Period

The recommended calibration period for this instrument is one year. The user is recommended to request the Company's Service Center to perform regular calibration every year.

Cleaning this Instrument

-
-
-

Other topics about Maintenance

This section describes how to clean the instrument.

To protect yourself from electrical shock, be sure to unplug the power cable from the outlet before cleaning the instrument.

Never clean the internal components of the instrument.

Cleaning an LCD

Use one of the following methods to clean the display surface regularly.

- For normal cleaning, rub the surface gently with a dry, soft cloth.

- When stains are difficult to remove, gently wipe the surface with cloth damped with a small amount of dehydrated ethanol.
  You can clean the standard LCD (no touch screen function) with a cloth dipped in water and then wrung tightly.

- Do not use chemicals other than dehydrated ethanol to wet the cleaning cloth. To clean the touch screen LCD, do not wet the cloth with water.

Maintenance of Test Ports and Other Connectors/Ports

The ports of the E5063A are fitted with BNC and N Types connectors. Stains or other damage to these connectors would significantly affect the accuracy in measurements. Always pay attention to the following precautions.

- Always keep the connectors free from stains and dust.

- Do not touch the contact surface on the connectors.

- Do not plug damaged or scratched connectors into the test ports.

- Use compressed air for cleaning connectors. Do not use abrasives under any circumstance.

The above precautions must also be observed in maintaining connectors and ports other than these test ports.

Cleaning Parts Other than the LCD, Test Ports, and Other Connectors/Ports

To remove stains on parts other than the LCD, test ports, and other connectors/ports of the instrument, wipe them gently with a soft cloth that is dry or wetted with a small amount of water and wrung tightly.

Replacement of Parts with Limited Service Life

This instrument incorporates parts with limited service life as shown in the following table. Using the recommended replacement time as a guide, request the Agilent Service Center to replace these parts. However, a part may need to be replaced at an earlier time than that listed in the table, depending on conditions such as location, frequency of use, and where it is stored.

- Each service life and recommended replacement time listed below is for reference only and does not imply a guarantee of the part's service life.

Part Name	Service Life (Parts supplier reference value)	Recommended replacement time
Solid State Drive (SSD)	5 years or 20,000 operating hours, whichever comes earlier. Exchanging hard disk drives causes the contents written after shipment from the factory (LAN setup, etc.)to be initialized to the state at the time of shipment. The programs and data stored in Drive D (user directory) are erased.	3 years
Main fan CPU fan	50,000 operating hours. The service life may be significantly shorter when used in a dusty and dirty environment.	5 years
Battery on Mother board	50,000 operating hours. The service life may be shorter if E5063A power has not been turned on for long time.	5 years
Power supply	50,000 operating hours (Depends on the service life of the power supply cooling fun) The service life may be significantly shorter when used in a dusty and dirty environment.	5 years
LCD screen backlight	50,000 operating hours. When the unit is used for automatic measurements in a production line and the on-screen information is not required, the life of the LCD backlight can be saved by turning it OFF. As for the method of turning the	5 years

	backlight OFF, refer to Turning OFF the LCD Screen Backlight.	
Touch screen (function)	One million times (dotting life)	5 years
USB receptacle	1,500 cycles insertion/extraction. The service life may be shorter when used in a dusty and dirty environment. In case that the insertion/extraction is in heavy usage such as Ecal in the production line, using USB extension cable may save the USB receptacle life.	N/A

Other topics about Maintenance

System Recovery

By executing system recovery, you can return the Windows operating system of the E5063A to the factory state or the user state at the setting the user performed save user state.

The procedure of system recovery is described in both Installation Guide and Service Guide.

Other topics about Maintenance

Updating Firmware

- Overview
- Procedure

Other topics about Maintenance

Overview

User can update E5063A firmware by themselves. The latest firmware can be downloaded from the http://www.agilent.com/find/ena_support.

Updating firmware does not include the following software update.

- Windows Operating System
- Driver for Windows
- Calibration Constant Data

NOTE　　　Firmware update can only be performed with administrator account.

Procedure

1. User should log in as an administrator authority to perform firmware update.

1. Download the latest firmware from the download site. It is prepared as execution file (**E5063A_xx.xx.exe**).

2. Run the **E5063A_xx.xx.exe** to extract the **E5063.msi**.

   If you use your local PC to download, save the file to a USB mass storage device in order to move it to E5063A, then connect the USB mass storage device into the front USB port of the E5063A.

3. Press **System** > **Service Menu** > **Update Firmware**.

4. In Open dialog box, select **E5063.msi**, then press **Open**. Then the windows installer appears.

5. Follow the instruction of windows installer. After the installation is finished, the instrument restarts.

6. Press **System** > **Firmware Revision** to confirm the firmware revision you have just installed.

Service Functions

This menu (**System** > **Service Menu** > **Test** and **Service Functions**) provides information about various test related to the E5063A.

For more information about any test related to the E5063A, like System Test, please refer to the E5063A Service manual which is available at www.agilent.com/find/e5063a-manual.

Under the service menu, the following topic is described in this help.

Other topics about Maintenance

Measurement Accessories

Measurement Accessories

- [Calibration Kits](#)
- [System Accessories](#)

Agilent Technologies provides various probes, cables and adapters. Refer to http://www.agilent.com/find/na_accessories for more information.

Other topics about Product Information

Calibration Kits

- [Overview](#)
- [Mechanical Calibration Kit](#)
- [ECal](#)

[Other topics about Measurement Accessories](#)

Overview

Calibration kits are used to improve the accuracy of the analyzer in various measurements.

The electronic calibration kit reduces the time required for calibration, misconnections, and wear and tear on connectors since it requires fewer changes of connection than the mechanical type.

Specifications for calibration kits and the availability of particular calibration kits are subject to change without prior notice.

Refer to http://www.agilent.com/find/na_accessories for more information.

Mechanical Calibration Kit

Model Name	Description	Connector Type	Frequency Range
85031B	Calibration Kit	APC-7	DC to 6 GHz
85032B/E	Calibration Kit	Type-N, 50 Ω	DC to 6 GHz
85032F	Mechanical Calibration Kit	Type-N, 50 Ω	DC to 9 GHz
85033D	Calibration Kit	3.5 mm	DC to 6 GHz
85033E	Mechanical Calibration Kit	3.5 mm	DC to 9 GHz
85036B	Calibration Kit	Type-N, 75 Ω	DC to 3 GHz
85036E	Economy Calibration Kit	Type-N, 75 Ω	DC to 3 GHz
85038A/F/M	Calibration Kit	7-16	DC to 7.5 GHz
85039B	Calibration Kit	Type F , 75 Ω	DC to 3 GHz
85050C	Precision Mechanical Calibration Kit	APC-7	DC to 18 GHz
85050D	Economy Mechanical Calibration Kit	APC-7	DC to 18 GHz

85052D	Economy Mechanical Calibration Kit	3.5 mm	DC to 26.5 GHz
85052C	Precision Mechanical Calibration Kit	3.5 mm	DC to 26.5 GHz
85054D	Economy Calibration Kit	Type-N, 50 Ω	DC to 18 GHz
85056D	Economy Calibration Kit	2.4 mm	DC to 50 GHz
85056K	Mechanical Calibration Kit	2.4 mm and 2.92 mm	DC to 50 GHz
K11644A	Mechanical Calibration Kit (waveguide)	WR-42	18 GHz to 26.5 GHz
P11644A	Mechanical Calibration Kit (waveguide)	WR-62	12.4 GHz to 18 GHz
X11644A	Mechanical Calibration Kit (waveguide)	WR-90	8.2 GHz to 12.4 GHz

For more information on the definition of calibration kit, refer to
http://na.tm.agilent.com/pna/caldefs/stddefs.html

ECal (electronic calibration) kit

The ECal supported by E5063A are defined below:

Model Name	Connector Type	Frequency Range
85091B	7 mm	300 kHz to 9 GHz
85091C	7 mm	300 kHz to 9 GHz
85092B	Type N	300 kHz to 9 GHz
85092C	Type N	300 kHz to 9 GHz
85093B	3.5 mm	300 kHz to 9 GHz
85093C	3.5 mm	300 kHz to 9 GHz
85096B	Type N, 75 Ω	300 kHz to 3 GHz
85096C	Type N, 75 Ω	300 kHz to 3 GHz
85098B	7-16	300 kHz to 7.5 GHz
85098C	7-16	300 kHz to 7.5 GHz
85099B	Type F	300 kHz to 3 GHz
85099C	Type F	300 kHz to 3 GHz

N4431A	3.5 mm, Type N	300 kHz to 9 GHz
N4431B	3.5 mm, Type N	9 kHz to 13.5 GHz
N4432A	Type N	300 kHz to 18 GHz
N4433A	3.5 mm	300 kHz to 20 GHz
N4690A	Type N	10 MHz to 18 GHz
N4690B	Type N	300 kHz to 18 GHz
N4691A	3.5 mm	10 MHz to 26.5 GHz
N4691B	3.5 mm	300 kHz to 26.5 GHz
N4692A	2.92 mm	10 MHz to 40 GHz
N4696A	7 mm	10 MHz to 18 GHz
N4696B	7 mm	300 kHz to 18 GHz

NOTE   When ECal is used with Option 2H5 and in the following condition, the waiting time is added automatically:

The maximum frequency of your ECal is over 13 GHz and the stop frequency on your ENA set over 8.5 GHz. (The waiting time is NOT added when stop frequency is set below 8.5GHz.)

System Accessories

- [USB/GPIB Interface](#)
- [GPIB Cables](#)
- [Agilent IO Libraries](#)

[Other topics about Measurement Accessories](#)

82357A/B USB/GPIB Interface

The 82357A/B can be used to control external GPIB devices by the E5063A. It can also be used to control E5063A by PC with USB. See "82357B USB/GPIB Interface High-Speed USB 2.0" in http://www.agilent.com/find/gpib for more information.

GPIB Cables

The following GPIB cables can be used to connect the analyzer with an external device such as a computer.

Product Number	Length
10833A	1.0 m (3.3 ft)
10833B	2.0 m (6.6 ft)
10833C	3.0 m (9.9 ft)
10833D	0.5 m (1.6 ft)

Agilent IO Libraries

Agilent IO Libraries is a collection of libraries that gives you the ability to use your instruments as a test and measurement program, and utilities that help you quickly and easily connect your instruments to your PC.

See the IO libraries manual for more information. The latest revision of IO Libraries can be downloaded from http://www.agilent.com/find/iolib.

CAUTION   Do not update Agilent IO Library on E5063A besides Agilent recommends to do so. Refer to http://www.agilent.com/find/ena_support for more information.

General Principles of Operation

General Principles of Operation

- [System Description](#)
- [Data Processing](#)

Other topics about Product Information

System Description

- [Synthesized Source](#)
- [Signal Separator](#)
- [Receiver](#)

Other topics about General Principles of Operation

A network analyzer supplies sweep signal to a DUT, measures its transmission and reflection, and displays the results as ratios against the input signal from the signal source. The E5063A network analyzer consists of the circuit modules shown in the following figure.

System Diagram for the E5063A Network Analyzer

Synthesized Source

The synthesized source generates a sweep signal in the specified frequency range.

The signal source is phase-locked to a highly reliable quartz crystal oscillator to maintain a high level of accuracy in its frequency as well as to achieve precise phase measurements.

The signal from the source is switched by the multiplexer in order to distribute it to either Port 1 or 2.

Signal Separator

The signal separator consists of bridges that detects input and output signals at the test ports. On the test port which the signal is output, the output signal and the reflection from the DUT are detected as the reference signal (R) and the test signal (T), respectively.

Receiver

The E5063A has two receivers. One is for the reference signal (R) and another is for the test signal (T). The signal from either Port 1 or 2 is switched by the multiplexer.

Each signal that is sent to the receiver is first converted into an IF signal by a mixer and then converted into a digital signal by an ADC (analog to digital converter). The digital data is then analyzed by a micro processor and measurement results will be displayed on the screen.
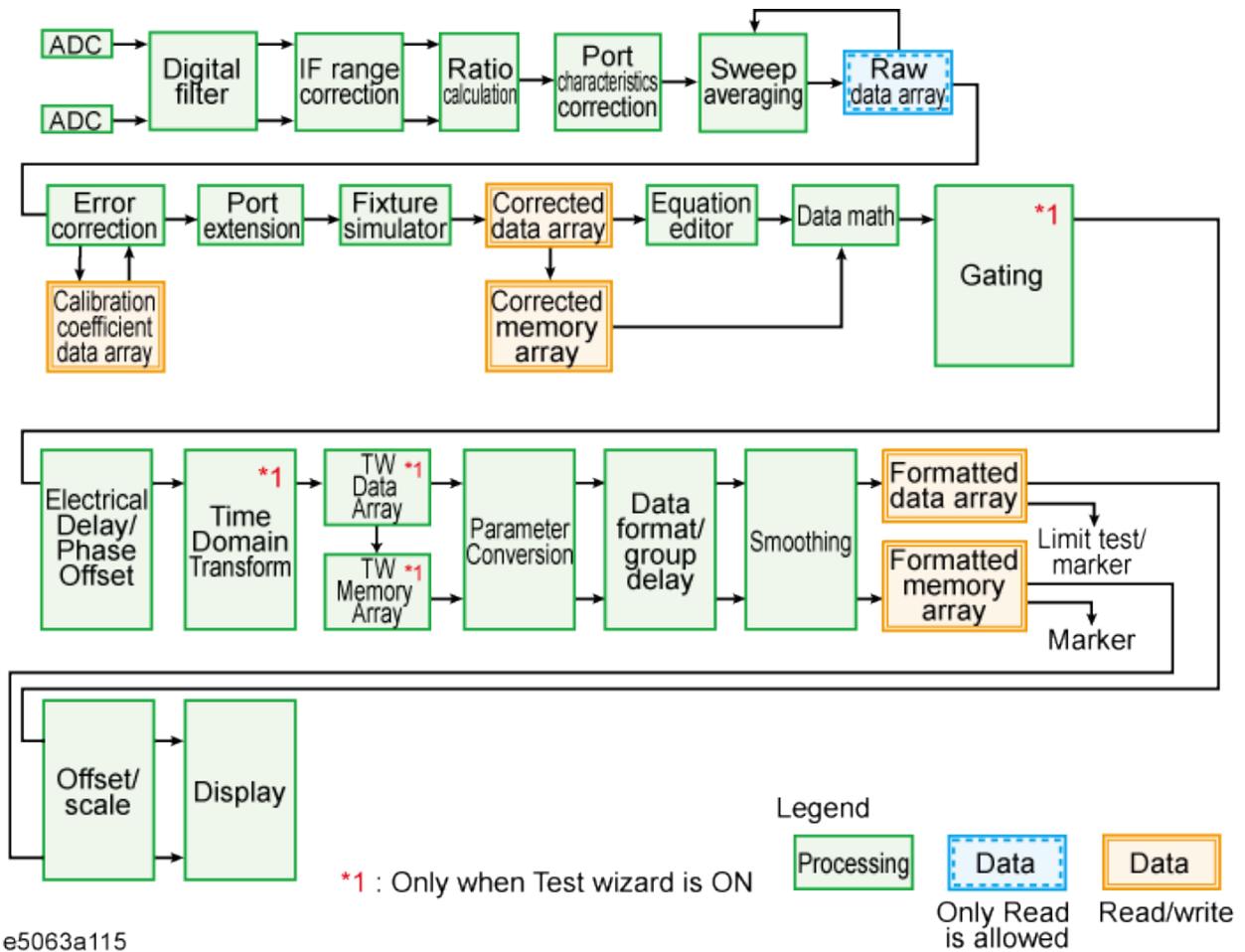
Data Processing

The internal data processing flowchart for the E5063A is shown in the following figure.

Data Processing Flowchart
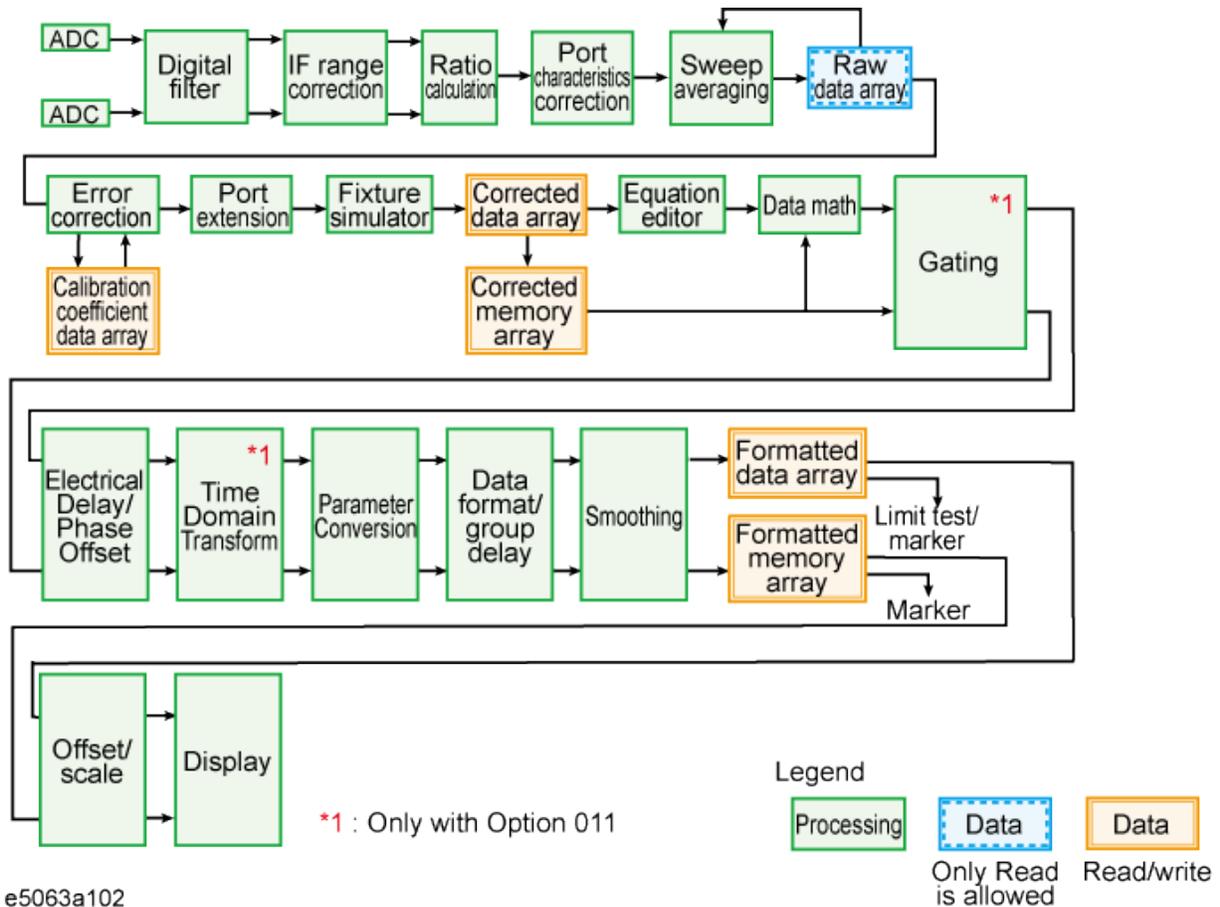
e5063a102

Data Processing Flowchart for Test Wizard

e5063a115

ADC

The ADCs (analog-to-digital converters) convert analog signals. One ADC is available for each signal and the conversion takes place simultaneously.

Digital Filter

The digital filter performs a discrete Fourier transformation (DFT) and picks up IF signals. Each IF signal is then converted into a complex number that has a real part and an imaginary part. The IF bandwidth of the analyzer is equivalent to the bandwidth of the DFT filter.

IF Range Correction

Input signals that went through ranging at the receiver are reverted (corrected) to previous values before the ranging.

Ratio Calculation

The ratio between two signals is determined by performing divisions on complex numbers. In the case of absolute measurements, the ratio of complex number cannot be calculated.

Port Characteristics Correction

The equivalent source match error, the directivity error, and the tracking error of each test port bridge are corrected. In the case of absolute measurements, the gain of each test port is corrected.

Sweep Averaging

The average of complex indices is determined based on data obtained from multiple sweep measurements. Sweep averaging is effective in reducing random noise in measurements.

Raw Data Array

The results from all data processing done up to this point are stored in this array as raw data. All prior data processing is performed as each sweep takes place. The user is not allowed to access (read/write) this raw data array.

Error Correction/Calibration Coefficient Data Array

When error correction is enabled, the process eliminates the system errors that are reproducible and stored in the calibration coefficient data array. It accommodates everything from the simple vector normalization to the full 12-term error correction. The user is allowed to access (read/write) this calibration coefficient data array. Gain correction is performed in absolute measurement.

Port Extension

This process carries out a simulation of adding or eliminating a variable length transmission path on each test port so that the reference plane of calibration is moved. The port extension is defined by an electrical delay (sec).

Fixture Simulator

A data conversion by the fixture simulator function is executed. The fixture simulator function is a collective term for six different functions: balanced-unbalanced conversion, addition of matching circuits, port reference impedance conversion, network elimination, addition of differential matching circuits, and differential reference impedance conversion.

Corrected Data Array

Unlike the raw data array, this array stores the results obtained after error corrections, port extensions, or the fixture simulator functions are applied. The user is allowed to read/write data from/to the corrected data array.

Corrected Memory Array

By pressing **Display** > **Data -> Mem**, the contents of the corrected data array is copied to this array. The user is allowed to read/write data from/to the corrected memory array.

Equation Editor

Equation Editor allows users to use a custom equation to display data in the E5063A. Equation Editor can be accessed through **Display** > **Equation Editor**.

Data Math

Data processing is carried out using the corrected data array and the corrected memory array. Four types of data processing addition, subtraction, multiplication, and division are available.

Electrical Delay/Phase Offset

An electrical delay and a phase offset are applied to each trace. By setting an electrical delay, a linear phase that is proportional to the frequency will be added or subtracted. On the other hand, setting a phase offset adds or subtracts a phase that is constant throughout the frequency range. Incidentally, data processing performed from this point on in the flowchart is applied to both the data array and the memory array.

Data Format/Group Delay

Complex data consisting of the real parts and the imaginary parts are converted into scalar data according to the data format of user's choice. Group delays are also calculated here.

Smoothing

By enabling the smoothing function, each point in a sweep measurement is replaced by a moving average of several measurement points nearby. The number of points used in calculating a moving average is determined by the smoothing aperture set by the user. The smoothing aperture is defined by a percentage against the sweep span.

Formatted Data Array/Formatted Memory Array

All results from data processing are stored in the formatted data array and the formatted memory array. The marker functions are applied to these arrays. The limit test is applied to the formatted data array. The user is allowed to read/write data from/to these arrays.

Offset/Scale

Each set of data is processed so that traces can be drawn on the screen. Particular scaling depending on the data format is applied using the position of the reference line, the value of the reference line, and the scale/graticule settings.

Display

The results obtained after data processing are displayed on the screen as traces.

Revision History

Revision History

- [Firmware Revision History](#)
- [Storage Revision History](#)

Other topics about Product Information

Firmware Revision History

- [A.01.0x](#)

Other topics about Revision History

A.01.0x

- First revision

Storage Revision History

Storage revision is based upon a number of factors such as windows OS and driver upgrade patch which are installed at the factory shipment. Firmware revision denotes E5063A measurement software. To know about the E5063A storage revision, refer to Checking the product information.

- KA10x

Other topics about Revision History

KA10x

- First revision