RIGOL

DNA Series

Vector Network Analyzer

User Guide

2026.09

Guaranty and Declaration

Copyright

© 2026 RIGOL TECHNOLOGIES CO., LTD. All Rights Reserved.

Trademark Information

RIGOL ® is the trademark of RIGOL TECHNOLOGIES CO., LTD.

Software Version

Software upgrade might change or add product features. Please obtain the latest version of the manual from **RIGOL** website or contact **RIGOL** to upgrade the software.

Notices

- RIGOL products are covered by P.R.C. and foreign patents, issued and pending.
- RIGOL reserves the right to modify or change parts of or all the specifications and pricing policies at the company's sole decision.
- Information in this publication replaces all previously released materials.
- Information in this publication is subject to change without notice.
- **RIGOL** shall not be liable for either incidental or consequential losses in connection with the furnishing, use, or performance of this manual, as well as any information contained.
- Any part of this document is forbidden to be copied, photocopied, or rearranged without prior written approval of **RIGOL**.

Product Certification

RIGOL guarantees that this product conforms to the national and industrial standards in China as well as the ISO9001:2015 standard and the ISO14001:2015 standard. Other international standard conformance certifications are in progress.

Contact Us

If you have any problem or requirement when using our products or this manual, please contact **RIGOL**.

E-mail: service@rigol.com

Website: www.rigol.comhttp://www.rigol.com/

Contents

Guaranty and Declaration **1**

1 Document Overview **6**

2 DNA Series Product Models **7**

2.1 DNA6000 Series 7

2.2 DNA6000-R Series 7

2.3 DNA5000 Series 7

2.4 DNA800 Series 8

3 Safety Requirement **8**

3.1 General Safety Summary 8

3.2 Safety Notices and Symbols 10

3.3 EMC Level 10

3.4 Ventilation Requirement 11

3.5 Working Environment 11

3.6 Care and Cleaning 12

3.7 Environmental Considerations 13

4 Quick Start **13**

4.1 General Inspection 13

4.2 Product Overview 13

4.2.1 Product Appearance 14

4.2.2 User Interface 14

4.3 To Prepare for Use 19

4.3.1 To Connect to Power 19

4.3.2 Turn-on Checkout 19

4.3.3 To Set the System Language 20

4.4 Basic Operation 20

4.4.1 Mouse / Keyboard / Touchscreen Operation Rules 20

4.4.2 Menu Operation 21

4.4.3 Parameter Setting 22

4.4.4 To Use the Built-in Help System 22

4.4.5 To View the Option and the Option Installation 22

4.5 Basic Test Procedures 23

5 Trace **25**

5.1 Trace 1-8/Trace 9-16 25

5.2 Trace Setup 26

6 Channel **27**

6.1 Channel 1-9 28

6.2 Channel Setup 28

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

6.3 Window 1-9 29
6.4 Window Setup 30
6.5 Sheet Setup 31
7 Frequency 32
8 Sweep 32
8.1 Main 32
8.2 Sweep Timing 34
8.3 Segment Control 36
9 Measure 37
9.1 S-Parameter 37
9.2 Balanced S-Parameter 39
9.2.1 Balanced S-Parameters 40
9.2.2 Topology 42
9.2.3 Imbalance/CMRR 45
9.3 Meas Setup 47
9.4 TDR Measurement (Option) 48
9.4.1 Functional Interface Introduction 49
9.4.2 Parameter Setup 50
9.4.3 Trace Setup 56
9.4.4 Eye Diagram/Mask Function 59
10 Data Format 66
10.1 Rectangular Display Formats 66
10.2 Polar 68
10.3 Smith 69
10.4 Inverted Smith 69
11 Marker 70
11.1 Marker 1-7/Marker 8-15 70
11.2 Marker Setup 72
11.3 Marker Display 73
11.4 Marker Functions 74
12 Search 75
12.1 Main 75
12.2 Peak 76
12.3 Target 78
12.4 Multi Peak & Target 78
12.5 Band-Pass/Band-Reject 80
12.5.1 Bandwidth Search 80
12.5.2 Band-Reject Search 80

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

13 Calibration 81
13.1 Main 82
13.1.1 Basic Calibration 82
13.1.2 Isolation Calibration 84
13.1.3 Electronic Calibration 85
13.1.4 Cal All 86
13.1.5 Correction/Correction Properties 88
13.1.6 Calibration Kits 89
13.1.7 To Save or Recall the CalSet File 95
13.1.8 Interpolation 96
13.2 Port Extension 98
13.3 Auto Port Extensions 100
13.4 Fixture 103
13.4.1 Port Match 104
13.4.2 Port Z 104
13.4.3 2-port DeEmbed 105
14 Power 107
15 Avg BW 107
15.1 Main 107
15.2 Smoothing 108
15.3 Group Delay Aperture 109
16 Trigger 110
16.1 Main 110
16.2 Input 112
16.3 Output 112
17 Scale 113
17.1 Main 113
17.2 Electrical Delay 114
17.3 Constants 114
18 Math Operation 115
18.1 Memory 115
18.2 Analysis 116
18.3 Limit Test 118
18.3.1 To Enable or Disable the Limit Test 119
18.3.2 Limit Table 119
18.3.3 Global Pass/Fail 120
18.4 DTF (Option) 121
18.5 Time Domain (TDA Option) 124
18.6 Time Gating (TDA Option) 125

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

18.7 Time Domain Setup (TDA Option) 127
18.8 Skew Measure (TDA Option) 129
18.9 Delay Measure (TDA Option) 129
18.10 Automated Test (TAS Option) 130
19 Setup 130
19.1 Layout 130
19.2 Sweep Setup 131
19.3 S-Parameter 131
19.4 Display Setup 131
19.5 Save 132
19.6 Recall 133
19.7 File 134
20 Save & Recall 136
21 System 136
21.1 I/O Setting 136
21.2 Basic Settings 139
21.3 About this Instrument 139
21.4 Options 140
21.5 SMB 140
22 Remote Control 141
23 Preset 142
24 Troubleshooting 148
25 Appendix 150
25.1 Appendix A: Options and Accessories 150
25.2 Appendix B: Warranty 150

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

1 Document Overview

This document is intended to provide a quick overview of the front and rear panels, user interface and basic operation of the DNA series vector network analyzer.

Tip:

For the latest version of this manual, download it from the official website of RIGOL (http://www.rigol.com).

Publication Number

UGR05100-1110

Software Version

00.00.11

Software upgrade might change or add product features. Please obtain the latest version of the manual from RIGOL website or contact RIGOL to upgrade the software.

Format Conventions in this Manual

1. **Key**

   The front panel key is denoted by the menu key icon. For example, 

indicates the "System" key.

2. **Menu**

   The menu item is denoted by the format of "Menu Name (Bold) + Character Shading" in the manual. For example, **Frequency** indicates clicking **Frequency** to enter the frequency setting menu.

3. **Operation Procedures**

   The next step of the operation is denoted by "**>**" in the manual. For example, **Frequency > Center** indicates first clicking **Frequency**, and then clicking **Center**.

Content Conventions in this Manual

The DNA series vector network analyzer includes the following models. Unless otherwise specified, this manual takes DNA6264 as an example to illustrate the functions and operation methods of the DNA series.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

2 DNA Series Product Models

2.1 DNA6000 Series

Model	Frequency	Number of Ports	Port Connector
DNA6082	5 kHz ~ 8.5 GHz	2	Type N Female
DNA6084	5 kHz ~ 8.5 GHz	4	Type N Female
DNA6142	5 kHz ~ 14 GHz	2	Type N Female
DNA6144	5 kHz ~ 14 GHz	4	Type N Female
DNA6202	5 kHz ~ 20 GHz	2	3.5 mm Male (Secured Type)
DNA6204	5 kHz ~ 20 GHz	4	3.5 mm Male (Secured Type)
DNA6262	5 kHz ~ 26.5 GHz	2	3.5 mm Male (Secured Type)
DNA6264	5 kHz ~ 26.5 GHz	4	3.5 mm Male (Secured Type)

2.2 DNA6000-R Series

Model	Frequency	Number of Ports	Port Connector
DNA6082-R	5 kHz ~ 8.5 GHz	2	Type N Female
DNA6084-R	5 kHz ~ 8.5 GHz	4	Type N Female
DNA6142-R	5 kHz ~ 14 GHz	2	Type N Female
DNA6144-R	5 kHz ~ 14 GHz	4	Type N Female
DNA6202-R	5 kHz ~ 20 GHz	2	3.5 mm Male (Secured Type)
DNA6204-R	5 kHz ~ 20 GHz	4	3.5 mm Male (Secured Type)
DNA6262-R	5 kHz ~ 26.5 GHz	2	3.5 mm Male (Secured Type)
DNA6264-R	5 kHz ~ 26.5 GHz	4	3.5 mm Male (Secured Type)

2.3 DNA5000 Series

Model	Frequency	Number of Ports	Port Connector
DNA5042	5 kHz ~ 4.5 GHz	2	Type N Female
DNA5082	5 kHz ~ 8.5 GHz	2	Type N Female
DNA5142	5 kHz ~ 14 GHz	2	Type N Female

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Model	Frequency	Number of Ports	Port Connector
DNA5202	5 kHz ~ 20 GHz	2	3.5 mm Male (Secured Type)
DNA5262	5 kHz ~ 26.5 GHz	2	3.5 mm Male (Secured Type)

2.4 DNA800 Series

Model	Frequency	Number of Ports	Port Connector
DNA804	5 kHz ~ 4.5 GHz	2	Type N Female
DNA808	5 kHz ~ 8.5 GHz	2	Type N Female
DNA814	5 kHz ~ 14 GHz	2	Type N Female

3 Safety Requirement

3.1 General Safety Summary

Please review the following safety precautions carefully before putting the instrument into operation so as to avoid any personal injury or damage to the instrument and any product connected to it. To prevent potential hazards, please follow the instructions specified in this manual to use the instrument properly.

- **Use Proper Power Cord.**

  Only the dedicated power cord designed for the instrument and authorized for use within the local country should be used.

- **Ground the Instrument.**

  The instrument is grounded through the Protective Earth lead of the power cord. To avoid electric shock, it is essential to connect the earth terminal of the power cord to the Protective Earth terminal before connecting any inputs or outputs.

- **Observe All Terminal Ratings.**

  To avoid fire or shock hazard, observe all ratings and markers on the instrument and check your manual for more information about ratings before connecting the instrument.

- **Use Proper Overvoltage Protection.**

  Ensure that no overvoltage (such as that caused by a bolt of lightning) can reach the product. Otherwise, the operator might be exposed to the danger of an electric shock.

- **Do Not Operate Without Covers.**

  Do not operate the instrument with covers or panels removed.

- **Do Not Insert Objects Into the Air Outlet.**

  Do not insert anything into the holes of the fan to avoid damaging the instrument.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Use Proper Fuse.**

  Please use the specified fuses.

- **Avoid Circuit or Wire Exposure.**

  Do not touch exposed junctions and components when the unit is powered on.

- **Do Not Operate With Suspected Failures.**

  If you suspect damage occurs to the instrument, have it inspected by RIGOL authorized personnel before further operations. Any maintenance, adjustment or replacement especially to circuits or accessories must be performed by RIGOL authorized personnel.

- **Provide Adequate Ventilation.**

  Inadequate ventilation may cause an increase of temperature in the instrument, which would cause damage to the instrument. So please keep the instrument well ventilated and inspect the air outlet and the fan regularly.

- **Do Not Operate in Wet Conditions.**

  To avoid short circuit inside the instrument or electric shock, never operate the instrument in a humid environment.

- **Do Not Operate in an Explosive Atmosphere.**

  To avoid personal injuries or damage to the instrument, never operate the instrument in an explosive atmosphere.

- **Keep Instrument Surfaces Clean and Dry.**

  To avoid dust or moisture from affecting the performance of the instrument, keep the surfaces of the instrument clean and dry.

- **Prevent Electrostatic Impact.**

  Operate the instrument in an electrostatic discharge protective environment to avoid damage induced by static discharges. Always ground both the internal and external conductors of cables to release static before making connections.

- **Use the Battery Properly.**

  Do not expose the battery (if available) to high temperature or fire. Keep it out of the reach of children. Improper change of a battery (lithium battery) may cause an explosion. Use the RIGOL specified battery only.

- **Handle with Caution.**

  Please handle with care during transportation to avoid damage to keys, knobs, interfaces, and other parts on the panels.



Warning

Equipment meeting Class A requirements may not offer adequate protection to broadcast services within residential environment.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

3.2 Safety Notices and Symbols

Safety Notices in this Manual:



Warning

Indicates a potentially hazardous situation or practice which, if not avoided, will result in serious injury or death.



CAUTION

Indicates a potentially hazardous situation or practice which, if not avoided, could result in damage to the product or loss of important data.

Safety Notices on the Product:

- **DANGER**

  It calls attention to an operation which, if not correctly performed, could result in immediate injury or hazard.

- **WARNING**

  It calls attention to an operation which, if not correctly performed, could result in potential injury or hazard.

- **CAUTION**

  It calls attention to an operation which, if not correctly performed, could result in damage to the product or other devices connected to the product.

Safety Symbols on the Product:



Hazardous Voltage



Safety Warning



Protective Earth Terminal



Chassis Ground



Test Ground

3.3 EMC Level

Class A (for non-residential products)

NOTE:

This equipment has been tested and found to comply with the limits for a Class A digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment. This equipment generates, uses, and can radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the user will be required to correct the interference at his own expense.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

3.4 Ventilation Requirement

This instrument uses a fan to force cooling. Please make sure that the air inlet and outlet areas are free from obstructions and have free air. When using the instrument in a bench-top or rack setting, provide at least 10 cm clearance beside, above and behind the instrument for adequate ventilation.



CAUTION

Inadequate ventilation may cause an increase of temperature in the instrument, which would cause damage to the instrument. So please keep the instrument well ventilated and inspect the air outlet and the fan regularly.

3.5 Working Environment

Temperature

Operating: 0℃ to +40℃

Non-operating: -20℃ to +60℃

Humidity

- **Operating:**

  Below +30℃: ≤95% RH (without condensation)

  +30℃ to +40℃: ≤75% RH (without condensation)

- **Non-operating:**

  Below +40℃: 5% to 90%, without condensation

  +40℃ to +60℃: 5% to 80%, without condensation



Warning

To avoid short circuit inside the instrument or electric shock, never operate the instrument in a humid environment.

Altitude

Operating altitude below 2,000 m (6,561.68 feet)

Protection Level Against Electric Shock

ESD ±8kV

Installation (Overvoltage) Category

This product is powered by a power adapter conforming to installation (overvoltage) category II.



Warning

Ensure that no overvoltage (such as that caused by lightning) reaches this product. Otherwise, the operator may be exposed to the danger of electric shock.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Installation (Overvoltage) Category Definitions

Installation (overvoltage) category I refers to signal level which is applicable to equipment measurement terminals connected to the source circuit. Among these terminals, precautions are taken to limit the transient voltage to a low level.

Installation (overvoltage) category II refers to the local power distribution level which is applicable to equipment connected to the AC line (AC power).

Pollution Degree

Pollution Degree 2

Pollution Degree Definition

- **Pollution Degree 1:** No pollution or only dry, nonconductive pollution occurs. The pollution has no effect. For example, a clean room or air-conditioned office environment.
- **Pollution Degree 2:** Normally only nonconductive pollution occurs. Temporary conductivity caused by condensation is to be expected. For example, indoor environment.
- **Pollution Degree 3:** Conductive pollution or dry nonconductive pollution that becomes conductive due to condensation occurs. For example, sheltered outdoor environment.
- **Pollution Degree 4:** The pollution generates persistent conductivity caused by conductive dust, rain, or snow. For example, outdoor areas.

Safety Class

Class III product

3.6 Care and Cleaning

Care

Do not store or leave the instrument where it may be exposed to direct sunlight for long periods of time.

Cleaning

Clean the instrument regularly according to its operating conditions.

1. Disconnect the instrument from all power sources.
2. Clean the external surfaces of the instrument with a soft cloth dampened with mild detergent or water. Avoid letting any water or other objects enter the chassis via the heat dissipation holes. When cleaning the LCD, take care to avoid scratching it.



CAUTION

To avoid damage to the instrument, do not expose it to caustic liquids.



Warning

To avoid short-circuit resulting from moisture or personal injuries, ensure that the instrument is completely dry before connecting it to the power supply.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

3.7 Environmental Considerations

The following symbol indicates that this product complies with the WEEE Directive 2012/19/EU.



The equipment may contain substances that could be harmful to the environment or human health. To avoid the release of such substances into the environment and avoid harm to human health, we recommend you to recycle this product appropriately to ensure that most materials are reused or recycled properly. Please contact your local authorities for disposal or recycling information.

You can click on the following link https://www.rigol.com/intl/services/environmental-protection-statement.html to download the latest version of the RoHS&WEEE certification file.

4 Quick Start

4.1 General Inspection

1. **Inspect the packaging**

   If the packaging has been damaged, do not dispose of the damaged packaging or cushioning materials until the shipment has been checked for completeness and has passed both electrical and mechanical tests.

   The consigner or carrier shall be liable for the damage to the instrument resulting from shipment. RIGOL would not be responsible for free maintenance/rework or replacement of the instrument.

2. **Inspect the instrument**

   In case of any mechanical damage, missing parts, or failure in passing the electrical and mechanical tests, contact your RIGOL sales representative.

3. **Check the accessories**

   Please check the accessories according to the packing lists. If the accessories are damaged or incomplete, please contact your RIGOL sales representative.

Recommended Calibration Interval

RIGOL suggests that the instrument should be calibrated every 18 months.

4.2 Product Overview

The DNA series vector network analyzer offers multiple calibration methods including frequency response, single port, response isolation, enhanced response, full two port, and electrical calibration. It supports various data display formats including log magnitude, linear magnitude, SWR (Standing Wave Ratio), phase, group delay, Smith chart, polar plot, etc. Equipped with standard USB, LAN, and HDMI interfaces, the instrument accurately measures the magnitude-frequency, phase-frequency, and group delay characteristics of microwave networks. This product can be widely applied in fields such as electronics,

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

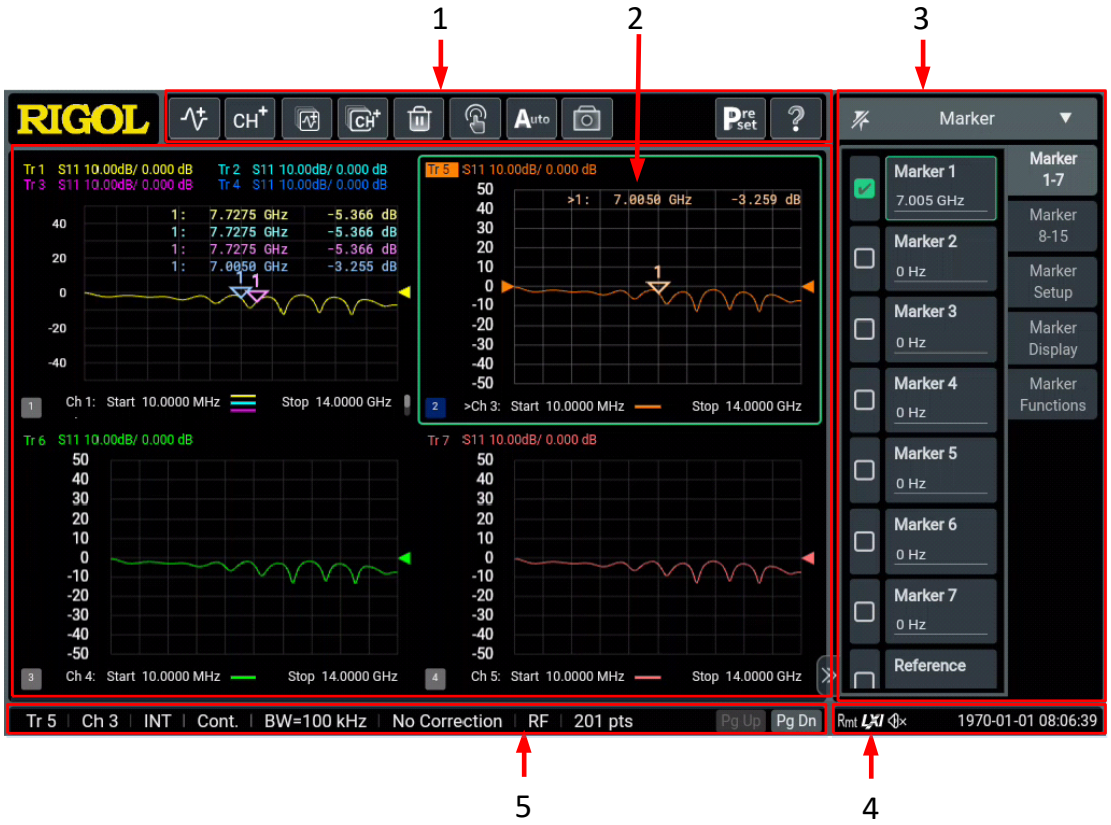
communication and microwave. It is commonly used test equipment in the research and production processes of industries and universities.

4.2.1 Product Appearance

The product appearance introduction includes descriptions of the front and rear panels; please refer to the corresponding Quick Guide manual for the product.

Product Series	Quick Guide
DNA6000 Series	DNA6000 Quick Guide
DNA5000 Series	DNA5000 Quick Guide
DNA6000-R Series	DNA6000-R Quick Guide
DNA800 Series	DNA800 Quick Guide

4.2.2 User Interface



No.	Description
1	Quick Operation Toolbar
2	Window Display Area
3	Function Menu Area

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

No.	Description
4	Notification Area
5	System Status Bar

4.2.2.1 Quick Operation Toolbar

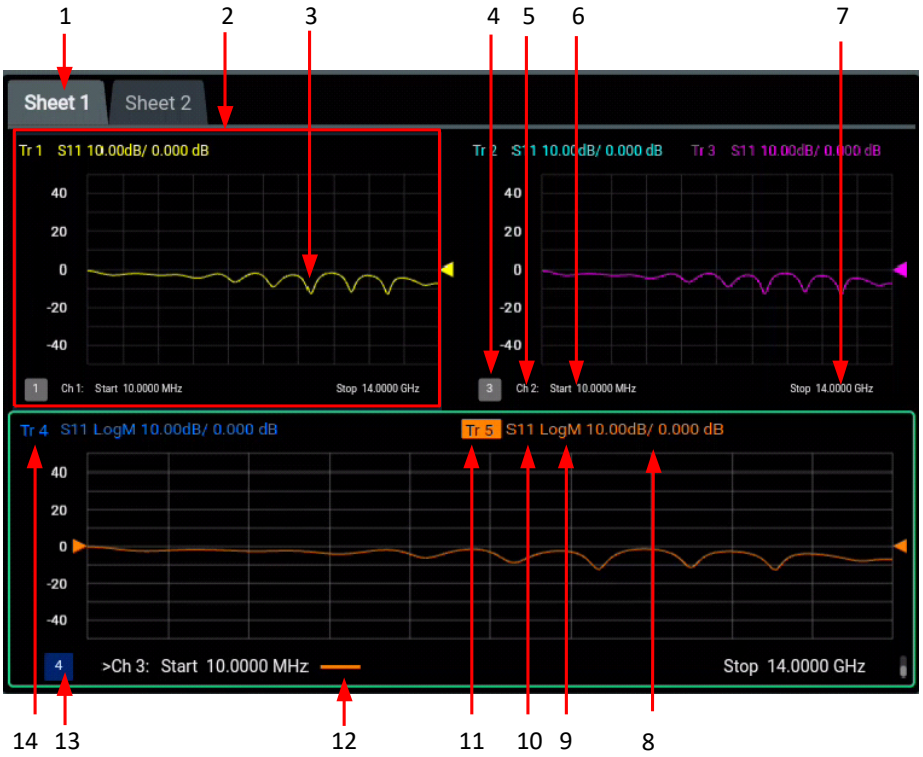
The following figure shows the quick operation toolbar, and the table below it lists the functions of the quick operation keys on the toolbar.



No.	Description
1	Adds a new trace (Trace)
2	Adds a new channel (Channel)
3	Adds a new window (Window) + trace (Trace)
4	Adds a new window (Window) + channel (Channel)
5	● Delete shortcut: Deletes the active trace by default; ● If the current window is empty, the active window will be deleted; ● If the current sheet is empty, the active sheet will be deleted; ● The last sheet cannot be deleted.
6	Manual trigger key.
7	Shortcut key for Auto Scale. Adjusts the parameters to the optimal state of the signal.
8	Saves the screenshot of the current screen.
9	Restores the system settings to the preset state, the same as the front-panel Preset key.
10	Displays the help document.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

4.2.2.2 Window Display Area



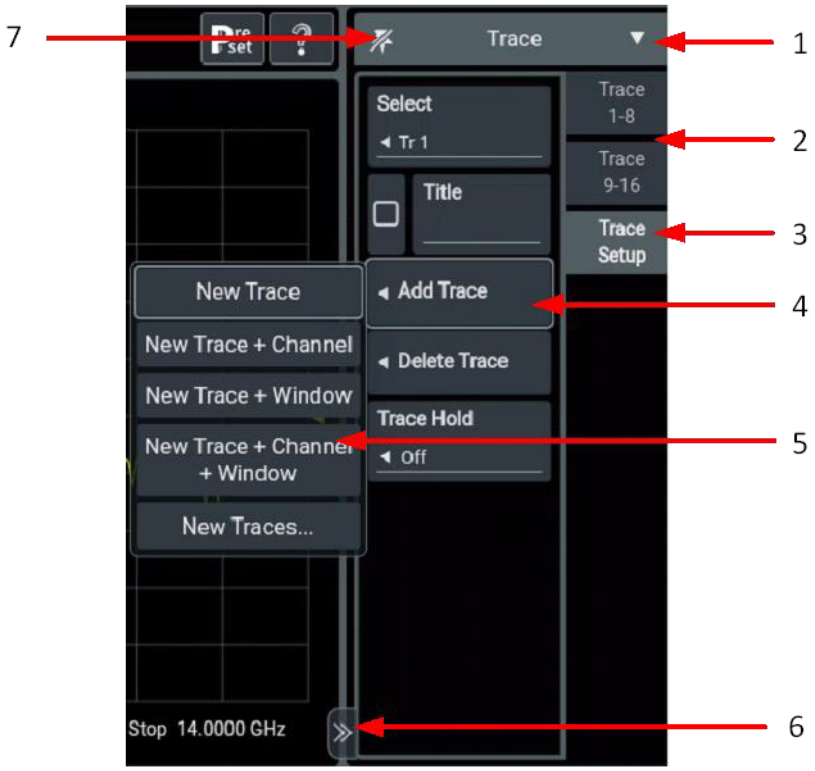
No.	Description
1	When there is only one sheet, the sheet identifier is not displayed. Click the identifier to switch or rename sheets. Each sheet can contain multiple windows.
2	● Window: It is used to display traces. The definitions of the horizontal axis and vertical axis are determined by measurement parameters and display formats. ● By default, a sheet can display a maximum of 4 windows. If more than 4 windows are added, you can switch pages by clicking the Previous Page and Next Page buttons at the bottom right of the screen. The maximum number of display windows per sheet is configurable, with optional values of 1, 2, 4, 6 and 9. ● Single-click to activate a selected window; double-click the window area to maximize the window, and double-click again to restore the multi-window layout. ● Drag the Y-axis of the window to modify the reference level value.
3	Trace, a set of measurement data points. Click to select a trace as active.
4	Window Number (Unselected State)
5	Channel Number
6	Start Value of Sweep Range
7	Stop Value of Sweep Range
8	Scale value/ reference level (related to the measurement data display format)
9	Trace Data Format; LogM shown in the figure refers to Logarithmic Magnitude.
10	Trace Title, which displays the measured S-parameter by default.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

No.	Description
11	Trace Number (Selected State)
12	Trace identifier under the channel; the identifier color is consistent with the corresponding trace color.
13	Window Number (Selected State)
14	Trace Number (Unselected State)

4.2.2.3 Function Menu Area

Enter the Function Menu to make the settings.



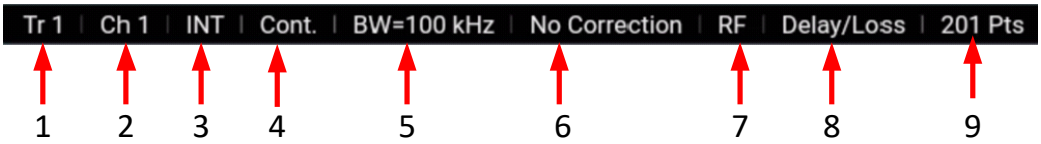
No.	Description
1	Function menu name. Click the drop-down button on the right to switch to another function menu.
2	Level-2 function menu (deselected state)
3	Level-2 function menu (selected state)
4	Level-3 function menu. Click < in the button to display the next-level menu

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

No.	Description
5	Level-4 function menu
6	Show/hide button for the function menu /
7	Function menu pin button[1]: unlocked by default. Once locked, the function menu remains open and will not auto-close.

4.2.2.4 System Status Bar

Displays the main parameters of the current system status.



1. Number of the currently selected trace
2. Number of the currently selected channel
3. Trigger source. Internal trigger is shown here and can be configured in the Trigger menu.
4. Trigger mode. Continuous trigger is shown here and can be configured in the Trigger menu.
5. IF bandwidth, which can be set in the Average and Bandwidth menu.
6. Calibration status. After calibration, the calibration type is displayed; when not calibrated, NO Correction is displayed.
7. Internal stimulus source status, which indicates the RF power enable status of all channels. The icon turns gray when disabled.
8. Enable status of the port extension function. Delay refers to time delay compensation, and Loss refers to loss compensation. This item will not be displayed when the port extension function is disabled.
9. Number of sweep points, which can be configured in the sweep menu.

4.2.2.5 Notification Area

Displays LAN icon, sound icon, and remote control icon as well as date and time. You can click this area to open the "System" menu.

1 Only the DNA800 series supports this function.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



No.	Description
1	Remote control icon. When you use Web Control to control the instrument remotely, Rmt will be displayed.
	LAN icon. When the LAN interface is successfully connected, LXI is displayed.
	Sound icon. Click on this icon to turn on or off the sound. When enabled, will be displayed; when disabled, will be displayed.
	USB storage device icon: When a USB storage device is detected, will be displayed.
	External Clock Icon: When you use the external 10 MHz clock signal Ext will be displayed.
2	Date: displays the system date. You can set it in the **Setting** sub-menu under **System**.
3	Time: displays the system time. You can set it in the **Setting** sub-menu under **System**.

4.3 To Prepare for Use

4.3.1 To Connect to Power

Refer to the product-specific Quick Start Guide for details.

4.3.2 Turn-on Checkout

- Power on
  - After connecting the instrument to the power source properly, press the power key on the lower-left corner of the front panel to power on the instrument. During the start-up process, the instrument performs a series of self-tests. After the self-test, the splash screen is displayed.
  - Click the Notification Area and select **Setting** > **Power Switch** in the displayed **System** menu. By default, the **Power Switch** menu is enabled. The instrument will be powered on automatically after the instrument is connected to power.
- **Shutdown**
  - Press the power key and a prompt message "Are you sure to shutdown?" is displayed. Click **OK** to shut down the instrument.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- Press O twice to directly shut down the instrument.
- Press O for three seconds to directly shut down the instrument.

4.3.3 To Set the System Language

This series supports multiple system languages. To select the desired language, click the notification area at the lower-right corner of the screen to select **Setting > Language**.

4.4 Basic Operation

This chapter introduces the basic operation of the instrument.

4.4.1 Mouse / Keyboard / Touchscreen Operation Rules

Connect Display and Control Devices[2]

Control can be implemented in two ways: via an external display, mouse and keyboard, or remotely through Web Control.

Mouse Operation Rule

Connect the mouse to the instrument via the USB HOST interface to perform the following operations. Note that you can only use the left mouse button to perform the left-click operation. Right-click and mouse scrolling operation are not allowed.

1. Click the mouse to select the menu and window.
2. Press and hold the left mouse button to drag the displayed data or window.
3. In the Marker menu, click the mouse to move the marker, but you cannot use the mouse to add a marker.

Keypad Board Operation Rule

After the keypad board is properly connected to the instrument via the USB HOST interface, you can use the shortcut keys on the keypad to perform the same function as what you do with the Function key.

Touchscreen Operation Rules

The front panel of this instrument is a multi-touch screen and supports the following gesture operations.

- **Touch Gesture:** Tap the icons or text on the screen gently with a single finger.
  - Tap menus and function keys to configure functions.
  - Tap the input box to pop up the virtual keyboard for parameter configuration.
  - Tap windows and their tab bars to perform relevant window operations.
- **Drag Gesture:** Press and hold the target with a single finger and drag.

2 The DNA6000-R series vector network analyzer is not equipped with a display screen. Functions can only be configured and measurement results viewed by using peripheral devices.

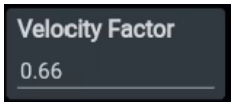
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- Change the position of windows or cursors.
- Drag slider controls in menus to adjust function settings.

4.4.2 Menu Operation

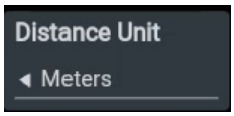
The menu operation includes the following types.

1 Input a Parameter Value



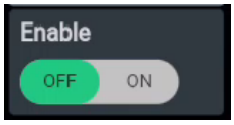
Click the specified menu, then use the numeric keys to modify the value directly.

2 **Select a Parameter**



Click the specified menu, its sub-menus are displayed. Select the desired parameter from the drop-down list.

3 **Switch between the Option Tabs**



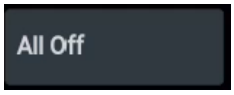
Click the desired tab to switch between the alternative tabs.

4 **Select a Sub-menu**



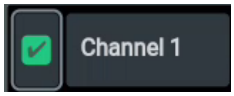
Click the specified menu with the cascading sub-menus to select the specified sub-menu.

5 **Direct Execution**



Click the specified menu to execute the corresponding function.

6 **Check to Enable the Function (without parameter)**



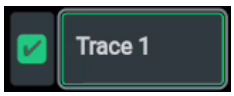
Check the checkbox of the specified menu to enable the function. If the checkbox turns green, with a tick in it, this function is enabled.

7 **Check to Enable the Function (with parameter)**



Check the checkbox of the specified menu to enable the function. Then click the parameter input field to input the value or characters to modify the parameter.
Check the checkbox of the specified menu to enable the function. If the checkbox turns green, with a tick in it, this function is enabled.

8 **Menu Selected**



Click the specified menu, and the selected menu is highlighted in a green box. You can click the desired marker, trace, channel, window, or sheet to make them be selected.

Tip:

The above menu operations can also be realized through externally connected mouse, touch screen operation, or Web Control operation.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

4.4.3 Parameter Setting

Parameter input can be completed via the front-panel numeric keypad, knob, or arrow keys, or via the touch screen or an external keyboard or mouse. Parameters can be set in the following ways.

1. Numeric keypad:

When an input box is selected, use the front-panel numeric keypad to set parameters.

2. Use the front-panel knob

When a parameter is editable, rotate the knob to increase (clockwise) or decrease (counterclockwise) the parameter by the specified step.

3. Arrow keys

When the parameter is in editing mode, use the arrow keys to increase or decrease

the parameter value at the specified step. Note that the step sizes for the Up/

Down arrow key and the Left/Right arrow key are different.

4. Touchscreen

Tap the touch screen to select an input box, and use the pop-up virtual keyboard to set parameters.

5. External keyboard

When an input box is selected, use an external keyboard to set parameters.

6. External mouse

Use an external mouse to select a parameter input box, and use the pop-up virtual keyboard to set parameters.

4.4.4 To Use the Built-in Help System

The built-in help document of the vector network analyzer provides information about the functions and menu introductions of the instrument.

Click on the help icon on the quick operation toolbar at the top of the main interface, then the Help documentation is displayed. You can get its help information by clicking on the link for the introduction of the specified function.

4.4.5 To View the Option and the Option Installation

This series vector network analyzer provides many options to meet various measurement requirements. If you need any of these options, order them according to the Order No. available in Data Sheet, and then install the options according to this section. Besides, you can also view the options currently installed on the vector network analyzer and activate the newly purchased option.

1. **View the Installed Option**

   If your instrument has currently installed the option, perform the following operations to view the name of the installed option and other detailed information about the option from the option list.

   Click the notification area in the lower-right corner of the screen to pop up the system menu. Click **Options** to view the options currently installed.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

2. **Install the Option**

   The option license is a string with a fixed number of characters. Each instrument has one unique license. The license file should be in specific format, with the filename extension "*.lic". After you purchase an option, you will obtain a key (used for obtaining the license). Then, you can install the option according to the following steps.

   a. **Obtain an option license**

      Log in to the **RIGOL** official website (http://www.rigol.com), click **SERVICE CENTER > LICENSE ACTIVATION** to enter the license activation interface.

      Input the correct key, serial number (click **About** to acquire the serial number of the instrument), and verification code. Click **Generate** to acquire the download link for the option license file.

   b. **Install the option**

      Install the option by sending SCPI commands. For details, refer to *Programming Guide* of the product. After installation, a prompt message "Option activated successfully" is displayed. After the option has been installed, you are recommended to restart the instrument.

Tip:

- During the installation process, you are not allowed to power off the instrument.
- Installing options by sending SCPI commands is supported. Installing options by inputting the license code manually is not supported.

4.5 Basic Test Procedures

The following procedures guide you how to make the measurement settings for the analyzer.

1. **Set the Measurement**

   Sets the analyzer, creates the measurement, and adjusts the measurement display.

   a. Use the preset settings or user-defined settings of the analyzer: you can preset the analyzer (Preset) or recall (Recall) the user-defined settings.

   b. Create a channel (Channel) and a trace (Trace).

   c. Select a measurement parameter (Measure).

   d. Set the sweep type and points (Sweep).

   e. Set the frequency span (Frequency).

   f. Set the port power (Power).

   g. Set the trigger (Trigger).

   h. Set the data format (Data Format).

   i. Set the scale (Scale).

   j. Set the user-defined window display (Window Setup and Sheet Setup).

2. **Optimize the Measurement**

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

A measurement has many interdependent settings. You can modify the settings to achieve faster processing or greater measurement accuracy.

Achieve Faster Processing

- Fast sweep (Sweep)

  Use the sweep frequency span required for the measurement device

  Use the minimum number of points required for the measurement

  Segment sweep: use segments to focus test data only where you need it.

  Auto sweep time: use this default to sweep as quickly as possible for the current settings.

- Disable the unnecessary functions, including Trace, Marker, Smoothing, Limit Test, etc.

- Use SCPI commands to remotely control (Remote Control) the VNA via LAN interface to perform automatic measurements.

Improve Measurement Accuracy

- Increase dynamic range
- Reduce noise, including Averaging State/Averaging Times, IF BW, and Smoothing.
- Group Delay (Group Delay Aperture)
- Set the electrical delay (Electrical Delay), phase offset (Phase Offset), and port extension (Port Extension) to improve the phase measurement accuracy.

3. **Perform the Measurement Calibration**

Performs the calibration to reduce the measurement errors.

- Basic calibration: Selects the calibration type (S-parameter Calibration Type Supported by VNA) and calibration kits (Calibration Kits). You can select the available calibration kits from the standard calibration kits, and can also load the user-defined calibration kit (To Save or Recall the CalSet File). You can also load the user-defined calibration sets file to recall the calibration data. Note that the calibration sets file you recalled shall match the current sweep settings.
- Electronic calibration: You can select the desired electronic calibration kit (Electronic Calibration) based on the selected calibration type and port to perform the calibration.
- Advanced calibration function: Port extension (Port Extension), port match (Port Match), port Z (Port Z), and 2-port DeEmbed (2-port DeEmbed).

4. **Analyze Data**

Analyzes the measurement results with marker, math operations, and limit tests.

- Marker function provides the readout of measured data, searches for specific values (Search), and modifies the stimulus settings. Each trace has 15 normal markers and one reference marker (used with Delta marker).
- Use the data math operation (Memory) to perform four types of math operations on the active trace and memory trace.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- The limit line measurement function (Limit Test) allows you to compare the measurement data with the user-defined conditions.

5. **Save and Recall Data**

   Saves (Save) and recalls (Recall) measurement data files to and from an internal or external storage device in various formats. The file formats include:

   - Instrument state and calibration data (*.csa, *.sta, and *.cal)
   - Measured data file (*.snp and *csv)

5 Trace

Traces are a series of measured data points. The data points in one measurement share the same trace number and are saved as one trace.

- Each sheet supports up to 100 traces. Up to 500 windows are supported.
- One channel supports 16 traces.

The trace state information is displayed at the top of each window. The trace number highlighted indicates the current active trace. For details, refer to User Interface.

The trace setup will affect the presentation and math operations of the measurement data. The following lists the trace related items for trace setup.

- Measurement Parameter, Parameter Conversion
- Format and Scale
- Averaging, Smoothing
- Correction ON/OFF
- Electrical Delay
- Phase Offset, Amplitude Offset
- Trace Math
- Markers
- TDA, DTF

5.1 Trace 1-8/Trace 9-16

Creates and selects the desired trace quickly. Click the checkbox of the specified trace to select it. Uncheck the checkbox of the specified trace to remove the trace.

Create the Trace

Click the checkbox of the specified trace from the **Trace 1-8** and **Trace 9-16** menus to create the trace quickly. When the trace is available on the screen, the checkbox ahead of the specified trace is

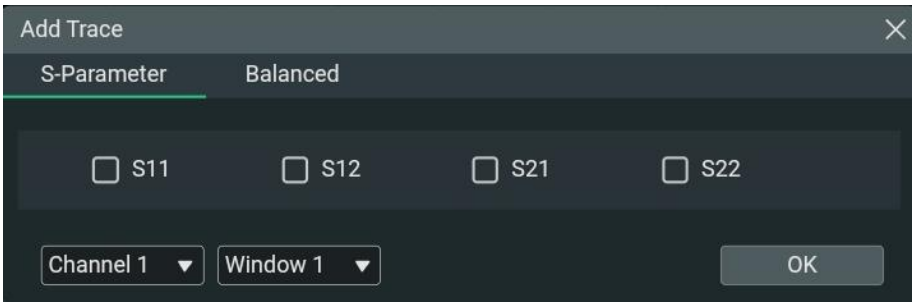
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

highlighted in green. When the specified trace is selected, the trace name is highlighted in green box.

After the trace is created, the trace is available in the current active window and the current active channel. S11 parameter is measured by default.

Add a Trace

Click **New Traces...**, then the trace adding interface is displayed.



Select the window, channel, and the measurement parameter for the trace. If there is no available channel or window to choose, you can add a new channel or window first, then create the trace for the channel and the window.

5.2 Trace Setup

In the Trace Setup menu, you can select the specified trace, add or delete the trace, modify the trace title, and set the trace hold.

Select a Trace

Select a trace as the current active trace. Click the drop-down button of "Select" to select the desired trace. Select the desired trace in the "Trace 1-8" or "Trace 9-16" menu. You can also click the target trace label at the upper-left corner of the current window to select the desired trace. To modify the trace title, first select the trace to make it active, then modify the title.

Modify the Trace Title

Select the specified trace and then modify the trace title. You can also select to display or hide the trace title.

- By default, the checkbox of Title is unchecked. The default trace title represents the measured S-parameter.
- To modify the trace title, check the checkbox of the **Title** menu, and then click the input field of the title to input the trace title name.

Add a Trace

To add a trace, perform the following operations.

- **New Trace:** adds a trace in the current active window. The trace is added on the active channel in the current active window. The trace uses the default properties.
- **New Trace + Channel:** creates a new channel in the current active window, and creates a new

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

trace for channel. The trace uses the default properties.

- **New Trace + Window:** creates a new window, and creates a new trace in the new window. The newly created trace belongs to the active channel.
- **New Trace + Channel + Window:** creates a new window, and creates a new channel in the current active window. Create a new trace for the channel. The trace uses the default properties.
- **New Traces...:** click **New Traces...**, then the trace adding interface is displayed. For detailed configurations, refer to Add a Trace.

Delete a Trace

Click **Delete Trace**, then the drop-down list of all the available traces is displayed. Click the specified trace to delete the trace.

Trace Hold

When Max or Min is selected for Trace Hold, hold the maximum and minimum response values at the stimulus values, helping users make extrema analysis.

- **Off:** By default, the trace hold function is disabled.
- **Max:** the trace keeps updating, and if the new response value is greater than the current max. hold value, then the new max. value prevails. The displayed trace always represents the max. response value at each frequency point since the max. trace hold is enabled.
- **Min:** the trace keeps updating, and if the new response value is smaller than the current hold value, then the new min. value prevails. The displayed trace always represents the minimum response value at each frequency point since the min trace hold is enabled.
- **Restart:** resets the trace data and restarts to launch the measurement based on the currently selected trace hold type.

6 Channel

In the **Channel** menu, you can configure the settings for the channel, window, and sheet.

The features of channels in the VNA:

1. Signal transmission: Each channel is responsible for sending RF signal from the test port to the DUT and receiving the reflection signal for measurement and analysis.
2. Independent: Each channel is independent and can be configured and controlled independently. Therefore, the instrument can make multiple measurements at the same time or calibrate on different ports.
3. Flexible to measure: Multiple channels are supported, enabling you to measure the S-parameters of different ports and make a comprehensive analysis for the multi-port network.
4. Calibration and correction: Each channel can be calibrated and corrected independently, ensuring the accuracy and reliability of the test results.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

5. Data processing: Each channel receives and processes the signal sent from the DUT for data acquisition, analysis, and display.

6.1 Channel 1-9

Channel determines how to measure and process signals in the VNA. Each channel generally consists of a transmit port and a receiver port, which are used to transmit and receive RF signal, then make signal analysis and measurement. Each channel has a unique number[3].

The channel settings determine how the trace data is measured. All the traces that are assigned to a channel share the same channel settings. To modify the channel setting, first select the channel.

Click **Channel 1-9** to enter its sub-menu. 9 channel menus are displayed. If the checkbox at the left part of the specified channel name is checked and highlighted in green, it indicates that the channel is available. If the channel name is highlighted in green box, it indicates that the channel is selected. By default, Channel 1 is selected.

- If a channel does not exist, check the checkbox of the specified channel name to create the channel quickly. If a channel already exists on the screen, uncheck the checkbox of the specified channel name to delete the channel quickly.
- After deleting the channel, all the traces on the channel will all be deleted.
- Each time you create a channel, a trace (S11 logM trace by default) is automatically created. A channel without trace does not exist.
- A window can contain no channels or traces. Even if you delete all the channels and traces from the window, the window without any channels and traces can still exist.

6.2 Channel Setup

Select a Channel

Click the drop-down button of **Select**, then all the available channels are displayed to choose. Choose the desired channel to be the selected channel. If you select one trace, then the channel where the trace is located is also selected.

If you need to modify the parameters of the specified channel, first select the channel.

Add a Channel

Click the drop-down button of **Add Channel** to add the desired channel.

- **New Trace + Channel:** creates a new channel, and creates a new trace for the new channel in the current active window.
- **New Trace + Channel + Window:** creates a new window and a new channel; creates a new trace for the new channel in the new window.

3 The DNA800 series supports a maximum of 250 independent channels. Other series supports a maximum of 500 independent channels.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

The newly created window, channel, and trace are automatically selected.

New IDs with the minimal number that is not in use will be assigned to the newly created window, channel, and trace

Delete a Channel

Click **Delete Channel**, then all the existing channels are displayed in the pop-up sub-menu, select the desired channel to delete it.

Once the channel is deleted, all the traces contained in the channel will all be deleted.

If the currently selected channel is deleted, the channel with the smaller ID number will become the newly selected channel.

6.3 Window 1-9

Windows are used to view traces. Each sheet can contain up to 100 traces. The total number of traces in all the windows in the current sheet cannot exceed 100. The instrument supports creating 500 windows, and each window has a unique ID, which is indicated at the lower-left corner of the window. At most, 9 windows can be displayed on the screen at the same time. If new windows are added, click **Pg Dn** to view more windows.

Window Management

Click **Window 1-9** to enter its sub-menu. 9 window menus are displayed. If the checkbox at the left part of the specified window name is checked and highlighted in green, it indicates that the window is available. If the window name is highlighted in green box, it indicates that the window is selected.

- If you want to create a window on the screen, check the checkbox of the specified window name to create the window quickly.
- If you want to delete a window that already exists on the screen, uncheck the checkbox of the specified window name to delete the window quickly.
- After deleting the window, all the traces in this window will all be deleted. If all traces in this channel are deleted, then this channel will also be deleted.

Trace Information in the Window

One of the important functions of the window is to display trace curves. A window can contain up to 100 traces. The trace information of the window is displayed at the top of the window. For details, refer to User Interface.

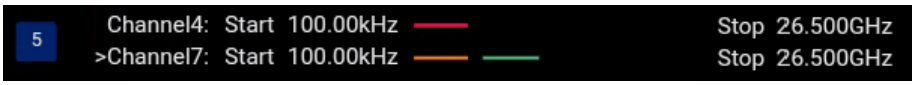


Traces with different data formats can all be displayed in one window. The definitions for the horizontal coordinate (X-axis) and vertical coordinate (Y-axis) in different trace formats are different. The scale type shown in the vertical coordinate indicates the Y-axis type of the currently activated trace.

Channel Information in the Window

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

The channel information of all the traces in the window is displayed at the bottom of the window.



- ">" following a channel No. indicates the currently selected active channel. Only one channel can be activated globally.
- The definitions for the Start and Stop values are determined by the sweep type of the channel. For power sweep, they indicate power values; for frequency sweep, they indicate frequency values.
- When multiple channels are available, they are displayed from top to bottom in sequence.
- The line (s) with different color (s) following the Start value indicates all the traces of this channel. The line color is the same as the color of the trace displayed in the window. If multiple traces are available for the channel, they will be displayed from left to right in sequence.

6.4 Window Setup

Select a Window

Click the drop-down button of **Select**, then all the available windows are displayed to choose. Choose the desired window to be the new active window. The selected window is highlighted in green box.

Window Title

Click the input field of "Title", then the virtual keypad is displayed. Input the title of the current active window with the virtual keypad.

Add a Window

Click the drop-down button of **Add Window** to add the desired window.

- **New Window:** creates a new window.
- **New Trace + Window:** creates a new window and creates a new trace for the currently activated channel in the window.
- **New Trace + Channel + Window:** creates a new window, creates a new channel, and creates a new trace for the new channel in the window.

The newly created window, channel, and trace are automatically activated. New IDs with the minimal number that is not in use will be assigned to the newly created window, channel, and trace.

Delete a Window

Click the drop-down button of **Delete Window**. All the available windows are displayed to choose. Select the desired window to delete it. Note that the window with the green dot located at the left indicates that it is currently selected and activated.

Once the window is deleted, all the traces contained in the window will all be deleted. If all traces in this channel are deleted, then this channel will also be deleted.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

If you delete the currently activated window, then the window with the minimal ID not in use among all of the current windows will automatically become the new active window.

6.5 Sheet Setup

Sheets are used to group multiple windows. Multiple sheet tabs can be displayed on the screen. By default, there is only one sheet when you power on the instrument. When multiple sheets are available, there will be multiple sheet tabs. Click the specified sheet tab to switch to the desired sheet. Each sheet can contain multiple windows. For details, refer to User Interface.

Select a Sheet

To select an active sheet, perform either of the following operations.

- Click the drop-down button of **Select**, then all the available sheets are available to choose. Select the desired sheet to make it active.
- Click on the sheet tab to switch the sheet. The selected sheet is the active sheet.

Sheet Title

Click the input field of "Title", then the virtual keypad is displayed. Input the title of the current active sheet with the virtual keypad. The sheet tab name will also be updated once you have modified the sheet title.

Add a Sheet

Click the drop-down button of **Add Sheet** to add the desired sheet.

- **New Sheet:** creates a new sheet.
- **New Trace + Sheet:** creates a new sheet and creates a new window in the sheet. Create a new trace for the currently activated channel and display it in the window.
- **New Trace + Channel + Sheet:** creates a new sheet and creates a new window in the sheet. Create a new channel. Create a new trace for the new channel and display it in the window.

The newly created sheet, window, channel, and trace are automatically activated. New IDs with the minimal number that is not in use will be assigned to the newly created sheet, window, channel, and trace.

Delete a Sheet

Click the drop-down button of **Delete Sheet**. All the available sheets are displayed to choose. Select the desired sheet to delete it. Note that the sheet with the green dot located at the left indicates that it is currently selected and activated.

Once the sheet is deleted, all the windows in the sheet and all the traces contained in the window will all be deleted. If all traces in this channel are deleted, then this channel will also be deleted.

If you delete the currently activated sheet, then the sheet with the minimal ID not in use will automatically become the active sheet. If there is only one sheet left, this last sheet cannot be deleted.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

7 Frequency

Sets the frequency parameters of the stimulus source.

Frequency Range

Specifies the frequency range for the measurement. Only when you set the sweep type to "Linear Frequency" or "Log Frequency", can you set the frequency range. After configuring the frequency range, the start frequency and stop frequency values are displayed at the bottom of the window.

Click **Frequency** > **Main** to set the start frequency, stop frequency, center frequency, and span. They have a coupling relationship. Once you modify one of them, the rest three parameters will also be modified accordingly. The frequency step (Step) is calculated based on the Span value.

- $f_{Center} = (f_{Start} + f_{Stop}) / 2$
- $f_{Span} = (f_{Stop} - f_{Start})$
- $f_{Step} = f_{Span}$ / (number of sweep points - 1)

CW Frequency

CW (Continuous Wave) refers to an electromagnetic wave characterized by constant frequency and constant amplitude. Only when the sweep type is "Power", can this menu be enabled.

The range of the CW frequency is from 5 kHz to F_{max}. By default, it is 1 GHz. Once the CW value is set, it will be displayed at the bottom of the window.

Tip: F_{max} denotes the instrument's maximum supported frequency (refer to Chapter 2, "DNA Series Product Models").

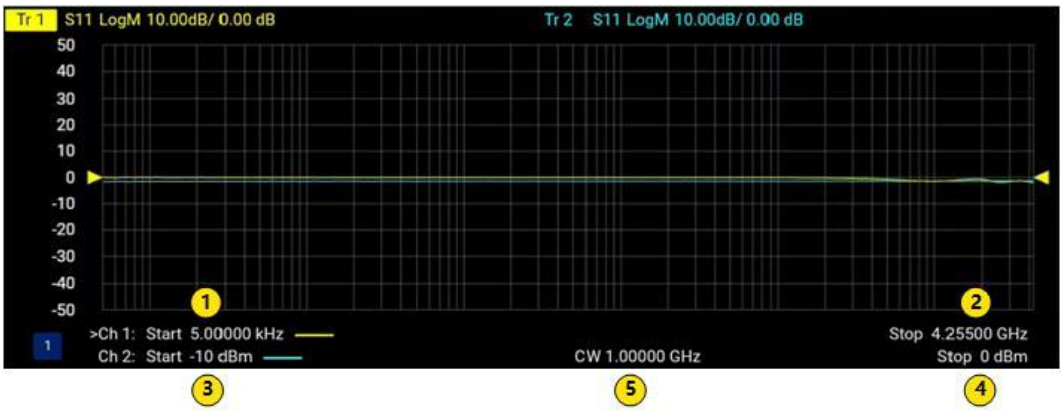
8 Sweep

A sweep is a series of consecutive data point measurements taken over a specified sequence of stimulus values. The sweep settings are valid for the current channel. All the measurement traces for each channel share the same sweep parameters.

8.1 Main

VNA supports multiple types of sweep. The relevant information is displayed below the window, as shown in the figure below.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Linear Frequency

X-axis represents the linear frequency, displayed with equal horizontal divisions in the X-axis. You need to set the start frequency, stop frequency, and sweep points for the linear frequency sweep.

- **Start Frequency:** sets the start value of the frequency sweep or the start point of the X-axis. It is displayed at the bottom of the window, following "Start", as ① indicated in the above figure. This menu functions the same as the start frequency in **Frequency** menu (click **Frequency > Main > Start**). Configuration of this parameter in both **Sweep** and **Frequency** menus has the same effect.
- **Stop Frequency:** sets the stop value of the frequency sweep or the stop point of the X-axis. It is displayed at the bottom of the window, following with the "Stop" icon, as ② indicated in the above figure. This menu functions the same as the stop frequency in Frequency menu (click **Frequency > Main > Stop**). Configuration of this parameter in both **Sweep** and **Frequency** menus has the same effect.

Log Frequency

In this mode, the frequency is stepped in logarithmic increments and the data is displayed on a logarithmic X-axis. This mode allows you to observe wide frequency range. The grid is not equally spaced and varies periodically. This is usually slower than a continuous sweep with the same number of points. It is slower than linear frequency sweep.

The Log frequency sweep also needs to set the start frequency, stop frequency, and sweep points. The setting method is the same as that in "Linear Frequency" sweep.

Power Sweep

Activates the power sweep at a single frequency that you specify. The power sweep either increases or decreases source power in discrete steps. X-axis represents the linear power.

- You need to set the start power, stop power, and sweep points for the power sweep. Here, start power indicates the start value of the power sweep or the start point of the X-axis. It is displayed at the bottom of the window, following "Start", as ③ indicated in the above figure. Stop power indicates the stop value of the power sweep or the end point of the X-axis. It is displayed at the bottom of the window, following "Stop", as ④ indicated in the above figure.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- To set the fixed frequency value of the sweep signal, click **Frequency** > **Main** > **CW**, then input the desired value into the input field of CW. The CW value is displayed at the bottom of the window, as ⑤ indicated in the above figure.

- Power sweep is optimized for speed. For highest measurement accuracy during a power sweep, it may be necessary to increase the Dwell Time to allow the source more time to settle. To set the dwell time, click **Sweep** > **Sweep Timing** > **Dwell Time**.

- The **Start** and **Stop** sub-menus in the **Sweep** menu have the same function as that in **Power** menu (click **Power** > **Main** > **Start** and **Stop**). Configuration of the two parameters in both **Sweep** and **Power** menus has the same effect.

Segment Sweep

Segment sweep refers to user-defined one or multiple frequency ranges of sweep (called segment). For each segment, you can define independent power level, IF bandwidth, IF bandwidth per port, sweep time, delay time, and dwell time. All the segments are swept in order, just as other sweep operation.

X-axis represents the linear incremented frequency value, and its display format is the same as described in the "linear frequency sweep". For detailed setting methods for the segment sweep, refer to the descriptions in Segment Control.

Sweep Points

Sweep points refer to the number of data points acquired in single sweep. The range of the sweep points is from 1 to 100,001. By default, it is 201.

- When the sweep type is "Segment", sweep points cannot be set.

- When you modify the sweep points, the instrument starts to re-sweep all the channels according to the channel No. sequence.

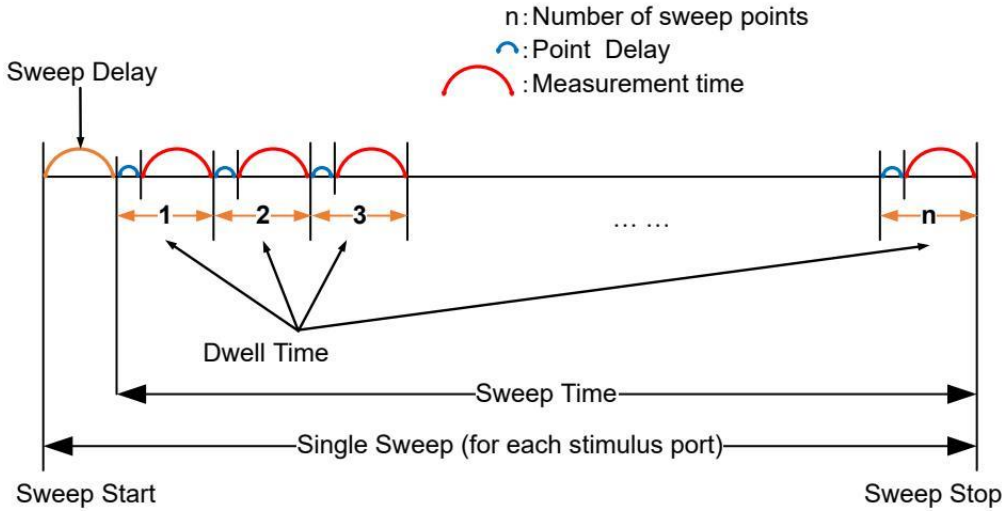
- To achieve the greatest trace resolution, use the maximum number of data points. However, the sweep time changes proportionally with the number of points. To obtain higher throughput, keep the number of points to a smaller value within an allowable trace resolution.

- To ensure an accurate measurement calibration, ensure that the number of sweep points used in calibration is consistent with that used in measurement.

8.2 Sweep Timing

The following figure lists the relationship between the sweep time, delay time, and dwell time. Point Delay in the figure refers to the dwell time.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Sweep Time

Sweep time specifies the time the VNA instrument takes to acquire RF data for a sweep.

The sweep time includes the dwell time but does not include the sweep delay time.

Auto Sweep Time

- **Auto (Default):** auto adjusts the sweep time, and the dwell time is fixed to be 0 s. If you set the dwell time to a value other than 0, then manual sweep time configuration is automatically enabled.
- **Manual:** sets the sweep time and dwell time manually. When you configure the dwell time to 0, then auto sweep time configuration is automatically enabled.

Dwell Time

Specifies the time the VNA instrument stays at each measurement point to complete the signal stimulus, response acquisition, and data processing. The max. dwell time is 20 s.

Increasing the dwell time will slow the sweep speed, but it increases the measurement accuracy for the electrically long device.

Delay Time

Specifies the time to wait just before acquisition begins for each sweep. By default, it is 0 s. Its range is from 0 s to 20 s. This delay is in addition to Dwell Time (per point) and External Trigger delay if enabled.

Sweep Sequence

- **STD:** by default, it is standard sweep. The analyzer sweeps all data points for each source port in sequence. For example, first sweep the S11/S21 parameters of all the data points, then sweep the S12/S22 parameters of all the data points.
- **Point:** the analyzer measures all S-parameters at each frequency point before stepping to the next frequency. The display trace is updated as each data point is measured. Point sweep usually slows the sweep and is useful only in rare circumstances.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

8.3 Segment Control

If you set the sweep type to Segment, you need to configure settings for each segment. Each channel supports up to 32 segments. This instrument supports up to 4,096 segments.

Add a Segment

Adds a sweep segment at last segment. A new line of segment parameters will be added in the segment table.

- The start frequency of the newly added segment is the stop frequency value of the last segment.
- The added segment will not change the current active segment.

Insert a Segment

Inserts a sweep segment before the selected segment. A new line of segment parameters will be inserted before the selected segment in the segment table.

- For the inserted segment, by default, its start frequency is the stop frequency of its previous segment; its stop frequency is the start frequency of its next segment.
- After the new segment is inserted, the new inserted segment is, by default, selected as the current active segment.

Delete a Segment

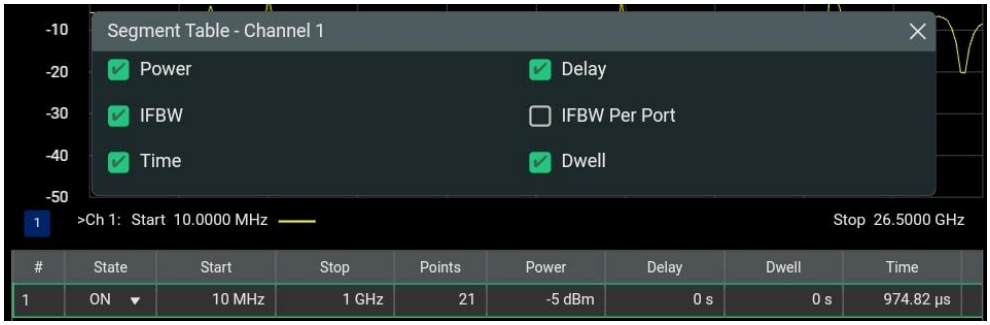
Deletes a specified segment from the segment table. If there is only one segment, after the segment is deleted, the sweep type automatically changes to linear frequency sweep.

Delete All Segments

Deletes all of the segments from the segment table. After all segments are deleted from the segment table, the sweep type automatically changes to linear frequency sweep.

Configure the Segment Table

The segment table and the segment table configuration information are displayed in the following figure.



By default, the segment table includes the following parameters: segment No., on/off status of the segment, start frequency, stop frequency, and sweep points. Check the checkbox of the specified parameters that you want to display in the segment table, and then the relevant parameters will be displayed in the segment table. Note that "IFBW" and "IFBW Per Port" are mutually exclusive, and they

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

cannot be selected at the same time.

Click the specified parameter value in the segment, then the numeric keypad is displayed, input the desire value to complete parameter configuration. For the state setting, click the drop-down button of **State** to enable or disable the state of the specified segment. Restrictions for segment sweep configuration:

- The frequency range of a segment is not allowed to overlap the frequency range of any other segment. The start frequency of each segment must be greater than the stop frequency of its previous segment.
- When the start frequency of a modified segment is less than the stop frequency of the previous segment, a prompt message will appear, and the start and stop frequencies of the previous segment will be changed to the start frequency of the current segment.
- At least one segment must be enabled, otherwise, the sweep type automatically switches to the linear frequency sweep.
- Up to 32 segments are supported.
- The combined number of data points for all segments in a sweep cannot exceed the max number of data points per trace.
- Each segment is ordered based on the frequency value from high to low.

Display the Segment

Controls whether to display the segment table below the window.

- **Auto:** (Default)
  - Only when you click **Sweep** > **Segment Table** to enter the sub-menu of **Segment Table**, can the segment table be displayed below the current active window automatically.
  - When you select other sub-menus other than **Segment Table** under **Sweep**, the segment table will not display below the current active window.
- **On:** When the current active channel is segment sweep type, the segment table of the current active channel is displayed below the current active window.
- **Off:** segment table is not displayed below the current active window.

9 Measure

The **Measure** menu enables you to set the measurement parameters for the trace.

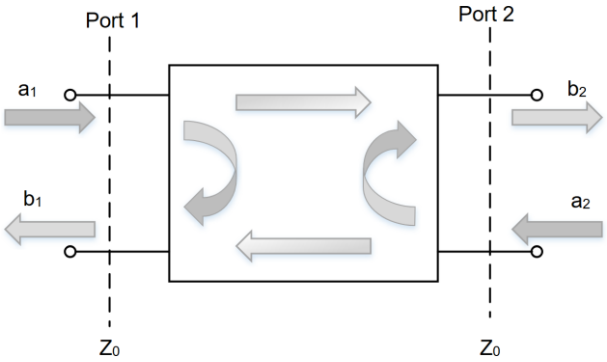
9.1 S-Parameter

S-parameters are used to describe the situation of transmission and reflection signals when a signal meets a discontinuity in impedance. S-parameters are relative parameters, defined as the ratio of two complex voltages, containing amplitude and phase information of the related signals.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- Response port, also known as the receiver port of the vector network analyzer. The transmitted signal goes to this port via the DUT.
- Stimulus port, also known as the transmit port of the vector network analyzer. The output signal on this port goes to the DUT.

For the 2-port vector network analyzer, there are 4 S-parameters: S11, S21, S12, and S22. The definition for each S-parameter is shown in the following figure.



When the source goes to Port 1, the measurement is said to be in the forward direction. When the source goes to Port 2, the measurement is said to be in the reverse direction. The analyzer switches the source and receiver automatically to make a forward or reverse measurement. Therefore, the analyzer can measure all of the four S-parameters for a 2-port device with a single connection.

- S11 - forward reflection
- S21 - forward transmission
- S12 - reverse transmission
- S22 - reverse reflection

The formula is shown as follows:

$S11 = \frac{b_1}{a_1}\Big\vert_{a_2=0}$ $S21 = \frac{b_2}{a_1}\Big\vert_{a_2=0}$	$S12 = \frac{b_1}{a_2}\Big\vert_{a_1=0}$ $S22 = \frac{b_2}{a_2}\Big\vert_{a_1=0}$

Click the specified S-parameter to set the specified measurement parameters for the currently activated trace.

Table 9.1 Common Measurement with S-Parameters

Measurement Type	Measurement Parameter
Reflection Measurement	Return Loss

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Measurement Type	Measurement Parameter
	SWR Reflection Coefficient Input Impedance S11, S22
Transmission Measurement	Insertion Loss Transmission Coefficient Gain/Loss Group Delay Deviation from Linear Phase Electrical Delay S21, S12

9.2 Balanced S-Parameter

Balanced S-parameters are designed to precisely characterize the signal behavior between the differential (balanced) system and the single-ended system. They can not only describe the transmission and reflection characteristics of signals between different ports, but also reveal the conversion between differential mode and common mode.

The instrument supports a combination of three basic signal modes: Single-ended (s), differential (d), and common mode (c). By properly configuring the port topology of the DUT and its mapping to the physical ports of the instrument, you can obtain S-Parameters, the imbalance of the balanced ports, and key metrics such as the common mode rejection ratio (CMRR) in all modes directly.

Tip:

This feature is only supported by the 4-port model of the Vector Network Analyzer (VNA).

Port Type

- **Single-ended Port (SE)**:

  Consists of one signal wire and one ground wire, and transmits signals with ground wire as the reference basis. It is the most basic type of topology, with the reference impedance of 50 Ω. Its advantage lies in its simple structure, and its disadvantage is poor interference immunity, suitable for low-speed, short-range signal transmission.

- **Balanced Port (BAL)**:

  Consists of 2 signal lines (positive/negative) with equal amplitude and reversed phase, without relying on ground wire, to offset external interference through differential signal transmission. Its reference impedance is typically 100 Ω (differential mode) or 25 Ω (common mode), applicable to high-speed, long-range transmission.

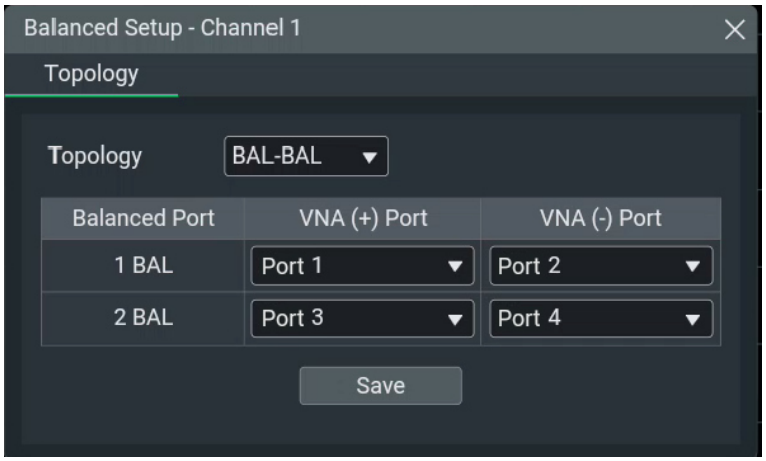
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Signal Mode

- **Differential Mode**: An effective signal mode, defined as the signal difference (A-B) of two signal lines. It is a core useful signal of balanced transmission with inherent interference immunity.
- **Common Mode**: An interference signal mode, defined as the average value ((A+B)/2) of the signal of the two signal lines. It is mainly generated by external electromagnetic interference, asymmetric device structure and other factors. Ideally, it should be strongly suppressed by the device.
- **Mixed Mode**: In practical applications, the signal of a balanced port is a mixture of differential mode and common mode. The Balanced S-parameters can quantify the ratio of the two modes and their mutual conversion.

Configuration

Click **Measure** > **Balanced** > **Topology** to enter the Balanced Setup interface.



1. Select the topology type.
2. Configure the ports based on the actual DUT connection.
3. Click **Save** to save the configuration. In the Balanced sub-menu, you can see the commonly used balanced S-parameters. To view more balanced S-parameters, refer to the items under the **Other** menu.
4. After the current trace is selected, you can select the desired balanced S-parameter for the trace.

9.2.1 Balanced S-Parameters[4]

The balanced S-parameters are expressed in Sabxy. Here,

- a: Device output mode (d = differential, c = common mode, s = single-ended)
- b: Device input mode (d = differential, c = common mode, s = single-ended)

4 For two-port models, only Sdd11, Scc11, Sdc11, and Scd11 are supported.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- x: Device output logical port number
- y: Device input logical port number

For example,

Ssc21 indicates the transmission from "Input Logical Port 1 (Common Mode Balanced Port) to Output Logical Port 2 (Single-Ended Port)".

Sdd21 indicates the differential transmission from "Input Logical Port 1 (Differential Balanced Port) to "Output Logical Port 2 (Differential Balanced Port)".

Calculation Formula for Balanced S-Parameters

The Balanced S-parameters are calculated based on a linear combination of the single-ended S-parameters. The formula is as follows:

Table 9.2 Reflection Parameter (x = y)

Parameter	Meaning	Formula
Sdd11	Port 1 Differential-Mode Reflection	$0.5\times (S_{11} - S_{12} - S_{21} + S_{22})$
Scc11	Port 1 Common-Mode Reflection	$0.5\times (S_{11} + S_{12} + S_{21} + S_{22})$
Sdc11	Port 1 Common-to-Differential Mode-Conversion Reflection	$0.5\times (S_{11} + S_{12} - S_{21} - S_{22})$
Scd11	Port 1 Differential-to-Common Mode-Conversion Reflection	$0.5\times (S_{11} - S_{12} + S_{21} - S_{22})$
Sdd22	Port 2 Differential-Mode Reflection	$0.5\times (S_{33} - S_{34} - S_{43} + S_{44})$
Scc22	Port 2 Common-Mode Reflection	$0.5\times (S_{33} + S_{34} + S_{43} + S_{44})$
Sdc22	Port 2 Common-to-Differential Mode-Conversion Reflection	$0.5\times (S_{33} + S_{34} - S_{43} - S_{44})$
Scd22	Port 2 Differential-to-Common Mode-Conversion Reflection	$0.5\times (S_{33} - S_{34} + S_{43} - S_{44})$
Sss11	Port 1 (Single-Ended to Single-Ended) Reflection	S_{11}
Sss22	Port 2 (Single-Ended to Single-Ended) Reflection	S_{22}
Sss33	Port 3 (Single-Ended to Single-Ended) Reflection	S_{33}
Sss44	Port 4 (Single-Ended to Single-Ended) Reflection	S_{44}

Table 9.3 Balanced-Port Transmission (BAL1 ⇄ BAL2)

Parameter	Meaning	Formula
Sdd12	Port 1 Differential → Port 2 Differential Transmission	$0.5\times (S_{13} - S_{14} - S_{23} + S_{24})$
Sdd21	Port 2 Differential → Port 1 Differential Transmission	$0.5\times (S_{31} - S_{32} - S_{41} + S_{42})$
Scc12	Port 1 Common-Mode → Port 2 Common-Mode Transmission	$0.5\times (S_{13} + S_{14} + S_{23} + S_{24})$

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Scc21	Port 2 Common-Mode → Port 1 Common-Mode Transmission	0.5× (S_{31} + S_{32} + S_{41} + S_{42})
Sdc12	Port 2 Common-Mode → Port 1 Differential Transmission	0.5× (S_{13} + S_{14} − S_{23} − S_{24})
Sdc21	Port 1 Common-Mode → Port 2 Differential Transmission	0.5× (S_{31} + S_{32} − S_{41} − S_{42})
Scd12	Port 2 Differential → Port 1 Common-Mode Transmission	0.5× (S_{13} − S_{14} + S_{23} − S_{24})
Scd21	Port 1 Differential → Port 2 Common-Mode Transmission	0.5× (S_{31} − S_{32} + S_{41} − S_{42})

Table 9.4 Differential ⇄ Single-Ended Cross Terms (ds: Differential Output / sd: Single-Ended Output)

Parameter	Formula	Parameter	Formula
Sds11	(S_{11} − S_{21})/√2	Ssd11	(S_{11} − S_{12})/√2
Sds21	(S_{31} − S_{41})/√2	Ssd21	(S_{31} − S_{32})/√2
Sds12	(S_{12} − S_{22})/√2	Ssd12	(S_{13} − S_{14})/√2
Sds22	(S_{32} − S_{42})/√2	Ssd22	(S_{33} − S_{34})/√2
Sds13	(S_{13} − S_{23})/√2	Ssd13	(S_{21} − S_{22})/√2
Sds23	(S_{33} − S_{43})/√2	Ssd23	(S_{23} − S_{24})/√2
Sds14	(S_{14} − S_{24})/√2	Ssd14	(S_{41} − S_{42})/√2
Sds24	(S_{34} − S_{44})/√2	Ssd24	(S_{43} − S_{44})/√2

Table 9.5 Common-Mode ⇄ Single-Ended Cross Terms (cs: Common-Mode Output / sc: Single-Ended Output)

Parameter	Formula	Parameter	Formula
Scs11	(S_{11} + S_{21})/√2	Ssc11	(S_{11} + S_{12})/√2
Scs21	(S_{31} + S_{41})/√2	Ssc21	(S_{31} + S_{32})/√2
Scs12	(S_{12} + S_{22})/√2	Ssc12	(S_{13} + S_{14})/√2
Scs22	(S_{32} + S_{42})/√2	Ssc22	(S_{33} + S_{34})/√2
Scs13	(S_{13} + S_{23})/√2	Ssc13	(S_{21} + S_{22})/√2
Scs23	(S_{33} + S_{43})/√2	Ssc23	(S_{23} + S_{24})/√2
Scs14	(S_{14} + S_{24})/√2	Ssc14	(S_{41} + S_{42})/√2
Scs24	(S_{34} + S_{44})/√2	Ssc24	(S_{43} + S_{44})/√2

9.2.2 Topology

The 2-port instrument supports only the BAL topology type. Other types of topologies are supported by the instrument model with 4 ports and above. The physical ports cannot be mapped repeatedly, otherwise the configuration is invalid. The topology configuration is valid for the channel level.

The relationship between the logical ports and the physical ports is based on the selected topology

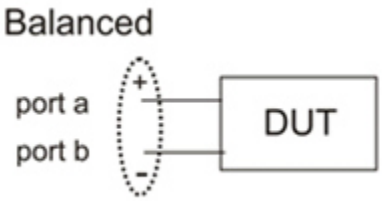
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

type. The mapping rules for the physical ports are as follows:

- Single-ended logical ports (SE, Single-ended): maps to 1 physical port.
- Balanced logical ports (BAL, Balance): maps to 2 non-overlapping physical ports (distinguishing positive line/negative line).
- Duplicate mapping of physical ports is not allowed; otherwise, the configuration will be invalid.

BAL (Single-ended Balanced Port Component)

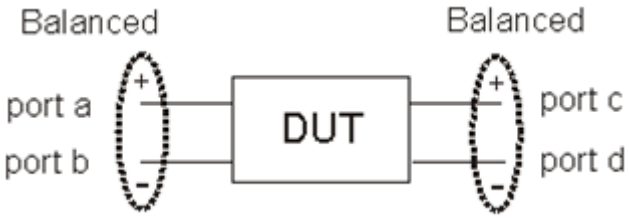
Only 1 balanced port is included. It supports the reflection characteristic measurement, without transmission-type parameters.



- **Port Mapping:**
  Physical ports (Port a (+) and Port b (-)) are mapped to one balanced logical port BAL1.
- **Measurement Parameters:** Sdd11, Scc11, Sdc11, Scd11 (Reflection class)

BAL-BAL (Balanced-Balanced Port Component)

Both inputs and outputs are balanced ports, supporting full reflection, transmission, and mode conversion measurements.

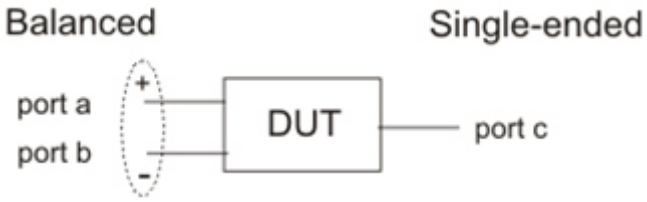


- **Port Mapping:**
  Physical ports (Port a (+) and Port b (-)) are mapped to one balanced logical port BAL1 (input).
  Port c (+) and Port d (-) are mapped to one balanced logical port BAL2 (output).
- Measurement Parameters:
  - Reflection parameters: Sdd11, Sdd22, Scc11, Scc22, Sdc11, Sdc22, Scd11, and Scd22
  - Transmission parameters: Sdd21, Sdd12, Scc21, Scc12, Sdc21, Sdc12, Scd21, and Scd12
  - Imbalance/CMRR: Imbal1, Imbal2, and CMRR

BAL-SE (Balanced-Single-ended Port Component)

Balanced port as input and single-ended port as output, supporting balance → single-ended conversion-related measurements.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



- **Port Mapping:**

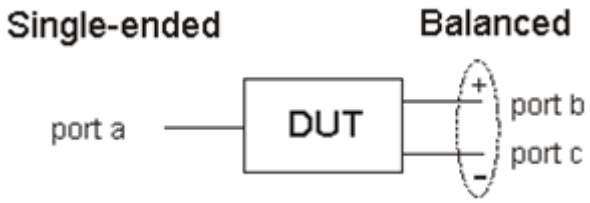
  Physical ports (Port a (+) and Port b (-)) are mapped to one balanced logical port BAL1 (input).

  Port c is mapped to 1 single-ended logical port SE2 (output).

- Measurement Parameters:
  - Reflection parameters: Sdd11, Scc11, and Sss22
  - Transmission parameters: Sds12, Scs12, Ssd21, and Ssc21
  - Imbalance/CMRR: Imbal, CMRR1, and CMRR2

SE-BAL (Single-ended-Balanced Port Component)

Single-ended port as input and balanced port (default topology type) as output, supporting single-ended-balanced conversion-related measurements.



- **Port Mapping:**

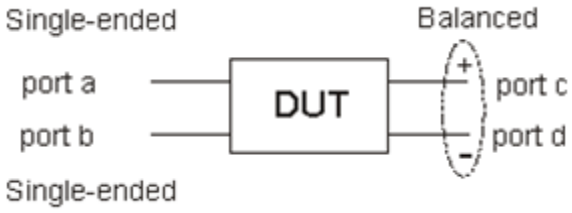
  Physical port (Port a) is mapped to 1 single-ended logical port SE1 (input).

  Port b (+) and Port c (-) are mapped to 1 balanced logical port BAL2 (input).

- Measurement Parameters:
  - Reflection parameters: Sss11, Sdd22, Scc22, Sdc22, and Scd22
  - Transmission parameters: Ssd12, Ssc12, Sds21, and Scs21
  - Imbalance/CMRR: Imbal, CMRR1, and CMRR2

SE-SE-BAL (Single-ended-Single-ended-Balanced Port Component)

2 independent single-ended ports as inputs and 1 balanced port as output, supporting single-ended-single-ended-balanced conversion-related measurements.



Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Port Mapping:**

  Physical port Port a is mapped to 1 single-ended logical port SE1 (input).

  Port b is mapped to 1 single-ended logical port SE2 (input).

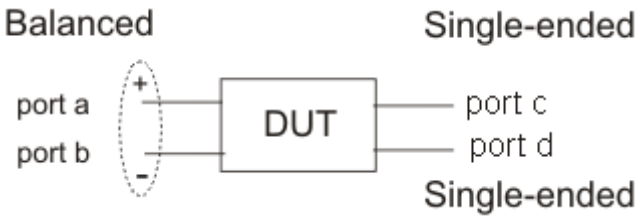
  Port c (+) and Port d (-) are mapped to 1 balanced logical port BAL1 (output).

- Measurement Parameters:

  - Reflection parameters: Sss11, Sss22, Sdd33, Scc33, Sdc33, and Scd33
  - Transmission parameters: Ssd13, Ssd23, Ssc13, Ssc23, Sds31, Sds32, Scs31, and Scs32
  - Imbalance/CMRR: Imbal1, Imbal2, CMRR1, and CMRR2

BAL-SE-SE (Balanced-Single-ended-Single-ended Port Component)

1 balanced port as input and 2 independent single-ended ports as outputs, supporting balanced-single-ended-single-ended conversion-related measurements.



- **Port Mapping:**

  Physical ports Port a (+) and Port b (-) are mapped to 1 balanced logical port BAL1 (input).

  Port c is mapped to 1 single-ended logical port SE2 (output).

  Port d is mapped to 1 single-ended logical port SE3 (output).

- Measurement Parameters:

  - Reflection parameters: Sdd11, Scc11, Sdc11, Scd11, Sss22, and Sss33
  - Transmission parameters: Sds12, Sds13, Scs12, Scs13, Ssd21, Ssd31, Ssc21, and Ssc31
  - Imbalance/CMRR: Imbal1, Imbal2, CMRR1, and CMRR2

9.2.3 Imbalance/CMRR[5]

Imbalance of the Balanced Ports

Imbalance (Imba) of the balanced port is an indicator to measure the degree of matching between the two physical ports that constitute a balanced port.

Imba can be analyzed from three parameters: Gain, phase, and system performance.

- **Differential and common-mode gain**

  The differential-mode gain describes the signal response of the balanced port for the differential

5 Supported only on four-port instruments.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

mode (A-B) stimulus. The common-mode gain describes the signal response of the balanced port for the common-mode ((A+B)/2) stimulus. The ideal balanced port should have a high differential mode gain and a low common mode gain.

- **Phase imbalance**

  measures the phase difference between two physical ports. Ideally, the phases of the two ports should be completely opposite, with a difference of 180 degrees.

- **Common mode rejection ratio (CMRR)**

  CMRR is the ratio of transmission characteristics of balanced port differential mode to that of common mode. A higher CMRR value indicates lower imbalance of the balanced port (good port match); otherwise, a lower CMRR value indicates higher imbalance of the balanced port (port mismatch).

The calculation of Imba is strongly related to the balanced topology of the DUT, and is only applicable to vector network analyzers with 4 ports and above.

Table 9.6 Imbalance

Topology	Formula	Remarks
BAL	Imbal = Sba/Sbb	• Here, a and b are two physical ports of the same balanced port • Sba = single-ended transmission from Port b to Port a. • Sbb = single-ended reflection from Port b to Port b. • Transmission/Reflection ratio of two physical channels on the same balanced port
BAL-SE	Imbal = Sca/Scb	• c indicates the single-ended logical port. • a and b are two physical ports of the balanced port. • Sca = single-ended transmission from Port c to Port a. • Scb = single-ended transmission from Port c to Port b.
SE-BAL	Imbal = Sba/Sca	• a indicates the single-ended logical port. • b and c are two physical ports of the balanced port. • Sba = single-ended transmission from Port b to Port a. • Sca = single-ended transmission from Port c to Port a.
BAL-BAL	Imbal1 = (Sac − Sad)/ (Sbc − Sbd) Imbal2 = (Sca − Scb)/ (Sda − Sdb)	• a and b are the physical ports of Balanced Port 1. • c and d are the physical ports of Balanced Port 2. • Imbal1 is the transmission path from Balanced Port 1 to Balanced Port 2. • Imbal2 is the transmission path from Balanced Port 2 to Balanced Port 1.
SE-SE-BAL	Same as BAL-BAL	• a and b are single-ended logical ports. • c and d are physical ports of the balanced port.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Topology	Formula	Remarks
		• Corresponds to the transmission path from single-ended port to balanced port.
BAL-SE-SE	Same as BAL-BAL	• a and b are two physical ports of the balanced port. • c and d are two single-ended logical ports. • Corresponds to the transmission path from balanced port to single-ended port.

CMRR

CMRR is the ratio of transmission characteristics of balanced port differential mode (d) to that of the common mode (c). CMRR reflects the device's rejection of the common-mode signal. The higher the ratio, the greater the differential device's rejection of the common-mode, capable of transmitting the differential signals efficiently. On the contrary, the lower the CMRR value, the more vulnerable to common mode noise interference.

CMRR is strongly related to the balanced S-parameters and the balanced topology of the DUT.

Topology	CMRR Formula	Description
BAL	N/A	Port reflection characteristics only, without differential/common mode signal transmission, unable to calculate CMRR
BAL-BAL	CMRR = Scc21/Sdd21	Sdd21 indicates the transmission-type balanced S-parameter in differential mode; Scc21 indicates the same path transmission-type balanced S-parameter in common mode, and is the ratio of the transmission of core differential mode to common mode.
BAL-SE	CMRR1 = Ssc21/Ssd21 CMRR2 = Scs12/Sds12	Each corresponds to the bidirectional transmission path between the balanced port and the single-ended port. They are ratios of transmission-type balanced S-parameter of the differential mode to that of the common mode for the same path.
SE-BAL	CMRR1 = Scs21/Sds21 CMRR2 = Ssc12/Ssd12	Corresponds to the bidirectional transmission path between the single-ended port and the balanced port.
BAL-SE-SE	CMRR1 = Scs12/Sds12 CMRR2 = Scs13/Sds13	Corresponds to the independent transmission path between the balanced port and two single-ended ports.
SE-SE-BAL	CMRR1 = Scs31/Sds31 CMRR2 = Scs32/Sds32	Corresponds to the independent transmission path between two single-ended ports and the balanced port.

9.3 Meas Setup

Parameter Conversion

Refer to descriptions in Parameter Conversion in **Math** > **Analysis** menu. When you modify the settings in parameter conversion in either **Meas Setup** or **Analysis** sub-menu of **Math**, the modification will be global to parameter conversion in both of the menus.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Correction

For the correction function, refer to descriptions in Correction/Correction Properties (click **Calibration** > **Main** to enter the calibration menu). When you modify the correction state in any function menu, the modification will be global to correction settings in other menus.

Trace Hold

For the trace hold function, refer to descriptions in Trace Hold in Trace section (click **Trace** > **Trace Setup** > **Trace Hold**). When you modify the Trace hold function in any function menu, the modification will be global to trace hold settings in other menus.

9.4 TDR Measurement (Option)

This option is supported only on the DNA6000, DNA6000-R, and DNA5000 series products.

ETDR (Enhanced Time Domain Reflectometry) is a technique that analyzes the electrical characteristics of a DUT by simulating a step or pulse stimulus signal sent to the DUT and observing the reflected/transmitted waveforms.
In the field of high-speed digital signal integrity, ETDR can intuitively reveal impedance discontinuities, time delays, losses, and parasitic effects in PCB traces, connectors, packages, and interconnect networks, and is one of the core tools for signal integrity analysis.
The ETDR function supports single-ended/differential multi-port configurations and can simultaneously output time-domain reflection/transmission waveforms, S-Parameter frequency-domain responses, and eye diagram analysis results, enabling a single measurement with multi-dimensional characterization.

ETDR is mainly used for signal integrity analysis of high-speed digital systems. Typical application scenarios and key test points are as follows:

Application Scenarios	Main Measurement Objectives	Typical DUT
PCB Trace Integrity	Characteristic Impedance, Time Delay, Overshoot	High-Speed PCB Differential Pairs, Power Planes
Connector/Package Evaluation	Insertion Loss, Return Loss, Impedance Discontinuity	SFP/QSFP, BGA Packages, RF Connectors
High-Speed Link Simulation	Eye Diagram Margin, Jitter, Rise Time, etc.	SerDes Channels, Backplane Traces
Interconnect Network Diagnosis	Locate Open/Short/Impedance-Discontinuity Positions	Cables, Solder Joints, Vias
Differential Signal Analysis	Differential Impedance, Mode Conversion (Mixed-Mode S-Parameter)	Differential Pairs, Balanced Connectors

Select Measurement > TDR to enter TDR mode.

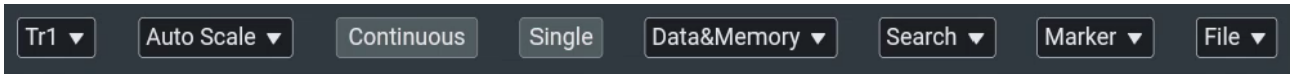
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

9.4.1 Functional Interface Introduction



1. By default, the time-domain analysis view is displayed. The horizontal axis of the window is time and the vertical axis amplitude; the range can be set, and the specific display content depends on the selected topology.
2. By default, the frequency-domain S-Parameter analysis is displayed. The horizontal axis of the window is frequency and the vertical axis amplitude; the range can be set, and the specific display content depends on the selected topology.
3. The shortcut settings menu of the TDR function.

9.4.1.1 Settings Menu Toolbar



- Trace drop-down list, for quickly selecting the current trace
- Auto Zoom supports automatically adjusting the X/Y-axis range for the currently selected trace so that the waveform is displayed at a reasonable scale within the current plot area.

Type	Description
X	On the active trace only, automatically adjust the X-axis range
Y	On the active trace only, automatically adjust the Y-axis range,
X&Y	On the active trace only, automatically adjust the X&Y-axis range,
All Trace	On all traces, automatically adjust the X-axis range

- Continuous: Sets the trigger mode of the current channel to continuous measurement.
- Single: Sets the trigger mode of the current channel to single measurement.
- Data & Memory: Sets the display content of the selected trace, including real-time data and memory data
- Search: Performs peak-value search or time search.
- Marker: Adds, deletes, or selects a cursor.
- File: Selects save state, load state, save trace data, save all data, save SNP data, or image

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

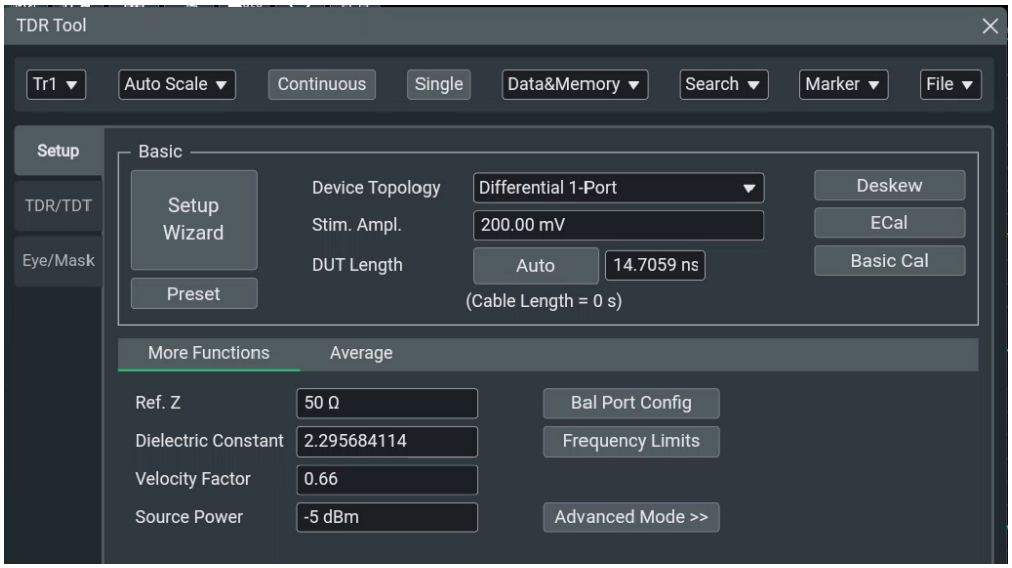
storage

9.4.1.2 Settings Menu Function Keys

	Maximize TDR Menu
	Minimize TDR Menu
	Click the X button to exit TDR mode and return to Standard mode; Click the Preset shortcut in the upper-right corner of the screen to also return to Standard mode.

9.4.2 Parameter Setup

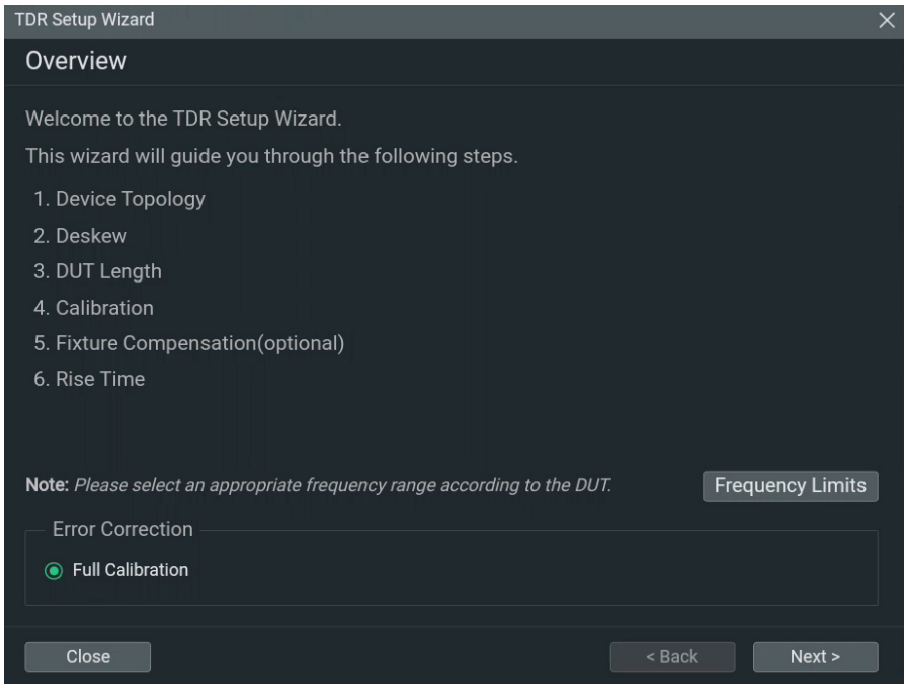
The Settings menu is used to configure the DUT topology type, stimulus amplitude, DUT length, and other basic measurement Parameter; perform Deskew and ECal calibration; and set the reference impedance, dielectric constant, velocity factor, and scan frequency limits as the basic configuration for measurements.
Users can configure via navigation settings or fully manual settings.
ETDR measurement is highly sensitive to calibration quality. It is recommended to complete Deskew (time-delay alignment) and Calibration (error calibration) before each measurement to eliminate the time-delay and loss errors introduced by cables, connectors, and ports.
It is recommended to use the wizard and follow its instructions to perform calibration step by step.



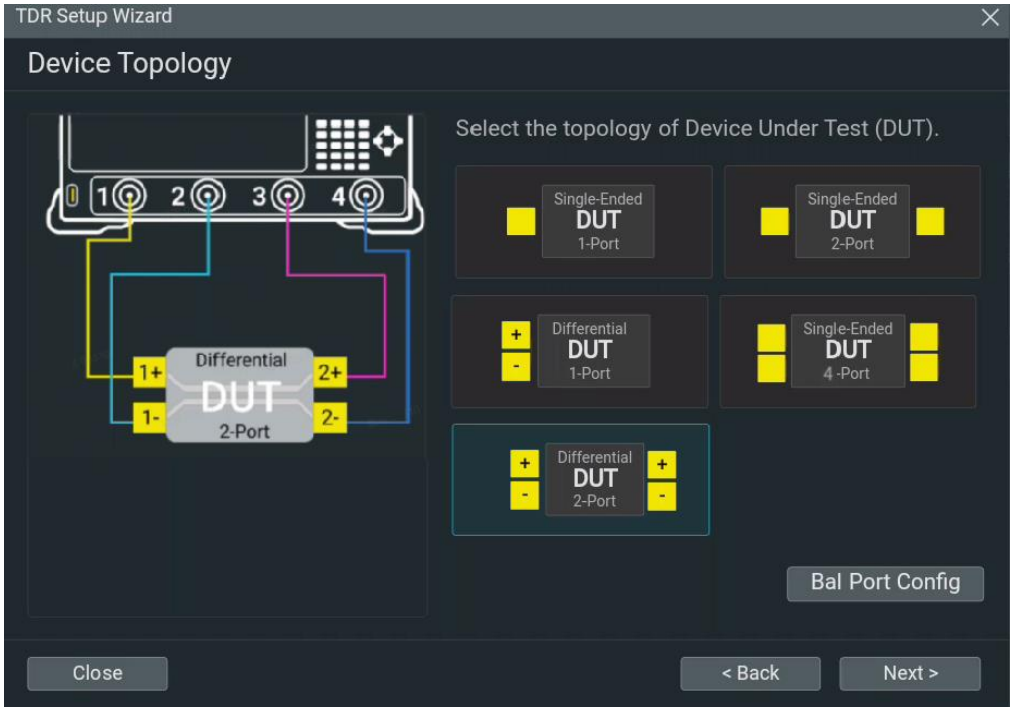
9.4.2.1 Setup Wizard

You can perform calibration according to the wizard; click to enter the navigation interface.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



1. Sets the frequency limits, including the start and stop frequencies.
2. Select the topology type and configure the balanced ports according to the physical connection of the actual test scenario.

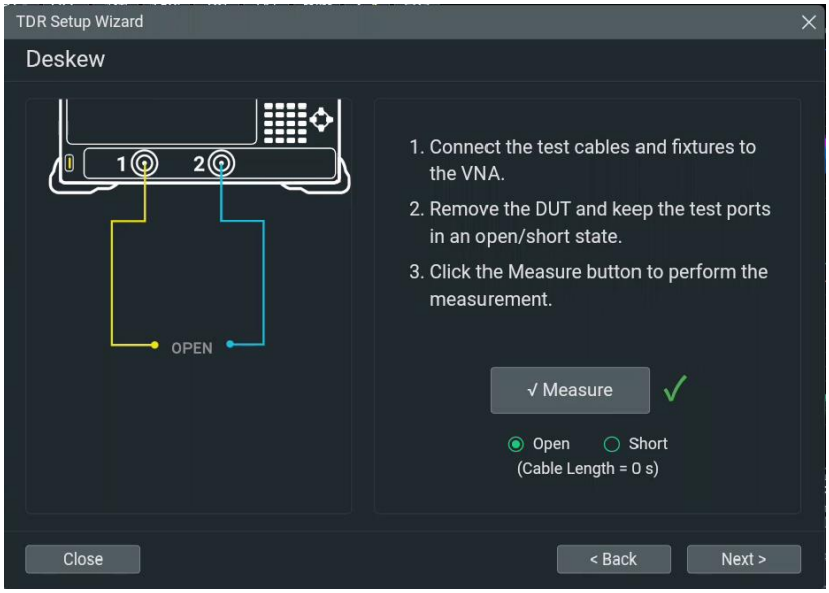




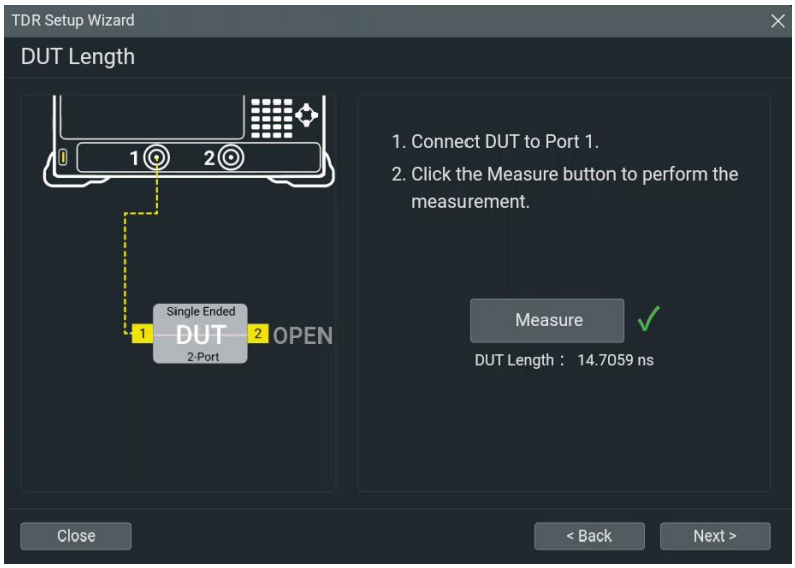
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

3. Select the calibration type and perform Deskew.

After Deskew measurement is complete, a green check mark is displayed, along with the corresponding cable length.

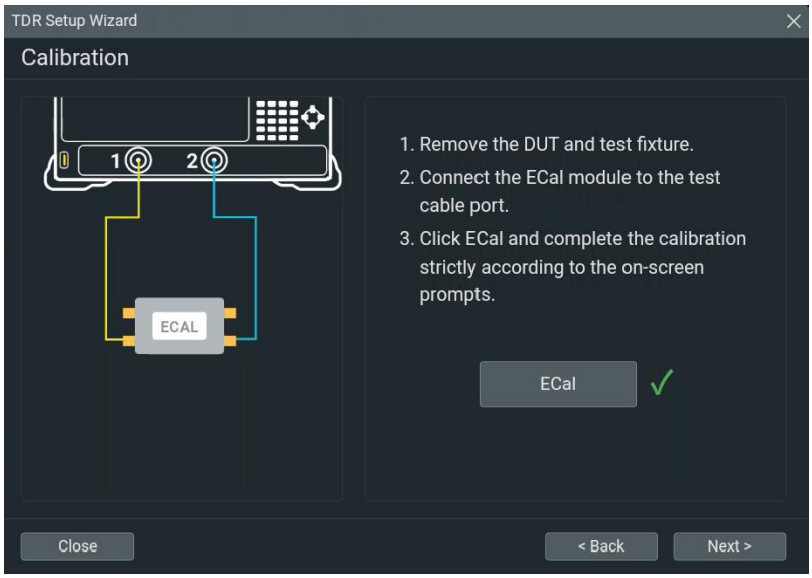


4. Connect the DUT single-ended to Port 1 to automatically measure the DUT length.
   After the DUT length measurement is complete, a green check mark is displayed, along with the corresponding DUT length.



5. Calibration.

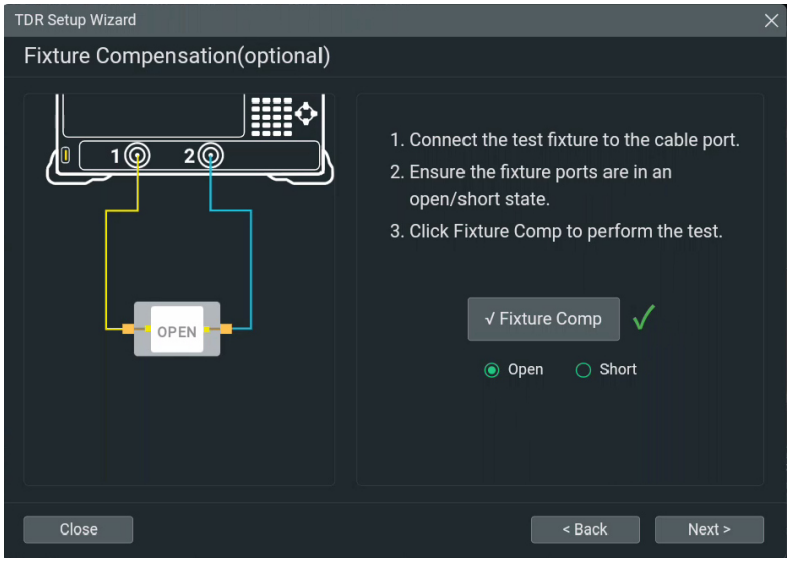
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Perform electronic calibration (ECal) according to the actual test scenario. For detailed operating steps, please refer to the Electronic Calibration chapter.

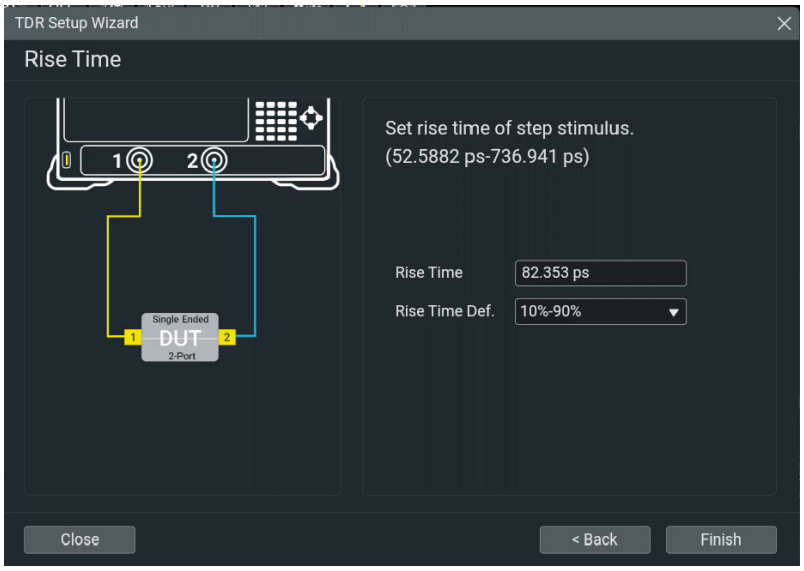
6. Fixture Compensation (Optional Step)

If the DUT needs to be connected via a fixture, after calibration you can connect only the fixture and then perform fixture compensation. For detailed operating steps, please refer to the Fixture chapter.



7. Finally, set the rise time range and value used for simulation to complete the entire wizard operation.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



9.4.2.2 Basic Settings

9.4.2.2.1 DUT Topology Type

Select the DUT topology/port type. The following options are supported:

- Single-Ended 1-Port: Single-ended single port, measures reflection only (e.g., measuring cable impedance
- Single-Ended 2-Port: Single-ended dual port, measures both reflection and transmission
- Single-Ended 4-Port: Single-ended four-port, suitable for multi-channel parallel signals (available on 4-port models only)
- Differential 1-Port: Differential single port, measures differential reflection
- Differential 2-Port: Differential dual port, measures both differential reflection and transmission (available on 4-port models only)

Tip:
After changing the topology type, the calibration data will be lost.

9.4.2.2.2 DUT Length

Two setting methods are supported:

- **Auto**: The instrument automatically estimates the DUT length based on the reflection waveform, suitable for DUTs of unknown length;
  After selecting Auto mode, a setup navigation menu pops up; upon completion of the measurement, a green check mark and the measurement result are displayed.
- **Manual**: Set according to the actual physical length of the DUT, user-defined.
  Default value: 125/ F_{max} (F_{max} in GHz, result in ns);
  Range: 125/F_{max} to 416 ns. (F_{max} is the maximum frequency supported by the instrument.)

Tip: The instrument automatically adjusts the frequency-domain data configuration based on the DUT length to ensure that the time-domain measurement range sufficiently covers the DUT. When the length measured in Auto mode is below the minimum limit, that limit is used to ensure measurement effectiveness. In addition, calibration must be performed after modifying the DUT Length.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

9.4.2.2.3 Stimulus Amplitude

Stimulus Amplitude (Stim.Ampl) is used to set the amplitude of the simulated step stimulus; this amplitude is directly reflected in the time-domain voltage-format response waveform. Default value 200 mV, range 1 mV to 5 V.

9.4.2.2.4 Perform Calibration

TDR provides several error-correction options—Deskew, Deskew & Loss, Electronic Calibration, and Basic Calibration—to remove the effects of test fixtures and cables; the calibration status is displayed in the bottom status bar.
Only one calibration method can be selected per calibration; the new calibration result overwrites the old one.
For detailed operations, please refer to the "Setup Wizard" chapter.

9.4.2.2.5 Initialize

After performing initialization:

- The DUT topology structure remains unchanged
- The calibration and correction data will be deleted
- Other Parameter return to default settings

9.4.2.3 More Settings

Reference Impedance (Ref.Z)
Used to set the reference impedance Z0 (Port Reference Impedance) in time-domain measurement; default value 50 Ω, range 0.001 Ω to 1000 Ω

Dielectric Constant / Velocity Factor

The dielectric constant (εr, relative permittivity) determines the propagation velocity of electromagnetic waves in the medium: v = c/√εr (1 ≤ εr ≤ 100).The larger the dielectric constant, the slower the signal propagates in the medium.
The velocity factor (VF) is the propagation velocity of electromagnetic waves in a medium (0 < *VF* ≤ 1), which is the ratio of its propagation velocity to the speed of light in vacuum

The dielectric constant (εr) and velocity factor (VF) of the cable are interrelated; changing one automatically changes the other

$\epsilon r = \frac{1}{VF^2}$Conversion Formula:

Source Power
Sets the output power level of the stimulus source; range -40 dBm to 10 dBm, default -5 dBm.

Balanced Port Configuration (Bal Port Config)
For specific configuration guidance, please refer to Balanced S-Parameter chapter.

Frequency Limits
Sets the start and stop frequency limits for time-domain measurement; the settable range is narrower than the hardware bandwidth.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

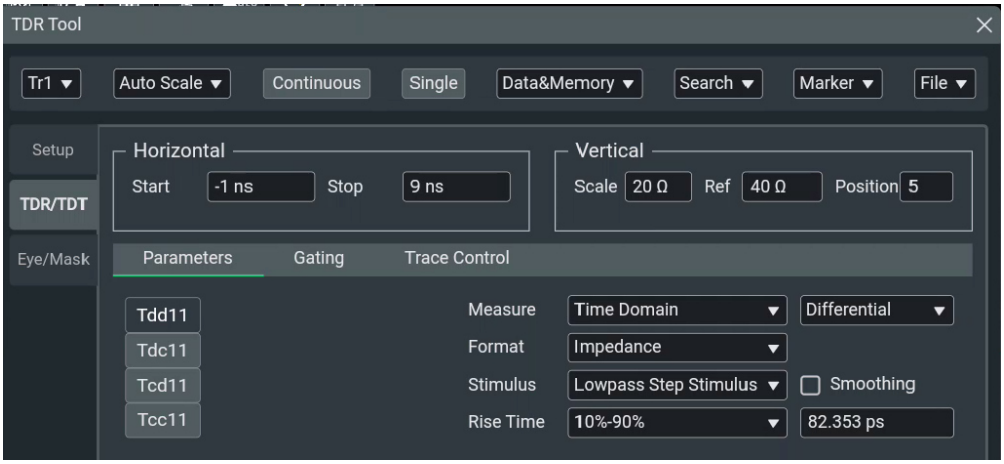
Advanced Mode
TDR measurement is in Basic Mode by default; in this mode, only Parameter configuration via the TDR menu is supported. For more advanced settings, please click **Advanced Mode** button to switch to Advanced Mode; the system will automatically pop up the system menu for operation.

9.4.2.4 Averaging Function

The Averaging function reduces trace noise and is off by default.
Sets the number of scans; after enabling the Averaging function, the results of the specified number of scan measurements will be averaged.
In averaging mode, you can set the current IF Bandwidth; please refer to the "Avg BW" chapter for settings.

9.4.3 Trace Setup

Modify the trace's test Parameter, scale, format, etc.
Configuration menu:



9.4.3.1 Set Trace Display Window

Sets the range or scale and offset of the measurement trace displayed in the window.

- **Horizontal: When**the measurement type is time-domain, you can set the start and stop range of the X-axis measurement; the unit of the Parameter automatically switches.
- **Vertical: Sets**the scale, reference level, and reference position of the trace Y-axis.

9.4.3.2 Trace Measurement Parameter

Supports setting the measurement type, format type, stimulus source type, and other Parameter of each trace.

- **Measurement Type**

The following measurement types can be set for the specified trace:

Measurement Mode	Measurement Topology Type	Measurement Type	Physical Meaning
SParameter	Single-Ended	Sxx (e.g., S11, S22)	Port Reflection Coefficient (Frequency Domain)
		Sxy (e.g., S21, S12)	Port Transmission Coefficient

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Measurement Mode	Measurement Topology Type	Measurement Type	Physical Meaning
			(Frequency Domain)
	Differential	Sddxx\Sddxy	Differential-Differential
		Scdxx\Scdxy	Common-Mode → Differential
		Sdcxx\Sdcxy	Differential → Common-Mode
Time Domain	Single-Ended	Txx (e.g., T11, T22)	Time-Domain Reflection (TDR)
		Txy (e.g., T21, T12)	Time-Domain Transmission (TDT)
	Differential	Tddxx / Tddyy	Differential Reflection (TDR)
		Tddyx / Tddxy	Differential Transmission (TDT)
		Tccxx / Tccyy	Common-Mode Reflection
		Tccyx / Tccxy	Common-Mode Transmission
		Tcd / Tdc	Mode-Conversion Time Domain

- **Format**

Supports setting the formatted display under the specified measurement type. For details, please refer to the data format chapter.

Measurement Type	Data Format
SParameter	Linear Magnitude, Smith Chart, Log Magnitude, Admittance Chart, Real Part, VSWR, Phase, Group Delay, Unwrapped Phase, Positive Phase, Imaginary Part, Polar
Time Domain	Log Magnitude, Linear Magnitude, Real Part, Impedance, Voltage

- **Stimulus Source**

The analog signal types used for time-domain conversion include: **Low-Pass Pulse** and **Low-Pass Step**.

- **Rise Time**

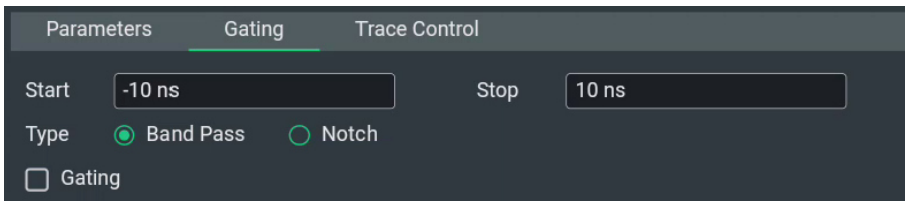
Rise time can be set only in time-domain measurement with the Low-Pass Step stimulus source. The rise time range can be 10-90% or 20-80%, default value $0.7/F_{max}$, where F_{max} is the maximum frequency set on the instrument.

- **Smoothing**

The Smoothing function is off by default; for details, please refer to the smoothing function description in the Avg BW chapter.

9.4.3.3 Measurement Gating

Supports gating operations on time-domain waveforms, enabling pass gate and notch gate to capture the segment to be retained for analysis.



Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Gate Enable**

Supports individually setting the gate for each trace; off by default. After enabling, it is displayed on the interface **Gate** marker. The position is determined by the Start/Stop settings.

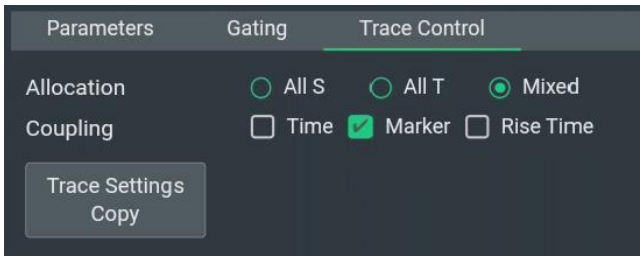
- **Gate Type**
  - Band-Pass Gate: Default mode, filters out response signals outside the gated range
  - Notch Gate: Filters out response signals within the gated range

- **Gate Start/Stop Position Setting**

Sets the start and stop positions of the gate. When setting the start or stop position, its impedance should be close to the reference impedance.
Start range: -Tp to Stop Time, where Tp is the TDR calculation period and Stop Time is the end time; default value -10 ns
Stop range: -Tp to Tp, where Tp is the TDR calculation period; default value 10 ns

9.4.3.4 Trace Control

Used to set the trace's measurement Parameter and display mode.



Trace Type
Sets the data to be plotted; the trace updates automatically after the configuration is modified.

- Mixed: (Default) Displays both time-domain and frequency-domain test data simultaneously;
- Time Domain Only: Displays only the time-domain data under the current topology
- Frequency Domain Only: Displays only the frequency-domain data under the current topology

Coupling Type
Sets the coupling Parameter for traces of the same measurement type (time-domain or frequency-domain)

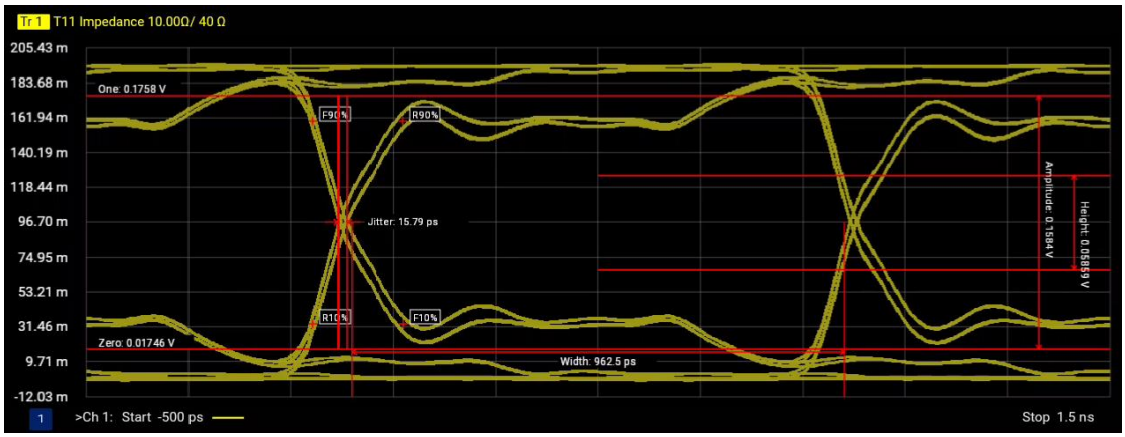
- **Time:** Supported by time-domain traces only; uncoupled by default. After enabling, all time-domain traces share the same X-axis display range.
- **Marker:** Coupled by default; all traces maintain the same marker point state and position.
- **Rise Time:** Uncoupled by default**,** After enabling coupling, all other traces use the same rise time.

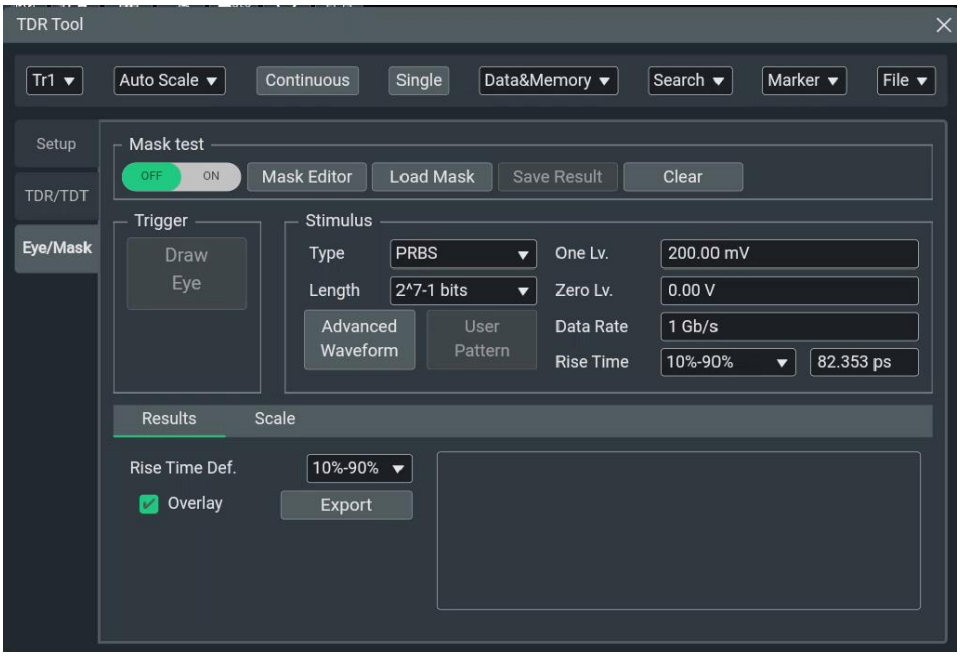
Trace Setup Copy
The trace copy function supports quickly applying the configuration parameter of a specified trace to a designated target trace.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

9.4.4 Eye Diagram/Mask Function

The instrument defines a virtual bit pattern through the built-in **Virtual Pattern Generator**, then convolves the defined pattern with the DUT's impulse response to finally generate an eye diagram based on high-precision measurement data. This function requires no additional hardware pulse pattern generator.
The eye diagram section occupies the entire interface and is not displayed simultaneously with the previous results.





9.4.4.1 Mask Test

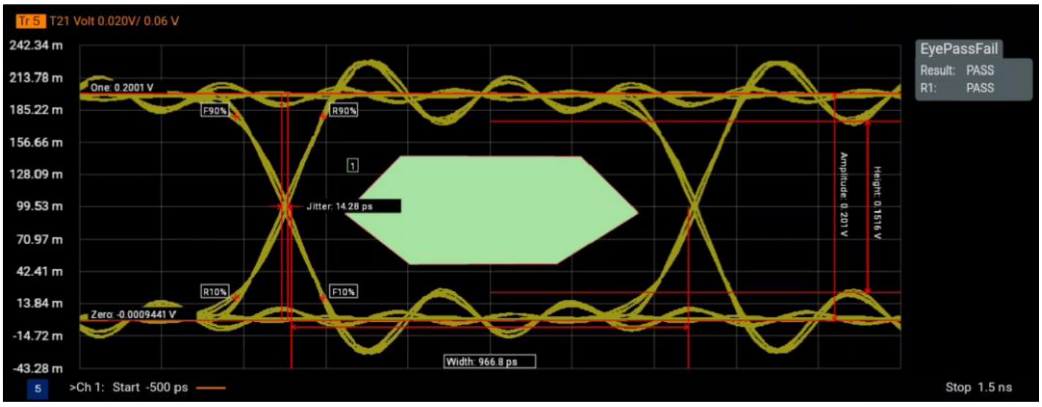
The mask test can be used to verify whether the displayed waveform conforms to the industry standard definition of electrical waveforms. To meet industry standard requirements, the input waveform must always remain outside the mask shaded area.

9.4.4.1.1 Mask Test Enable

The mask test is off by default; after enabling the mask test function, the waveform view enters the eye diagram mask test state and displays the test result.
The mask test checks whether the waveform of the specified pattern intersects the eye diagram mask;

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

if not, the test result is Pass, otherwise Fail.

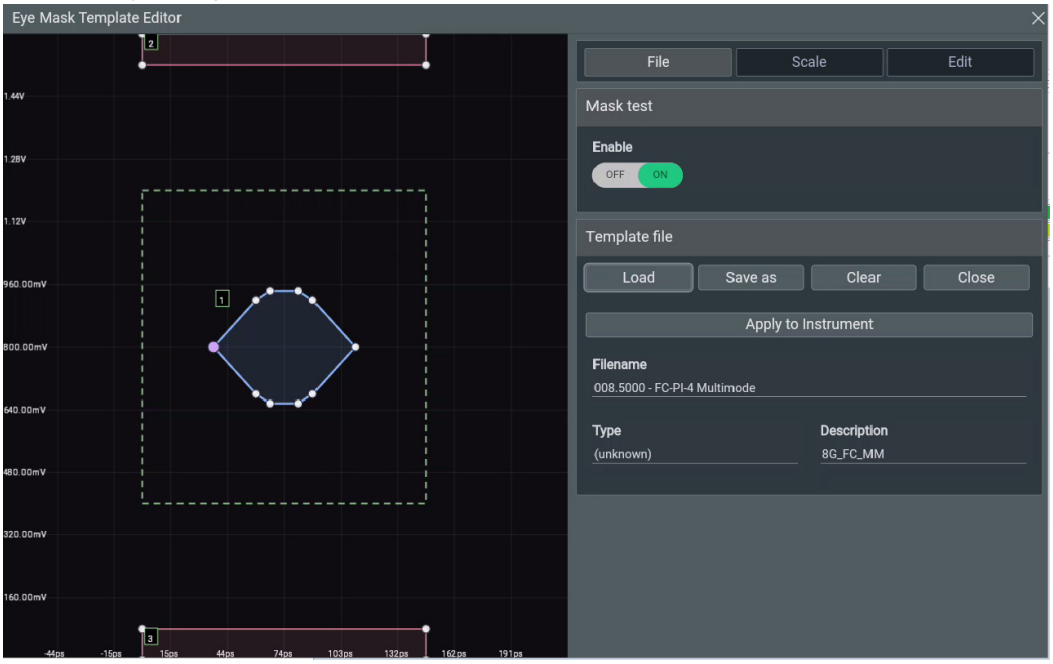


9.4.4.1.2 Load Mask

Click to enter the file management window, select a predefined eye diagram mask file (*.msk), and click **Confirm**, to complete the eye diagram mask loading.

9.4.4.1.3 Edit Mask

Click to enter the "Eye Diagram Mask Editor" as follows.



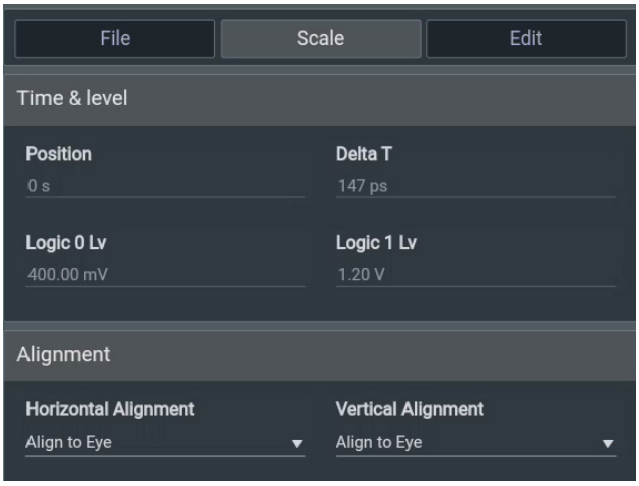
The load or draw area on the left consists of two parts: the first is a 10×10 grid of eye diagram dashed reference lines on a black background, representing the reference template range; the second is a blue semi-transparent mask, representing the mask position.
Reference system: The horizontal axis of the reference system is time, and the vertical axis voltage.
Blue mask: The drawing area is drawn sequentially from top to bottom in the order defined by the vertices, and finally closed.

Eye Diagram Mask File
View the file name, type, and description of the currently used eye diagram mask file.
Supports reloading the mask file, saving as the current file, clearing the current file, closing the mask editing menu, applying the current file to the instrument, and enabling/disabling the eye diagram mask

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

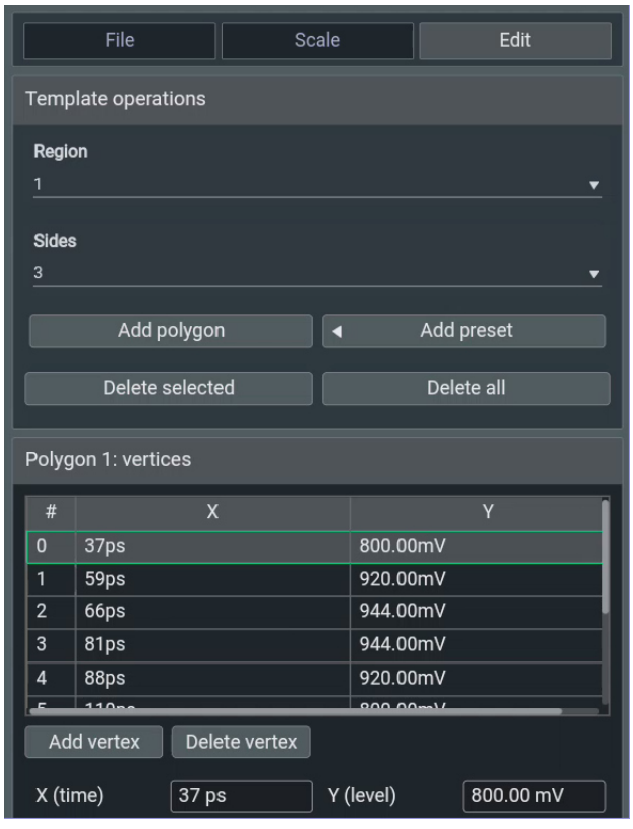
measurement operation.

Zoom Eye Diagram Mask



- Position: The x-coordinate of the left boundary of the reference rectangle.
- Δ Time: The width of the abscissa of the reference rectangle, i.e., the length of time.
- Logic 0 Level: The voltage position of the lower boundary of the rectangle.
- Logic 1 Level: The voltage position of the upper boundary of the rectangle.
- Align: Auto-scaling by default; selecting Manual supports dragging the eye diagram mask on the left to change its shape horizontally or vertically.

Edit Eye Diagram Mask



- **Select Area:** Supports multiple eye diagram masks; select the area to choose the eye diagram mask currently being edited.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Delete Mask:** Click the corresponding button to delete the specified mask, or all eye diagram masks.
- **Number of Sides:** You should first set the number of sides of the polygon, then click the "Add Polygon" button to add an eye diagram mask.
- **Edit Polygon**
  1. Before adding a polygon, you can preset it.
  2. First select the preset area: inner-diamond, upper-side band, lower-side band, outer-frame rectangle.
  3. The polygon vertices contained in the selected area are displayed in the polygon vertex list below.
  4. Select a specified vertex in the vertex list to modify its coordinates in the X (time) and Y (level) input boxes below. Or directly drag the polygon to change its shape.
  5. You can also directly delete the selected vertex, or add a vertex after selecting one.

Save Result
Click to enter the file management window to store the eye diagram mask test result data to a specified path.

Clear
Click to pop up a confirmation prompt window, and select Confirm button, to clear the eye diagram mask data.

9.4.4.2 Trigger

- Triggers drawing of the eye diagram for the specified trace.
- Only supports drawing of time-domain transmission result traces, such as T21. Otherwise triggering is not allowed
- Each click triggers drawing only once, so the Run/Stop signal will not trigger eye diagram drawing at this time.
- Eye diagram drawing may take a long time.

9.4.4.3 Stimulus

9.4.4.3.1 Basic Parameter

Pattern
Supports the following specified types of pattern sequences.
PRBS: Default option, pseudo-random bit sequence, used for eye diagram, jitter, bit error rate, and general signal integrity testing;
K28.5: The 8b/10b standard comma control character pattern defined by IBM; the 20-bit sequence is 00111110101100000101, used for synchronization and control signal testing of high-speed serial protocols.

- USER: User-defined;
- STAT: Statistical bit pattern, generated by statistical calculation of jitter metrics.

Length
Length selection is supported only when the pattern is PRBS. The selectable lengths include:
2^3-1 bits, 2^5-1 bits, 2^7-1 bits (default), 2^9-1 bits, 2^10-1 bits, 2^11-1 bits, 2^15-1 bits.

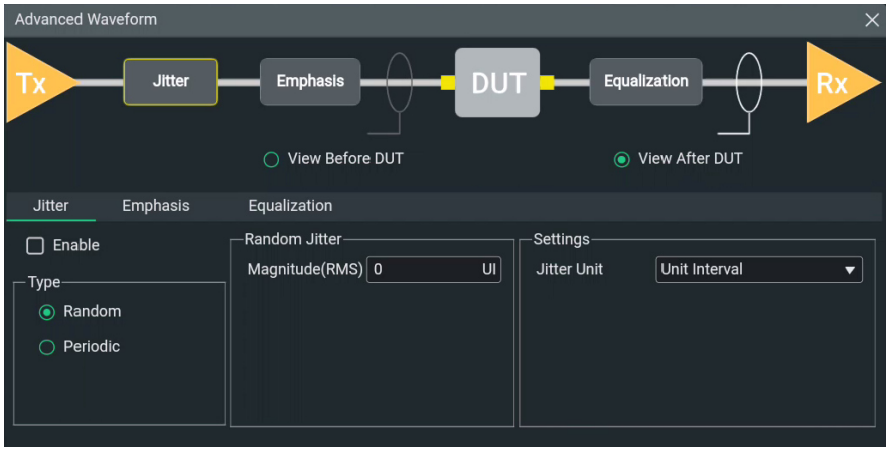
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

The longer the length, the longer the measurement time.

Bit Stream Parameter

- **Logic 1 Level:** Sets the voltage value of Logic 1. Range -5 V to 5 V, default 200 mV.
- **Logic 0 Level:** Sets the voltage value of Logic 0. Range -5 V to 5 V, default 0 V.
- **Data Rate: Range:** 1/(2·DUTLength) to (4/5)·F_{max}, default value 1 Gb/s.
- **Rise Time:**
  10% to 90% (default), range 446 /F_{max} to 1/ (20*DUTLength), default value 700 / F_{max}.
  20% to 80%, range 317 / F_{max} to 1/(20*DUTLength), default value 700 / F_{max}.
  F_{max} is the maximum frequency supported by the instrument.

9.4.4.3.2 Advanced Waveform



Display Before DUT / Display After DUT: Supports viewing the eye diagram of the original pattern before the DUT and the eye diagram after the DUT.

Jitter

Adds random jitter or periodic jitter to the edge time of the stimulus source signal to degrade the input signal, used to evaluate the link's sensitivity to clock uncertainty and thus assess the link margin.

The jitter injection function is off by default. Check **Enable**, and the jitter icon in the diagram lights up. Supports setting the following two types of jitter

- **Random Jitter**
  Supports setting the jitter amplitude; default value 0, range 0 to 0.25 UI, where UI is the unit interval.
- **Periodic Jitter**
  Supports setting the jitter amplitude; default value 0, range 0 to 1 UI, where UI is the unit interval.
  Supports setting the jitter frequency; default value 1 MHz, range 0 to data rate / 1 kHz.

Supports setting the jitter unit:

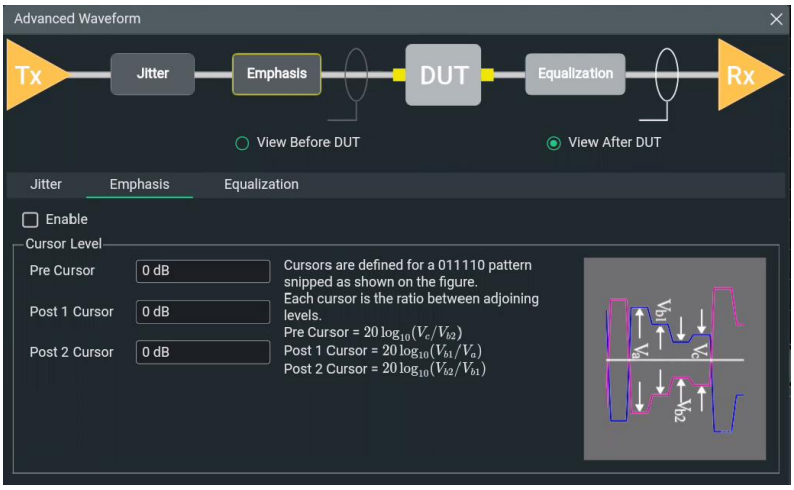
- Unit Interval, equal to 1/data rate, equivalent to 1 UI;
- Second, converts the unit interval to time display

Pre-emphasis

Pre-emphasis is a way to perform "frequency compensation" on the FFE (Feed-Forward Equalizer)

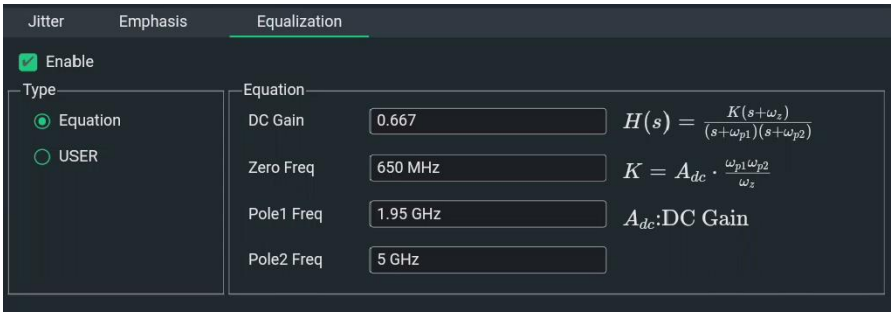
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

waveform at the transmitter, enhancing high-frequency components to offset the channel's high-frequency loss, so that the receiver sees steeper edges and a larger eye opening.



The pre-emphasis function is off by default. Check **Enable**, and the pre-emphasis icon in the diagram lights up. Changing the enable state triggers a redraw of the eye diagram.
The cursor levels support configuring the gain coefficients of the FFE (transmit equalizer) taps (Pre/Post1/Post2), used to control the pre-emphasis strength, simulate the high-frequency compensation of actual transceivers, and thereby observe the equalized receiver eye diagram.
The default values of the Pre cursor, Post1 cursor, and Post2 cursor are 0, with a range of -20 dB to 20 dB

Equalization
When a signal travels a long distance and reaches the receiver (Rx), it may be severely distorted (eye closed).
Equalization is a signal processing technique performed at the receiver (Rx) (such as CTLE (Continuous-Time Linear Equalization) or DFE (Decision-Feedback Equalization)). It acts like an amplifier/filter, specifically amplifying high frequencies and suppressing noise, attempting to recover the original signal.



The equalization function is off by default. Check **Enable**, and the pre-equalization icon in the diagram lights up. Changing the enable state triggers a redraw of the eye diagram. However, enabling equalization does not affect the S-Parameter measurement result.
Equalization can be configured in the following two ways

- **Formula**

The equalization calculation formula is shown in the figure above; each Parameter is described as follows:

- DC Gain: Default value 0.667, settable range 0 to 10
- Zero Frequency: Provides a high-frequency peak; default value 650 MHz, settable range 0 Hz

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

to 76 GHz

- Pole 1 Frequency (*fp*1): The first pole frequency (low-frequency pole); default value 1.95 GHz, settable range 0 Hz to 76 GHz
- Pole 2 Frequency (*fp*2): The second pole frequency (high-frequency rolloff pole); default value 5 GHz, settable range 0 Hz to 76 GHz
- Requirements: fp1 < fp2

- **Custom**

Supports manually setting the equation or loading a file by the user.
If the file format or naming is incorrect, loading will fail.

9.4.4.3.3 User Pattern

When the pattern is USER, the following bit-stream editing box can be entered.



- Supports the user manually entering a sequence of '0' and '1' in the bit-stream editing box.
- Load: Supports the user importing a sequence file; no specific format, just 0/1 characters, UTF-8 encoding.
- Save as: Supports saving the user-edited sequence as a txt text format.
- Update: Pushes the string stream in the current editing box into the background data
- Clear: Supports clearing the user-edited content.
- Close: After closing the editing box, the file is loaded successfully; it is recommended to display it on the main interface.

9.4.4.4 Result

Displays the measurement results, including the following Parameter

- Level 1: High level
- Level 0: Low level
- Amplitude
- Average Level
- Eye Height
- Rise Time
- Fall Time
- Jitter (RMS)
- Jitter Peak-to-Peak
- Eye Width
- Duty Cycle Distortion
- Signal-to-Noise Ratio
- Eye Opening Factor
- Duty Cycle Distortion (%)
- Crossing Percentage

Select Rise Time

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Optional: 10-90% (default) or 20-80%. Changing the rise time requires clicking the trigger button to redraw the eye diagram for the result to update.

Parameter Overlay Display
Enabled by default; Jitter p-p, Logic 0 Level, Logic 1 Level, Amplitude, Height (V), Width, and other Parameters are overlaid on the grid-area waveform. Modifying the Parameter overlay display does not require separately triggering eye diagram drawing.

Export
Enter the file management menu to save the current test results as a *.csv file to a specified directory.

9.4.4.5 Scale

The eye diagram display is in Auto mode by default, automatically adjusting the Y-axis scale (Scal/Div) and the Y-axis offset value (Offset).
Manual setting is also supported

- **Scale/Div:** The settable range is (10^{-18}) V to 5 V, default 200 mV.
- **Offset:** The settable range is -5 V to 5 V, default 0 V.

10 Data Format

The data format is a way of presenting measurement data in the form of graphs. Select the desired data format that matches the test device.

In the **Format** menu, click the desired data format to set the format for the current active trace.

10.1 Rectangular Display Formats

Rectangular display formats (also known as Cartesian coordinates, X/Y, or rectilinear). The rectangular display is especially useful for displaying the frequency response information of the test device.

- **X-axis:** displays stimulus data (frequency, data, or time).
- **Y-axis:** displays measurement data.

Log Mag

Displays the magnitude of the signal, without phase. The Y-axis is represented in dB, wrapping a large range of values for easy scaling, convenient to present large variations of the signal. It is applicable to the signal with a large magnitude or large variations as the Log Mag format can better display the details of the signal.

Typical application: used in return loss, insertion loss or gain measurement.

Lin Mag

Y-axis displays positive values of the signal magnitude only, without unit. It is applicable to the calibrated measurement with the Watts (W) or Volt (V) as the unit. For the measurement concerning the power or voltage item, if you have completed the calibration, using this format can directly obtain the magnitude value in linear form.

Typical measurement: reflection and transmission coefficients (magnitude), time domain transfer.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Group Delay

Displays the signal transmission (propagation) time through a device. Y-axis: Time in seconds.

Typical measurement: As the transmission speed of the frequency components of the signal in the medium is different, the time delay occurs, resulting in phase deviation. The phase deviation between various frequency components changes with the frequency, causing signal distortion. If the group delay of a system is a constant value for all the frequency span, it indicates that the system is an all-pass system, meaning that the transmission speed at all frequencies is the same, which will not cause signal distortion.

Phase

- **Measurement principle:** measures the phase of the signal relative to the calibration reference plane with a range of +/- 180 degrees.
- **Display information:** displays only phase, without magnitude.
- **Y-axis unit:** phase in degrees.
- **Trace characteristics:** When phase values exceed ±180°, the trace wraps every 180 degrees to facilitate observation and analysis of phase variations within the limited display range.

  This prevents large discontinuities in the displayed phase values, making it easier to observe phase variation and scaling, better for detailed analysis of the phase variation.
- **Typical measurement:** It mainly detects the deviation of the signal from the linear phase, describing the linear variation of the phase along with the time or frequency during system transmission or processing.

Unwrapped phase

Similar to the phase format, but without 180 degrees wrapping. Phase is unwrapped by comparing the phase from one data point to the next. If the phase difference between two points is greater than 180 degrees, or if the phase of the first data point is greater than 180 degrees from DC, then the phase measurement is probably inaccurate.

Positive Phase

Displays the phase wrapped from 0 to +360 degrees. That is, the phase value will not be a negative value. If it exceeds 360 degrees, it will restart from 0 degree.

SWR

Displays reflection measurement data calculated from the formula (1 + ρ)/ (1 - ρ), where ρ is reflection coefficient. It is valid only for reflection measurements, used for analyzing the relevant characteristics of the reflection signal. The Y-axis is unitless (U) for ratioed measurements.

Typical measurement: Standing Wave Ratio (SWR) is used to evaluate the match between transmission line and load. It is an important evaluation parameter for impedance match in RF and microwave systems. When SWR = 1, it indicates that it is totally matched, which is considered an ideal condition.

Real

Displays only the real (resistive) part of the measured complex data. It can be positive and negative

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

values. The Y-axis is unitless.

Typical measurement: time domain, auxiliary input voltage signal for service purposes.

Imaginary

Displays only the imaginary (reactive) part of the measured data. It can be positive and negative values. The Y-axis is unitless.

Typical measurement: impedance for designing matching network.

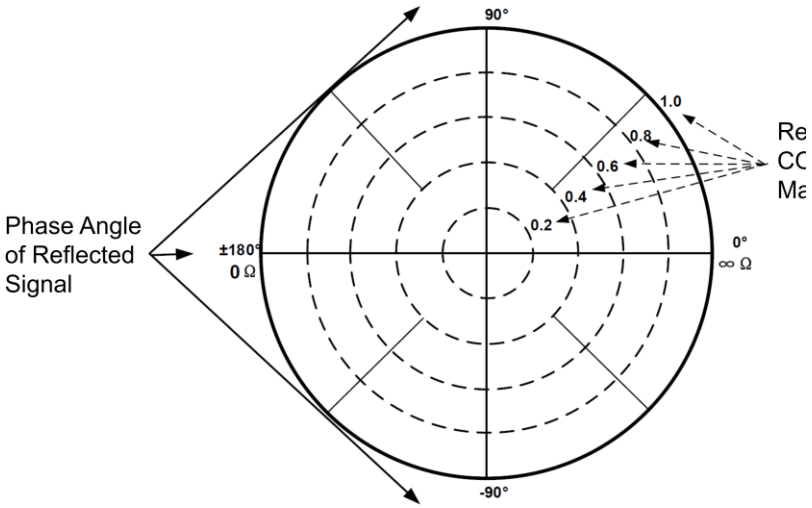
10.2 Polar

Polar format is used to view the magnitude and phase of the reflection coefficient (Γ) from the S11 or S22 measurement.

You can use markers to display the following information.

- Linear magnitude (in units) or log magnitude (in dB)
- Phase (in degrees)

The distance between the polar trace and the polar origin represents the magnitude (linear) of the measurement results. The position to the origin represents magnitude (linear); the angle deviation from the positive X-axis counterclockwise represents phase.



- The dashed circles represent reflection coefficient. The outermost circle represents a reflection coefficient (Γ) of 1, or total reflected signal. The center of the circle represents a reflection coefficient (Γ) of 0, or no reflected signal.
- The radial lines show the phase angle of reflected signal. The right-most position corresponds to zero phase angle, (that is, the reflected signal is at the same phase as the incident signal).
- Phase deviation of 90°, ±180°, and -90° correspond to the top, left-most, and bottom positions on the polar display, respectively.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

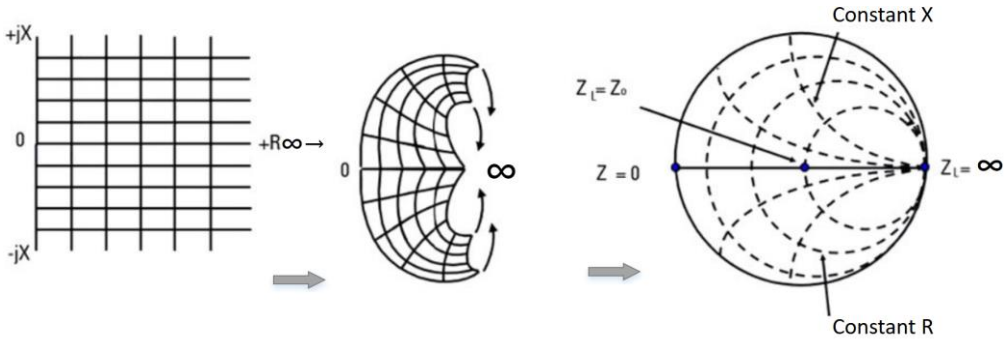
10.3 Smith

The Smith chart is a tool used to analyze the impedance and reflection coefficient graphically. In a Smith chart, the rectilinear impedance plane is reshaped to form a circular grid, mapping the complex reflection coefficient (Γ) to the test device's impedance.

With the Smith chart, you can read the series resistance (R) and reactance (X) of the test device, i.e., (R + jX).

- Resistance (expressed in Ω)
- Reactance as an equivalent capacitance (expressed in *F*) or inductance (expressed in *H*)

The Smith Chart is widely used in the design, debugging, and troubleshooting of the impedance match in the RF/microwave circuit.



Rectilinear Impedance Plane　　Smith Chart

- Every point on the Smith Chart represents a complex impedance made up of a real resistance (R) and an imaginary reactance (*R + jX)*.
- The horizontal axis (the solid line) is the real portion of the impedance - the resistance. The center of the horizontal axis always represents the system impedance. To the far right, the value is infinite ohms (open). To the far left, the value is zero ohms (short).
- The dashed circles that intersect the horizontal axis represent constant resistance.
- The dashed arcs that are tangent to the horizontal axis represent constant reactance.
- The upper half of the Smith chart is the area where the reactive component is positive and therefore inductive.
- The lower half of the Smith chart is the area where the reactive component is Negative and therefore capacitive.

10.4 Inverted Smith

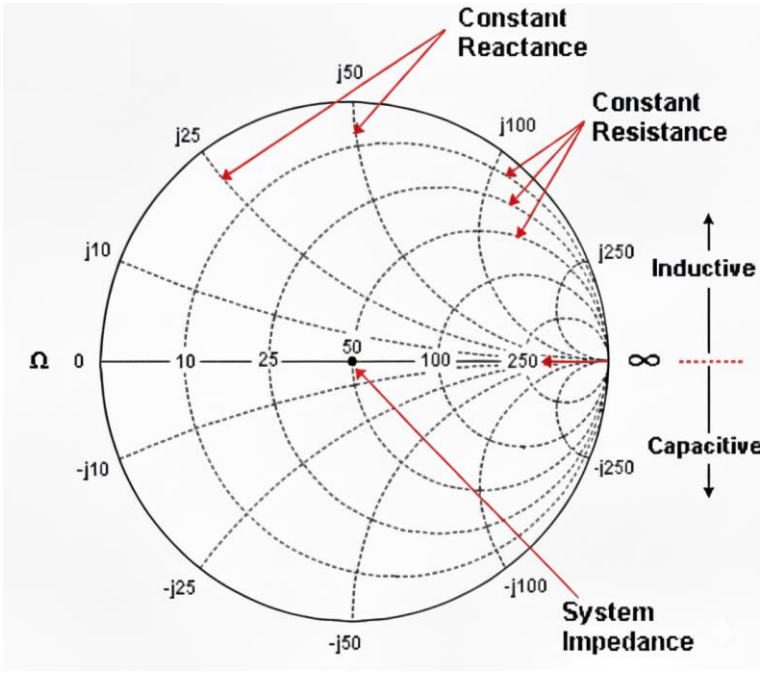
Inverted Smith also known as Admittance, is a tool dedicated for admittance analysis. It is similar to the standard Smith chart, but it analyzes the admittance rather than resistance.

The core principle of the inverted Smith Chart is to convert "impedance analysis" into "admittance

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

analysis", where admittance Y = G + jB (G is conductance, B is susceptance) is the reciprocal of impedance.

- The plot graticule is reversed right-to-left.
- Admittance (in units of siemens) instead of resistance.

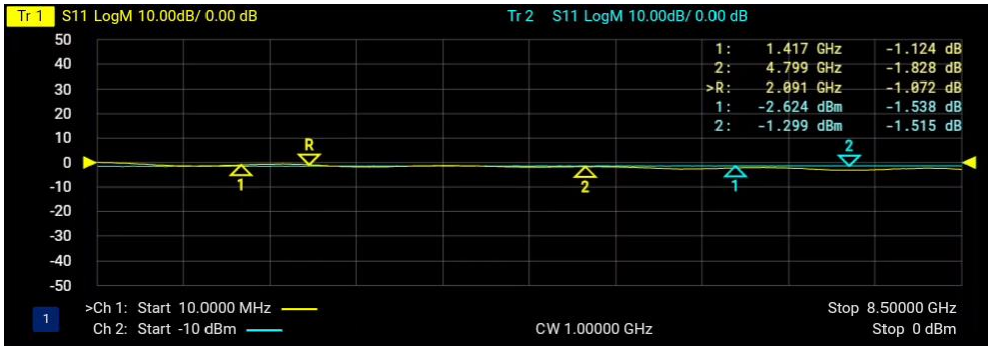


11 Marker

Marker is a triangle sign, which is used for marking the point on the trace. Markers provide readouts of the frequency of each point on the trace and measurement values. The values vary based on the measurement parameters of the trace and the trace format.

11.1 Marker 1-7/Marker 8-15

Up to 15 normal markers (Marker1 to Marker15) and one reference marker are available for each trace. By default, Marker 1 is selected.



Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Add/Delete a Marker

Check the checkbox of the specified marker, and the checkbox is highlighted in green, then the marker is added on the trace. Uncheck the checkbox of the specified marker to remove the marker from the screen.

The added marker will automatically become the current active marker. The marker appears above the current trace, displayed as "∇+marker number". All of the other markers inactive on the same trace appear below the trace, displayed as "Δ+marker number". Click the specified triangle marker icon on the trace in the window to make it become the active marker. You can also click the specified marker from the drop-down list of markers to select it as the active marker.

The marker data are displayed at the right section of the window where the trace is located. One window can display data of up to 20 traces. The marker data contain the following information.

- Marker number. If ">" is added before the marker number, it indicates that the marker is currently active.
- Trace where the marker is located. You can identify the trace by recognizing the color of the marker. The marker color is the same as the trace color.
- Marker's stimulus data (X-axis) and response data (Y-axis). The data format of the marker is the same as that of trace where the marker is located.

Move the Marker

Move the marker with the following methods. During moving, the marker will always move along with the trace. The marker icon can be overlapped with each other.

- Click the selected marker to set its X-axis value with the numeric keypad. Then the marker is moved to the specified position on the trace.
- Click the selected marker to set its X-axis value with the knob. Then the marker is moved to the specified position on the trace.
- Select the marker, then drag the marker to move it to the specified position on the trace. This method is not allowed when the data format of the trace is "Smith", "Inverted Smith", or "Polar".

The marker icon can be overlapped. Move the marker left and right, and the marker will always move along the trace.

Reference Marker

Check the checkbox of "Reference", the checkbox is highlighted in green, then a reference marker is added on the trace. Uncheck the checkbox of "Reference", then the reference marker is removed from the screen.

By default, the reference marker is created at the position where the current normal marker is located, but opposite with each other. If no normal marker is created on the current trace, then the reference marker is created in the central position of the trace, displayed as "R". The reference marker data (marked as ">R") are displayed at the right section of the window where the trace is located. The method on how to move the reference marker is the same as that for the normal marker.

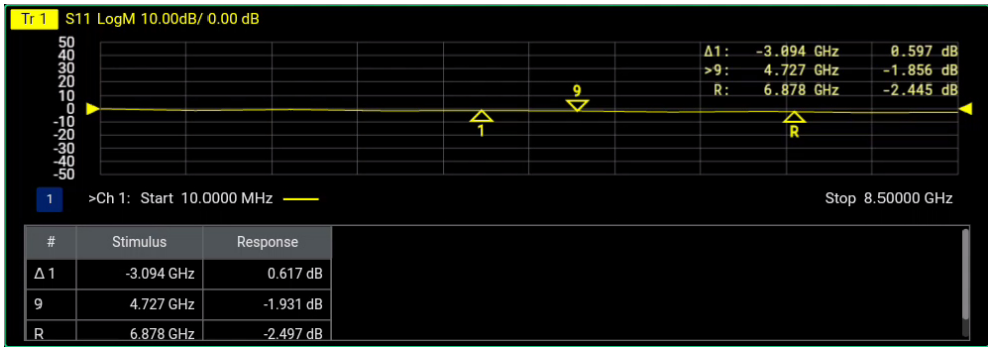
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

11.2 Marker Setup

Delta

Used to measure the difference between the reference point and a certain point on the trace, including the stimulus delta (X-axis value) and response delta (Y-axis value). By default, Delta is disabled. The delta marker value is displayed at the upper-right corner of the window or displayed in the marker table. The delta marker is displayed as "Δ+marker number".

- If a reference marker already exists for the current trace, when "Delta" is enabled, it calculates the difference between the current active marker and the reference marker. If the reference marker is the current active marker, then the Delta function is disabled.
- If no reference marker exists on the current trace, when "Delta" is enabled, create a reference marker in the center position of the trace, then adjust the positions of the reference marker and the active marker.



- When you change the position of the Delta marker, the position of the reference marker remains unchanged, but the frequency (or time) difference between the two markers will change along with it.
- The frequency (or Power) difference between the two markers and the measurement values difference Δ are displayed on the screen.

Discrete

Used to display values at only the discrete points where data is measured.

- OFF: By default, it is disabled. The marker readouts of all the data points measured on the trace are displayed.
- ON: Displays values at only the discrete points where data is measured. The marker readouts of data values that are interpolated from the data points will not be displayed. The discrete points where data is measured are determined by Sweep Points.

Type

- **Normal:** The marker has a fixed stimulus position (X-axis value), and by default, the response position (Y-axis value) of the marker changes with the trace data.
- **Fixed:** Causes the marker to have a fixed X-axis and Y-axis position based on its placement on

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

the trace when it was set to fixed. It does NOT move with trace data amplitude. It can be scrolled left and right on the X-axis by changing the marker stimulus value.

Format

The newly created marker uses the trace data format by default. If you want to modify the data format of the marker, the following formats are available.

Trace Default, Linear Magnitude, Log Magnitude, Phase, Group Delay, Real, Imaginary, SWR, Log/Phase, Lin/Phase, Real/Imag, R + jX, and G + jB. For details, refer to Data Format.

Coupling Mode

The coupled marker function causes markers on different traces to line up with the markers on the selected trace. When you move the specified marker, all the other markers that are coupled with the marker will be moved together.

Markers are coupled by marker number. If different traces have the same x-axis domain (e.g., frequency or power), then coupling occurs to the markers with the same marker number on these traces. Trace markers in a different x-axis domain will not be coupled.

The available coupling modes include:

- **OFF:** disables the marker coupling.
- **Channel:** By default, a marker on one trace is coupled to the same numbered markers on traces which share the same channel number as the original trace.
- **All:** A marker on one trace is coupled to the same numbered markers on all channels, all windows, and all traces.

Marker Table

- OFF: By default, it is disabled. The marker table will not be displayed.
- ON: when it is enabled, the marker table displaying all the marker data on the current active trace is displayed below the trace window, including the marker number, stimulus value, and response value.

#	Stimulus	Response
1	1.641 GHz	-0.857 dB
2	2.991 GHz	-1.643 dB
R	5.496 GHz	-2.445 dB

All Off

Click **All Off** to disable the display of all the markers on the current active trace. The coupled markers are also disabled.

11.3 Marker Display

Symbol Above

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- OFF: By default, the active marker is displayed above the trace and the non-active marker is displayed below the trace.
- ON: All the marker symbols are displayed above the trace. The active marker is filled solid.

Symbol

Sets the marker symbols. By default, it is Triangle. You can also select "Flag" or "Line".

- "Line" is not available to choose when the data format of the trace is "Smith", "Inverted Smith", or "Polar".
- Symbols can be set independently for each window.

Active Trace Readouts Only

- OFF: By default, all the marker readouts of all the traces are displayed at the upper-right corner of the window.
- ON: When it is enabled, only the marker readouts of the current active trace are displayed at the upper-right corner of the window.

Show Readouts

- ON: By default, the marker readouts of the traces are displayed at the upper-right corner of the window.
- OFF: When it is enabled, marker readouts of the traces will not be displayed in the window.

Readouts Per Trace

Sets the quantity of marker readouts to show in the window for each trace. By default, up to 5 readouts per trace and 20 readouts for each window can be displayed.

11.4 Marker Functions

The marker functions allow you to make stimulus settings with the current active marker. You can make a quick set of the following values.

- **Marker -> Start:**

  Sets the position (X-axis) of the current active marker to the start value of the sweep.
- **Marker -> Stop:**

  Sets the position (X-axis) of the current active marker to the stop value of the sweep.
- **Marker -> Center:**

  Sets the position (X-axis) of the current active marker to the center value of the sweep (middle value between the start value and stop value of the sweep).
- **Marker -> Ref Level:**

  Sets the response value (Y-axis) of the current active marker to the Reference Level of the current trace.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Marker -> Span:** Sets the span between the current active marker and Reference Marker to the sweep span. It is unavailable if there is no reference marker.

- **Marker -> Delay:**

  The phase slope at the active marker stimulus position is used to adjust the line length to the receiver input. This effectively flattens the phase trace around the active marker.

  Additional Electrical Delay adjustments are required on devices without constant Group Delay over the measured frequency span. This feature adds phase delay to a variation in phase versus frequency; therefore, it is only applicable for ratioed measurements.

- **Marker -> CW Frequency:** When the X-axis represents frequency, set the position of the active cursor as the CW frequency. After modifying the CW Frequency with this operation, enable Power Sweep to activate the updated CW frequency setting. This function has no effect if the current X-axis is not frequency-based.

12 Search

Performs search on the current active trace based on the set conditions and marks it with the marker. Before using the search function, create a marker for the current active trace, otherwise, some of the search function menus may be grayed out and disabled.

12.1 Main

Max

Click **Max**, then the current active marker moves to the point with the max. measurement value on the current active trace.

Min

Click **Min**, then the current active marker moves to the point with the min. measurement value on the current active trace.

Search Domain

Defines the marker on the trace to search within the specified span. Each marker can be configured with a search domain, and multiple markers can share one search domain.

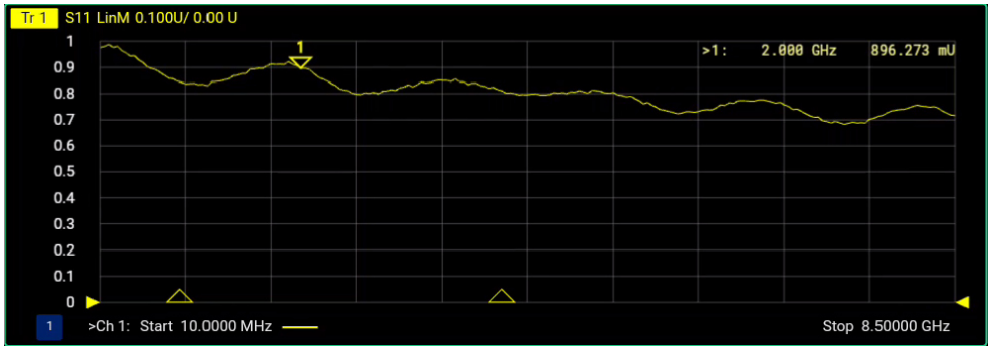
Click the drop-down button of **Search Domain** to select the specified span.

- **Full Span:** By default, it is full span. That is the whole X-axis, which is the same as the sweep range of the trace itself.

- **User 1 to User 16:** searches within the user-defined span. The X-axis of the marker is defined within the set search domain.

  At most, 16 user-defined search domains can be set. You need to set Domain Start and Domain Stop for each user-defined search domain. Each user-defined domain is independent of each other, and the user-defined domains can overlap with each other. When the search domain is set to the user-defined domain other than "Full Span", a pair of smaller triangles appear automatically in the X-axis to identify the search domain of the current active marker.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Tracking

When you set the tracking condition, the VNA will perform the search operation based on the current search conditions and search domain. Once the target point is found, the marker will locate it immediately. The marker position will move along with the update of the trace data.

Click the drop-down button of **Tracking** to set the tracking conditions for the current active marker. The tracking conditions include:

- **Off:** By default, tracking function is disabled.
- **Max:** The marker moves to the maximum response data value (Y-axis value) within the search domain.
- **Min:** The marker moves to the minimum response data value (Y-axis value) within the search domain.
- **Peak:** Related to the peak search type. For details, refer to Peak.
- **Target:** Related to the target search type. For details, refer to Target.
- **Multi Peak:** The marker moves to the multi-peak search result position on the trace based on the search domain. For details, refer to Multi Peak & Target.
- **Multi Target:** The marker moves to the multi-target search result position on the trace based on the search domain. For details, refer to Multi Peak & Target.

Tip:

- For different markers on the same trace, they can be set to Max, Min, Peak, and Target tracking. If they are set to Multi Peak or Multi Target tracking, then other tracking settings are invalid. all the markers share the same tracking.
- The markers that have enabled the tracking function will not be affected by marker coupling.

12.2 Peak

In the **Search** menu, click **Peak** to enter the peak search setting sub-menu. Before executing the peak search, set the search condition and search type first.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Peak Right Search/Peak Left Search

- Click **Peak Left Search** to search for the nearest peak which is located at the left side of the current peak and meets the peak search condition.
- Click **Peak Right Search** to search for the nearest peak which is located at the right side of the current peak and meets the peak search condition.

Next Peak Search

Click **Next Peak Search** to search for the peak whose amplitude on the trace is next to that of the current peak and meets the peak search condition. Once found, the current active marker will move to the position and marks it.

Peak Polarity

- Positive (default): searches for the positive peak, which refers to a peak whose measured value is greater than those of the measurement points on either side of it.
- Negative: searches for the negative peak, which refers to a peak whose measured value is smaller than those of the measurement points on either side of it.
- Both: A peak whose measured value is smaller and greater than those of the measurement points on either side of it. Both the positive peak and negative peak are considered as valid peaks.

Peak Threshold

Related to the "Peak Polarity" setting.

- For the positive peak (positive polarity), only the peaks whose measured values are greater than threshold can be considered as valid.
- For the negative peak (negative polarity), only the peaks whose measured values are smaller than threshold can be considered as valid.
- When the peak polarity is set to Both, the threshold value is not used to judge the valid peak.

Peak Excursion

The peak excursion is the difference between the peak measured value and the measured value of the adjacent peak of the opposite polarity. Only the peaks whose peak excursion value is greater than the set peak value can be considered as valid.

Tracking

For the tracking function, refer to descriptions about tracking function specified in Main section.

Peak Search

First set the search conditions for the valid peak, then click **Peak Search** to perform the peak search. Once the valid peak is found on the trace, the current active marker will move to the position and marks it. When no peak meets the specified peak search condition is found, the position of the marker remains unchanged.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

12.3 Target

The target search function searches for a target that matches the pre-defined target value and transition types (positive, negative, or both) and then moves the marker to that target.

In the **Search** menu, click **Target** to enter target search setting sub-menu. Before executing the target search, set the search condition and search type first.

Target Right/Left Search

- Target Right Search: The marker moves to the next valid target to the right of the first target encountered.
- Target Left Search: The marker moves to the next valid target to the left of the first target encountered.

Target Value

The target value refers to the searched response value (Y-axis value). If the response value of a certain point on the trace equals the target value, then the data point is the target point to be searched for.

Target Transition

- Positive: The measured value at the right side of the target point is greater than that at the left side of the target point.
- Negative: The measured value at the right side of the target point is smaller than that at the left side of the target point.
- Both: Both the positive and negative transition types are considered as valid target. It is the default target transition.

Tracking

For the tracking function, refer to descriptions about tracking function specified in Main.

Target Search

After configuring the target value, target transition, and search domain, click **Target Search** to perform the target search. Once the target point is found on the trace, the current active marker will move to the position and marks it.

12.4 Multi Peak & Target

Multi Peak Search

The multi peak search function enables you to display markers on multiple peaks on traces based on the multi peak excursion and multi peak polarity settings. Before executing the multi peak search, set the following parameters to define a valid peak.

- **Multi Peak Threshold:** Minimum amplitude (in dB). To be considered valid, the peak must be above the threshold level. The valley on either side can be below the threshold level.
- **Multi Peak Excursion:** The vertical distance (in dB) between the peak and the valleys on both

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

sides. To be considered a peak, data values must "fall off" from the peak on both sides by the excursion value.

- **Multi Peak Polarity:** Positive, negative, and both. For details, refer to Peak Polarity in Peak Search section.

After configuring the above parameters, click **Multi Peak** to start searching from the start position of the search domain to the right to search out all the valid peaks and marks them with the marker.

Up to 16 markers (15 normal markers and 1 reference marker) can be displayed on a trace. When the number of valid peaks exceeds 16, only the first 16 valid peaks can be searched to be displayed.

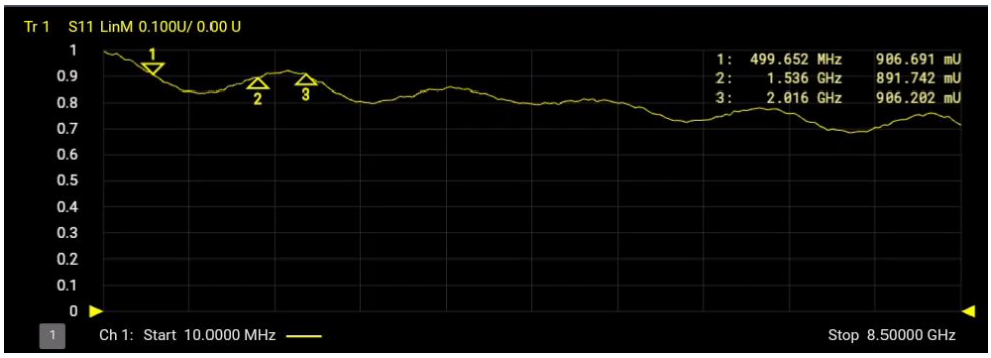
Multi Target Search

The multi target function searches for a target that matches the pre-defined multi target value and multi target transition. Before executing the multi target search, set the following parameters.

- **Multi Target Value:** sets the response value (Y-axis value) of the search target. The data format and unit of the target value are the same as that of the current trace.
- **Multi Target Transition:** Positive, negative, and both. For details, refer to Target Transition in Target Search section.

After configuring the above parameters, click **Multi Target** to start searching from the start position of the search domain to the right to search out all the valid targets and marks them with the marker.

Up to 16 markers (15 normal markers and 1 reference marker) can be displayed on a trace. When the number of valid targets exceeds 16, only the first 16 valid targets can be searched to be displayed.



Tracking

The tracking function causes the marker to search for the specified search condition with each new sweep. By default, it is OFF.

When the tracking function is enabled, the search begins with the first sweep based on the current search type and domain information. Therefore, before using the data, ensure that the search condition is in the desired state. If tracking is selected for a marker, you are not allowed to manually modify the stimulus settings for the marker.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

12.5 Band-Pass/Band-Reject

The band-pass/band-reject function is used to automatically analyze band-pass filters and band-reject (notch) filters, automatically calculating indicators such as center frequency, bandwidth, cutoff point, Q value, and insertion loss, and displaying the search results in the window.



Parameter	Meaning
Low	Low Cutoff Frequency
High	High Cutoff Frequency
BW (Operating Bandwidth)	BW=High−Low
Center (Center Frequency)	Center= (Low+High)/ 2 The center frequency of the high and low cutoff frequencies, not the frequency of the resonant peak itself;
Q (Quality Factor)	Q=Center/BW Characterizes the device's frequency-selectivity; the larger the Q, the better the frequency-selective characteristic
Loss (In-Band Insertion Loss)	The transmission loss at the resonant peak position

12.5.1 Bandwidth Search

After enabling the automatic bandwidth measurement function, the instrument automatically generates 4 sets of markers on the trace and calculates the bandwidth, center frequency, Q value, and insertion loss respectively; unchecking disables this measurement.

Bandwidth Marker Type

- Cursor: Searches bandwidth based on the current active cursor reference point;
- Peak: Automatically searches for the global maximum resonance peak of the trace and uses the peak as the basis for calculation.

Bandwidth Filter Passband Value

The bandwidth determination attenuation threshold; default -3dB, can also be customized. It indicates attenuating down from the passband peak level by this dB value; the two points intersecting the curve are the high and low cutoff frequencies.

12.5.2 Band-Reject Search

Check to enable notch search, which automatically calculates the stopband bandwidth, notch center

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

frequency, stopband Q value, and notch loss; unchecking disables it. It is specifically for band-reject devices with deep attenuation in the middle and pass-through on both sides.

Band-Reject Marker Type

- Cursor: Uses the manually marked position as the valley-point reference;
- Peak: Automatically locates the minimum-attenuation valley point (notch bottom) of the trace as the calculation reference.

Band-Reject Filter Level

The stopband determination threshold; default -3dB. It represents raising from the notch bottom by this dB level; the left and right intersection points with the curve serve as the stopband high and low cutoff points.

Track

- On: Each time the instrument scans and refreshes, it automatically re-executes the bandwidth/band-reject search, and the parameters update in real time following the trace.
- Off: Calculated only once when the search is triggered; subsequent scans do not automatically update the result.

13 Calibration

In the vector network analysis, the test system (including the DUT, cable, connector, fixture, etc.) inevitably may bring about system errors, resulting in the fact that the raw measurement value of the DUT cannot directly represent its true characteristics. Therefore, the VNA is equipped with a series of calibration features such as calibration compensation, fixture simulation, and port extension, to remove the identified errors from the measurement results accurately.

VNA provides a series of calibration solutions to address different application scenarios.

- The calibration compensation is mainly used to remove the inherent basic errors in the test system, which is the basis for achieving high-precision measurement.
- The fixture de-embedding function is intended to remove the effects of the test fixture on the measurement effectively and restore the true performance of the DUT.
- The port extension feature is applicable in compensating the amplitude offset and phase deviation caused by the length of the test cable to ensure the accuracy of the measurement.

The above features are designed for different test demands, but the core principle is to correct errors through S-parameter matrix and error-compensation matrix operations, mathematically compensating the system errors in the transmission path of the measurement signal fundamentally.

Note that when multiple correction functions are enabled simultaneously, the execution sequence of the specified function will directly affect the validity of the final measurement results. Ensure that the sequence of the physical connection of the DUT, the expected test network model setup and function execution are consistent. Only in this way can the measured data truly reflect the characteristics of the

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

DUT.

13.1 Main

The following table lists the S-parameter calibration type supported by VNA.

Table 13.1 S-parameter Calibration Type Supported by VNA

Number of Ports	Calibration Type	Calibration Item to be Performed
1 1-Port (reflection parameters only)	Response Open	Open response calibration
	Response Short	Short response calibration
	OSL	Open, Short, and Load response calibration
2	Response 1 to 2	Port 1→2 Through response calibration
	Response 2 to 1	Port 2→1 Through response calibration
	Enh Response 1 to 2	Enhanced response calibration • Port 1: Open, Short, and Load • Port 1→2: Through
	Enh Response 2 to 1	Enhanced response calibration • Port 2: Open, Short, and Load • Port 2→1: Through
	SOLT	All response calibration • Port 1: Open, Short, and Load • Port 2: Open, Short, and Load • Port 1⇄2: Through
	SOLR (Unknown-Thru Calibration)	All response calibration • Port 1: Open, Short, and Load • Port 2: Open, Short, and Load • Port 1⇄2: Through
	TRL	All response calibration • Port 1: Open, Short, and Load • Port 2: Open, Short, and Load • Port 1⇄2: Through • Port 1⇄2: Line delay line

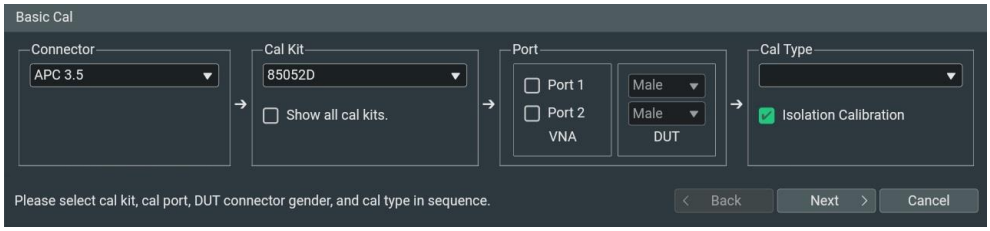
13.1.1 Basic Calibration

Basic Calibration can only calibrate the currently selected channel. First, select the channel to be calibrated in the measurement interface (directly click a trace on that channel, or select via the channel menu). For example, as shown in the figure below, calibrate channel 3;

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Click **Basic Cal** to enter the basic calibration interface.



- **Connector**

  Selects the desired connector type from the drop-down list to select APC 3.5, APC 2.4, APC 7, 7-16, Type F (75), Type N (75), or Type N (50).

- **Cal Kit**

  Selects the desired calibration kit model from the drop-down list based on the currently selected connector.

  If you check the checkbox of "Show all cal kits", then all of the calibration kits are available to choose without being limited to the connector type.

- **Port**

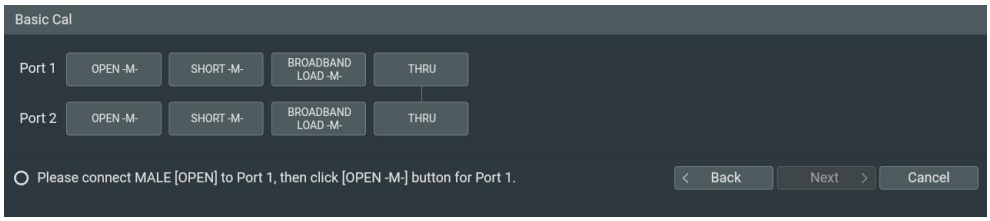
  Check the checkbox of the specified port number to be calibrated, then select the port type (Male or Female) of the DUT that connects the specified port. You can select one or two ports (Port 1 and Port 2) to calibrate.

- **Cal Type**

  Selects the desired calibration type from the drop-down list based on the currently selected port. For details, refer to Table 13.1.

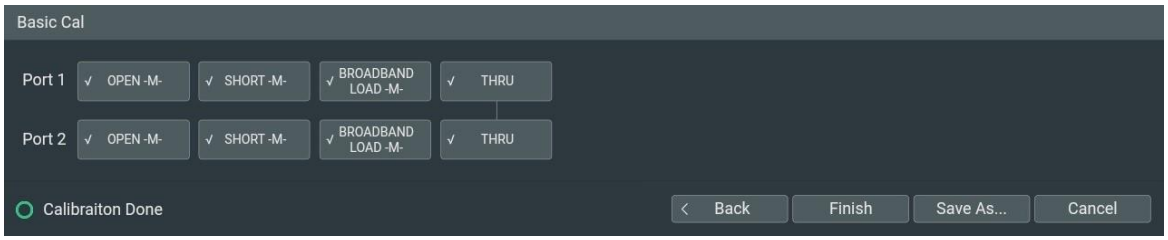
After completing the above configuration, click **Next** to enter the following interface. If you haven't set the parameters well, you can click **Back** to go to the previous step to configure the parameters.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



In the menu, you can view the currently selected port to be calibrated and the calibration items to be performed on the specified port. The calibration items include OPEN-M- (Open calibration measurement), SHORT-M- (Short calibration measurement), BROADBAND LOAD-M- (Load calibration measurement), and THRU (Through calibration measurement). Follow the procedures below to perform the basic calibration.

1. Connect the calibration kits to the test ports of VNA properly according to the pop-up messages.
2. Click the first calibration button, then the current calibration trace window is displayed below the window. When "√" (a tick icon) is displayed on the button, it indicates that this calibration item is completed.
3. Repeat Step1 and Step 2 until all the selected calibration items have been calibrated successfully.



When measurements for all standards are completed, the **Finish** and **Save As** buttons become enabled.

Click **Save As** to enter the file management interface (File Management), whereX412397b9483fd592d4390ed9d30b89073014775 calibration data can be saved to disk for direct recall in future measurements. The available file types for saving are listed below:
*.csa file: Contains all channel settings and corresponding calibration data. After loading a *.csa file, all channel configurations and calibration status can be restored directly.
*.cal file: Saves calibration data only for the currently -calibrated channel. Before loading a *.cal file, you must first configure the channel stimulus settings (identical to those used during the previous calibration). The *.cal file will take effect only after this configuration.

After saving by clicking either **Finish** and **Save As**, the calibration procedure menu closes and returns to the measurement interface. Calibration is automatically applied to the active channel at this point. Click **Finish** to abort the current calibration.

13.1.2 Isolation Calibration

Isolation calibration is not performed by default. If you need to additionally perform this operation, please follow these steps:

1. Check the "Isolation Calibration" option, and the calibration file will include this data;

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



2. Complete the calibration procedures for other conventional parameters in sequence;
3. Connect the load to the test port;
4. Manually perform isolation calibration.
5. After calibration is complete, you can save the calibration data and end this calibration.

13.1.3 Electronic Calibration

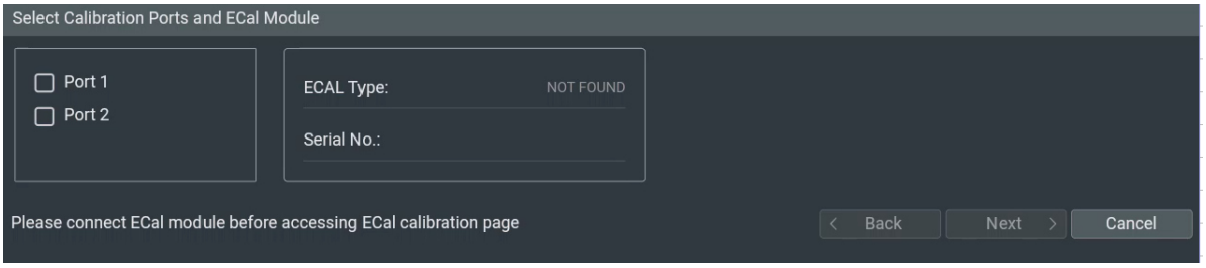
ECal is a solid-state calibration solution. Every ECal module contains electronic standards that are automatically switched into position during a measurement calibration. These electronic standards have been measured at the factory and the data are stored within the memory of the ECal module. The VNA uses the stored data, along with the measured data, to calculate the error terms for a measurement calibration.

For the ECal modules supported by VNA and its operation and connection methods, refer to *ECal Module Manual*.

Connect the ECal module port to the port of the analyzer. The ECal module is controlled and powered through a USB connection. Wear a grounded wrist strap when making ECal calibration. Do not disconnect USB devices and test port during ECal calibration. After connecting the ECal module properly, wait for completing warm-up until the indicator turns green, then start to calibrate.

The procedures for calibration operation are as follows:

1. Click **ECal** to enter the "Select Calibration Ports and ECal Module" interface. Set the following configuration.



- **Select the calibration type**

  Supports single port calibration or multiple ports (e.g. 2-Port) calibration. You can select the desired port based on your DUT.

- **Port ECal configuration**

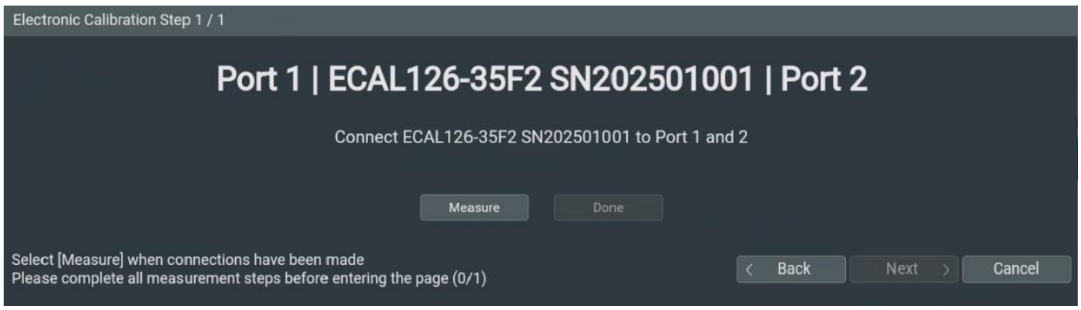
  Selects the desired calibration port based on the calibration type. For 2-port calibration, you must select two different ports.

- **ECal properties**

  After connecting properly, VNA can automatically recognize the ECal module and obtain the module information.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

2. Click **Next** to enter the following interface.



Complete the calibration operation according to the calibration wizard.

3. After completing the measurement, the following interface is displayed. Click **Save As** to save the calibration configuration file suffixed with "*.cal" to the specified path. Click **Finish** to complete the calibration.



13.1.4 Cal All

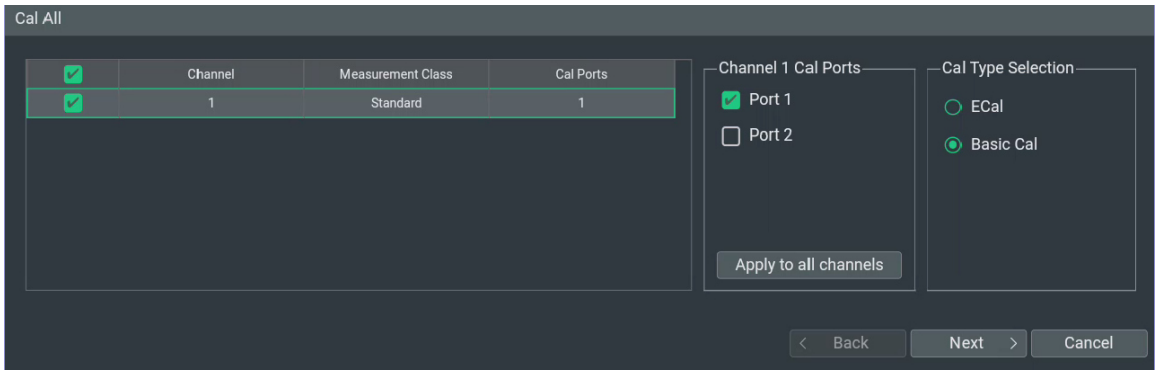
The Calibrate All Channels (Cal All) function is used to uniformly perform calibration on all relevant measurement channels of the Vector Network Analyzer (VNA). In multi-channel measurements, each channel may have different frequency ranges, power levels, bandwidths, scan points, and measurement parameters; calibrating each channel separately is cumbersome and affects work efficiency.

When using the Cal All function, the instrument can centrally complete the calibration procedure for all channels based on each channel's current configuration and apply the corresponding calibration results to each measurement channel. This function effectively simplifies multi-channel calibration operations, reduces repeated settings and manual intervention, and ensures that all channels are in a valid calibrated state, thereby improving the consistency, accuracy, and test efficiency of multi-channel measurements.

The calibration operation steps are as follows:

1. Click **Calibrate All Channels** function key to enter the "Calibrate All Channels" menu.

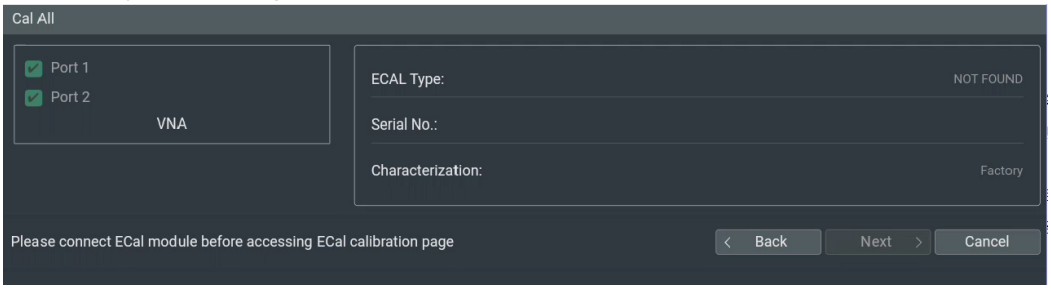
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



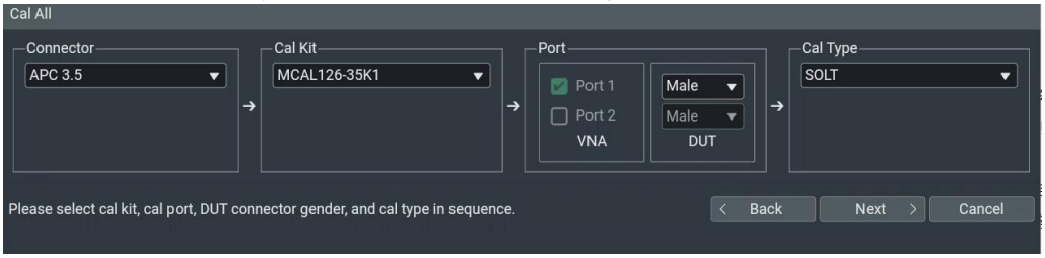
- **Select Calibration Channel**
  Select the channel to be calibrated.
- **Channel Calibration Ports**
  Select the ports of the selected channel to be calibrated; by default, the ports related to the S-parameters configured by the user are selected, and can be increased or decreased as needed. The "Apply to All Channels" button can apply the calibration ports of the current channel to all channels.
- **Select Calibration Type**
  Supports electronic calibration and basic calibration.

2. Click Next to set the calibration kit and calibration ports.
   - If electronic calibration type is selected, the following menu appears.

   Select the ports to be calibrated, connect the electronic calibration module correctly, and the VNA can automatically detect and obtain the information of the electronic calibration module and display it on the right.



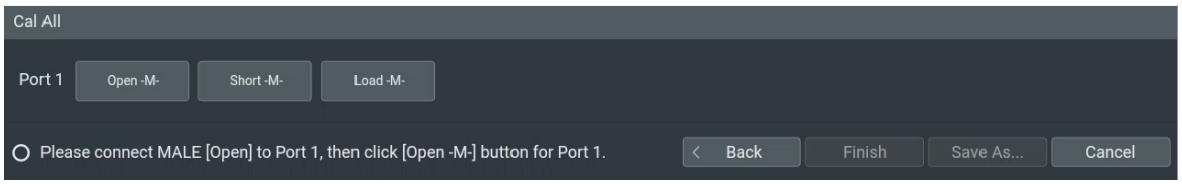
   - If basic calibration type is selected, the following menu appears.



   Connector: Select the connector type of the calibration kit through the drop-down menu, including: APC3.5, APC2.4, APC7, 7-16, Type F (75), Type N (75), Type N (50).
   Calibration Kit: Select the available calibration kit model for the currently selected "Connector" type through the drop-down menu.
   Port: Displays all ports to be calibrated, used to confirm the calibration ports.
   Calibration Type: **Select** the optional calibration type under the current configuration through the drop-down menu; see Table 13.1.

3. Click **Next**, to enter the following menu.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Complete the subsequent calibration operations according to the prompts in the menu.

3. After the measurement is complete, the interface is as shown below. Click **Save As** button, to save the calibration configuration file (*.cal) to a specified path; click **Finish** button, to complete this calibration.



13.1.5 Correction/Correction Properties

By default, the correction function is disabled. After completing the calibration or loading the calibration files successfully, correction is performed automatically, and its state turns out to be ON. If the instrument is not calibrated, correction cannot be enabled.

If the instrument has not been corrected, "No Correction" displays in System Status Bar below the window. After being corrected, it displays the correction state. The following table lists the detailed information of correction state.

Number of Ports	Calibration Type	Correction State
1	OSL	C 1-Port
	Response Open Response Short	C Resp
2	Response 1 to 2 Response 2 to 1	
	Enh Response 1 to 2 Enh Response 2 to 1	C Enh Resp

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

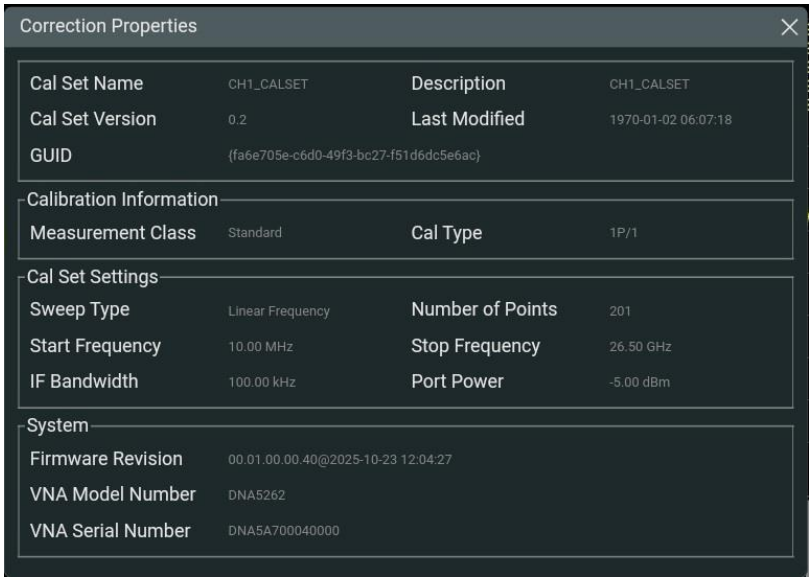
Number of Ports	Calibration Type	Correction State
	SOLT	C 2-Port

The following table lists the correction level and accuracy for different correction state.

Correction State	Description	Accuracy
C N-Port	N indicates the number of fully calibrated ports.	Highest
C Enh Resp	Indicates the enhanced response calibration. It is an aggregate of a 1-port calibration and a transmission response calibration.	Second Highest
C Resp	Open or Short circuit response calibration It indicates correction for the frequency response of the measurement. It does not correct for impedance mismatches.	Second Lowest
No Correction	Indicates that the correction is disabled.	Lowest
C*	Interpolated correction. For details, refer to Interpolation.	Uncertain
CΔ	Changed correction. For details, refer to Interpolation.	Uncertain

Correction Properties

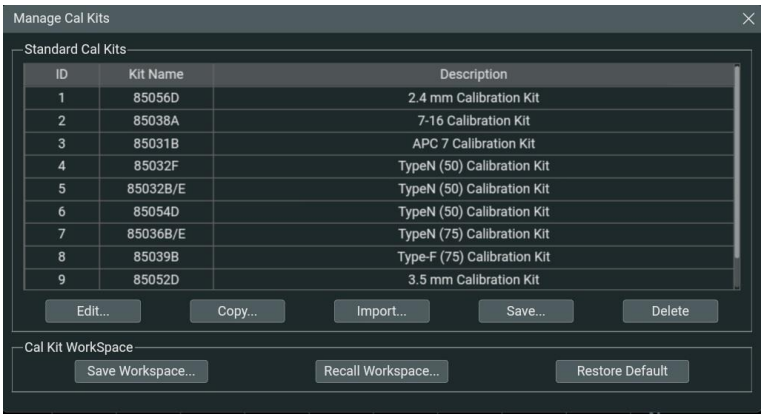
After completing the correction, click **Correction Properties** to view the correction related information. If the instrument has not been calibrated, this menu is grayed out and disabled.



13.1.6 Calibration Kits

Click **Cal Kit** to enter the "Manage Cal Kits" interface.

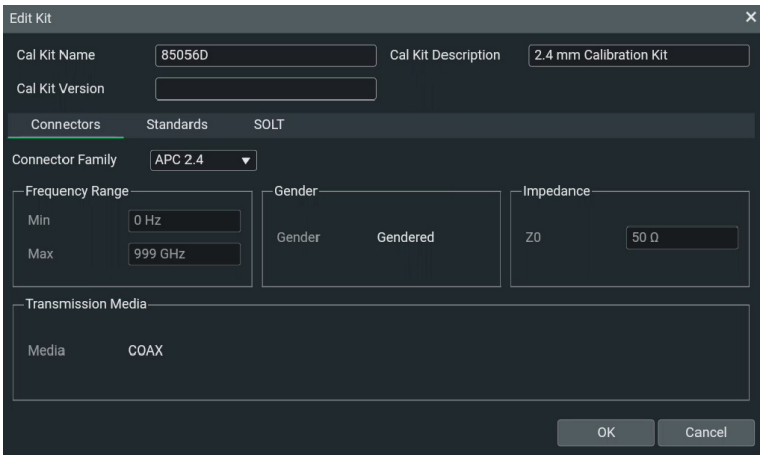
Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



There are multiple built-in calibration kits in the VNA.

13.1.6.1 Edit the Calibration Kits

Modifies the parameters for each calibration kit. Select one calibration kit, then click **Edit...** to enter the calibration kits editing interface.



Modify the parameters according to your needs. After completing editing, click **OK** to save the editing. Then exit the parameter setting interface automatically.

Basic Operations

- **Copy:** Select one calibration kit, then click **Copy...**, then the calibration kits copying interface is displayed. Modify the parameters according to your needs. After completing editing, click **OK** to copy the specified calibration kit to the calibration kits list. Then exit the calibration kits copying interface automatically. The newly copied calibration kit is added to the calibration kits list, with the default filename "original calibration kit name_N". Here, N indicates the number. By default, for the first copy version of the specified calibration kit, N is 1, it increases with an increment of 1.

- **Import:** Click **Import...**, then the file management interface (File Management) is displayed. Select the file suffixed with "*.xkt" from the specified path. Click **Confirm** to import the specified file from the specified path. You can import the user-defined calibration kit to the current calibration kit list. It is displayed at the last line of the current calibration kit list. Then you can use the user-defined

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

calibration kit to calibrate the VNA.

- **Save:** Select one calibration kit, then click **Save...** to enter the file management interface. Save the current calibration kit and its detailed parameter settings to the specified path, with the filename suffixed with "*.xkt".

- **Delete:** Select one calibration kit, then click **Delete**, then a prompt message prompting you whether to delete the specified calibration kit is displayed. Click **Confirm** to delete the specified calibration kit from the current calibration kits list.

Calibration Kit Workspace

In the "Manage Cal Kits" interface, the displayed "Standard Cal Kits" list is defined as one workspace.

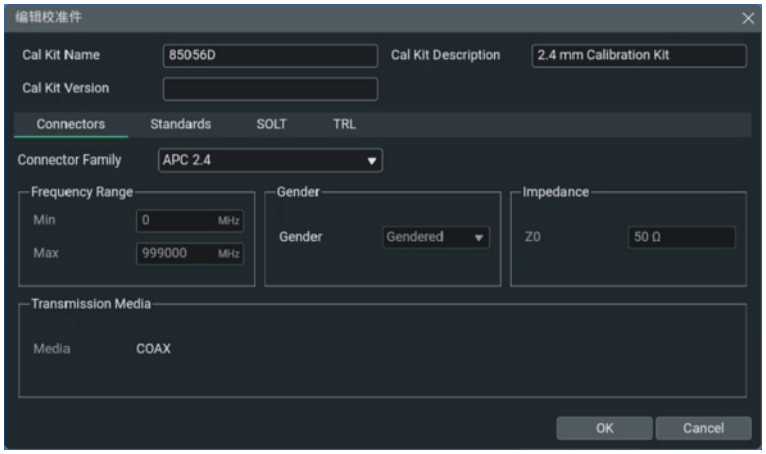
- **Save Workspace:** Click **Save Workspace...** to enter the file management interface. All of the calibration kits and the detailed parameters information of each calibration kit in the current workspace are saved to the specified path, with the filename suffixed with "*.xkw".

- **Recall Workspace:** Click **Recall Workspace...** to enter the file management interface. Select the specified file suffixed with "*.xkw" from the specified path. Click **Confirm** to recall the user-defined calibration kit list in replace of the current workspace. Then you can use the specified calibration kit in the user-defined workspace to calibrate the VNA.

- **Restore Default:** Click this button to restore the calibration kits list and detailed parameters of each calibration kit to the default settings.

13.1.6.2 Edit Calibration Kit

Select a calibration kit in the standard calibration kit area, and click **Edit** button to enter the editing menu of this calibration kit, as shown in the figure below.



13.1.6.2.1 Select Connector Type

Click the drop-down box to the right of Connector Family; the available connector types include: APC 3.5, APC 2.4, APC 7, Type N (50), Type N (75), Type F (50), Type F (75), Type B, 7-16, 2.92 mm, 1.85 mm, 1.0 mm.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

13.1.6.2.2 Configure Standards

Add Standard

Under the Standards sub-page, click **Add** button to add the four types of standards: OPEN, SHORT, LOAD, and THRU.

Delete Standard

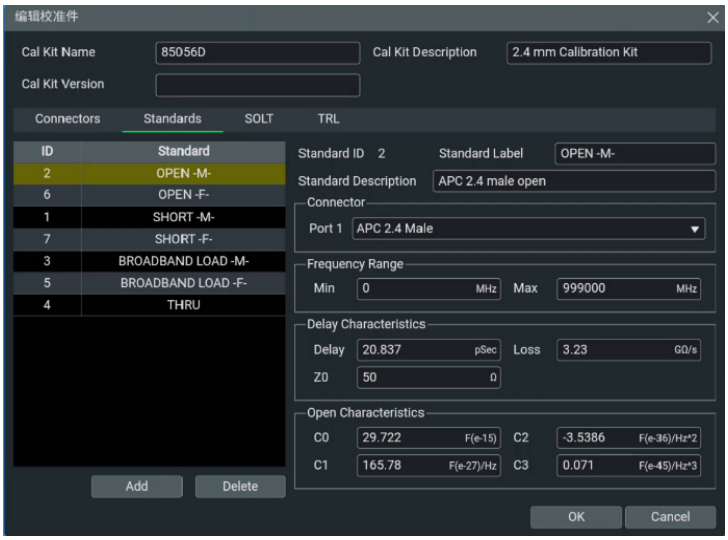
After selecting the calibration standard to be deleted in the Standards sub-page, click **Delete** button, and in the pop-up prompt box click **Confirm**, to delete the calibration standard.

Edit Standard Basic Information

- Standard ID: The serial number of the standard (unique sequence number within the kit, cannot be modified);
- Standard Label: The display name of the standard (calibration pop-up prompt text);
- Description: Remarks for the standard;
- Connector: Defines the connector specification and polarity of the standard (Male / Female / None);
- Frequency Range: The frequency range; Min is the lower limit frequency at which the standard can participate in calibration, and Max is the upper limit frequency, in MHz;
- Delay Characteristics: Delay characteristic parameters; Delay is the one-way transmission delay, in s; Loss is the conductor skin-effect loss, in GΩ; Z0: the nominal characteristic impedance of the transmission line.

Open Standard (OPEN)

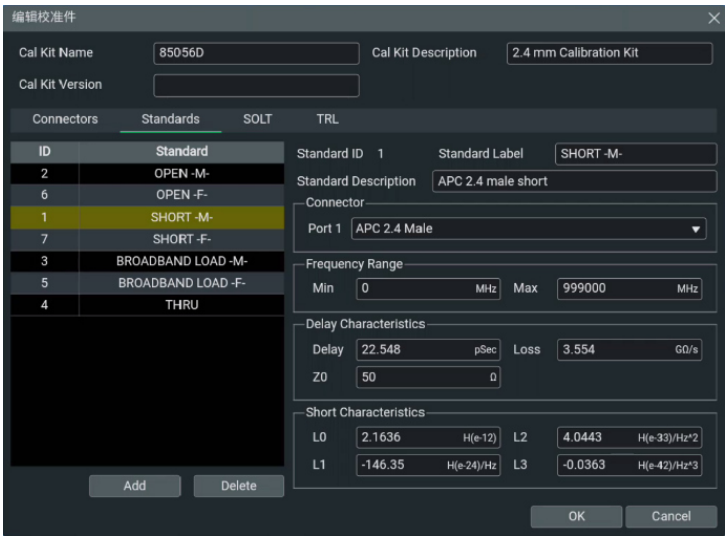
Supports independently configuring the edge stray capacitance coefficients: C0, C1, C2, C3.



Short Standard (SHORT)

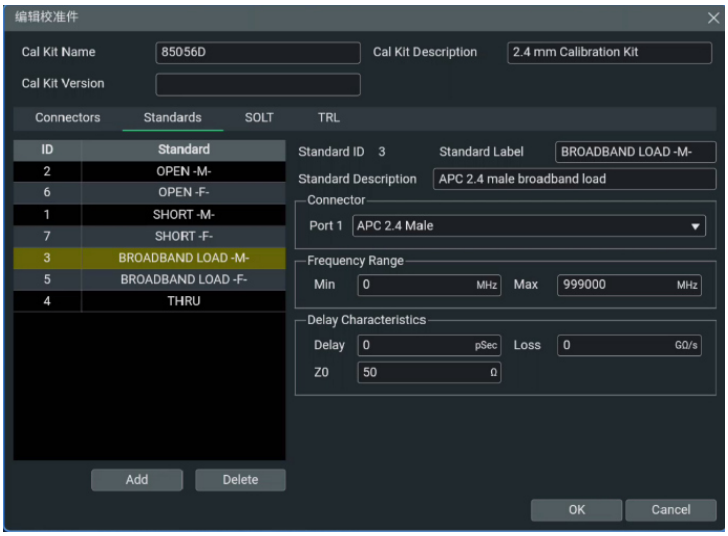
Supports independently configuring the residual inductance coefficients: L0, L1, L2, L3.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Match Load Standard (LOAD)

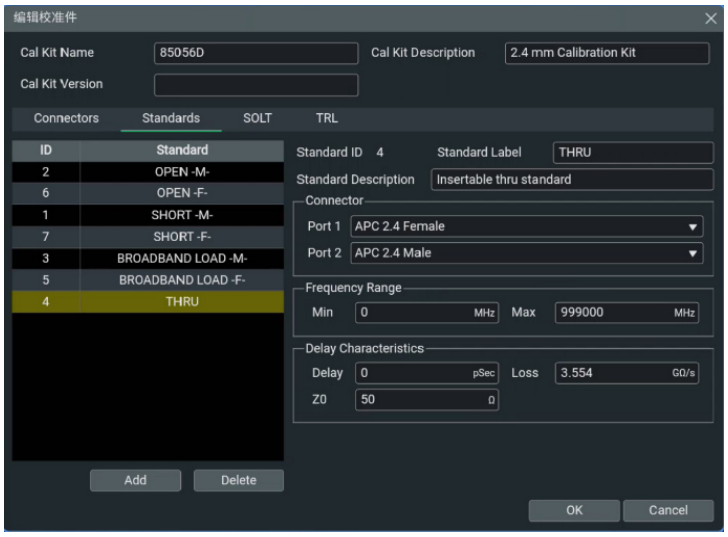
No independent configuration items; the default ideal reflectionless termination model is shown in the figure below.



Thru Standard

The thru calibration standard (THRU) has one set of connectors at each end, with a total of 2 connector configurations, as shown in the figure below.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

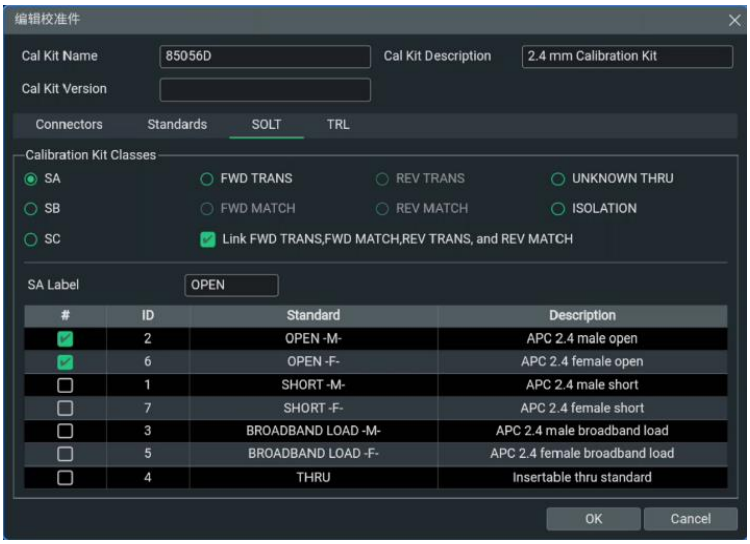


13.1.6.2.3 Configure SOLT Calibration Kit

SOLT calibration kit group definitions:

1. SA: Open standard, but not limited to open standards;
2. SB: Short standard;
3. SC: Match load standard;
4. Link FWD TRANS, FWD MATCH, REV TRANS, and REV MATCH: Forward transmission / match, reverse transmission / match, thru standard;
5. UNKNOWN THRU: Unknown thru, suitable for non-pluggable DUTs.

You can assign the corresponding ID standard to the corresponding group, as shown in the figure below.



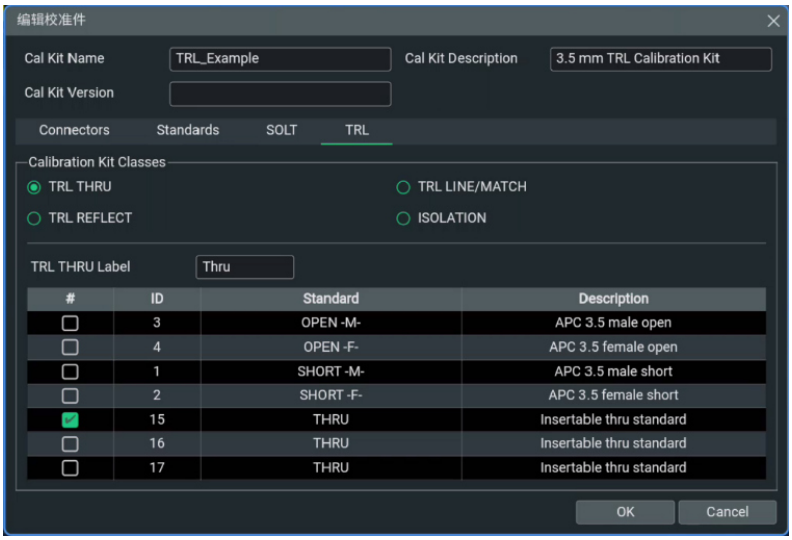
13.1.6.2.4 Configure TRL Calibration Kit

TRL calibration kit group definitions:

1. TRL THRU: TRL thru standard;
2. TRL REFLECT: TRL reflect standard;

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

3. TRL LINE: TRL transmission line standard; the transmission line and the thru together determine the measurement reference impedance after calibration is complete;
4. TRL MATCH: TRL match standard; when a transmission line meeting the required length and loss cannot be fabricated, a match standard can replace the LINE transmission line to complete TRL calibration.
You can assign the corresponding ID standard to the corresponding group, as shown in the figure below.



13.1.6.3 Calibration Kit Workspace Operation

The "Standard Calibration Kit" list displayed in the "Manage Calibration Kits" menu is defined as a workspace.

Save Workspace: Click this button to enter the "File Management" menu, where you can store all "calibration kits" in the current workspace and the detailed parameters of each calibration kit as a "*.xkw" file to a specified path.

Load Workspace: Click this button to enter the "File Management" menu, select the "*.xkw" file at the specified path, and click **Confirm** button to load a custom calibration kit list to replace the current workspace. Subsequently, you can select the required calibration kit from the custom workspace to calibrate the VNA.

Restore Default Settings: Click this button to restore the calibration kit list and the detailed parameters of each calibration kit to the initial state.

13.1.7 To Save or Recall the CalSet File

After completing the calibration, all the calibration data will be saved to the calibration sets. The calibration sets can be applied to the channels that have the same stimulus settings as this calibration set, thus reducing the time for performing the calibration again. The data saved in the calibration sets include ID and description, calibration set properties (stimulus setting, calibration type, port association, etc.), calibration data, error term data, GUID, etc.

After completing one calibration, the data will be saved to the calibration register of the specified channel automatically, overwriting the existing calibration data in the register. You can save it to the

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

specified path as a calibration sets file suffixed with "*.cal".

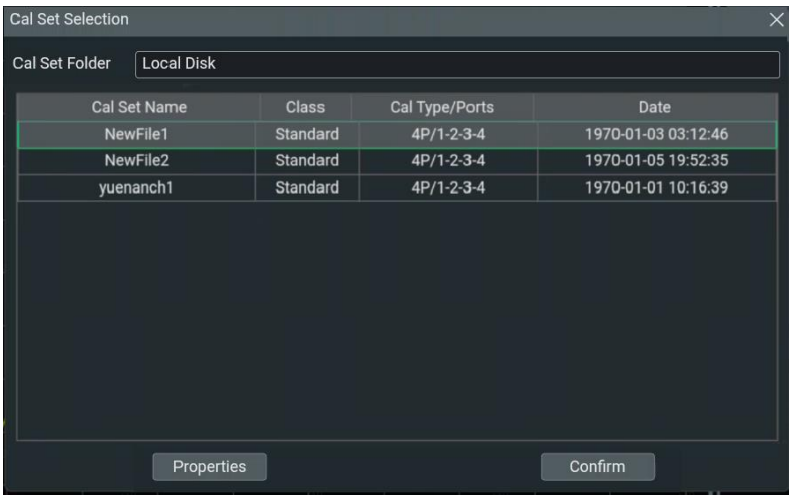
The saving and recalling operation of the calibration sets file can be performed in the file management interface. For details, refer to File.

Save CalSet

Click **Save CalSet**, then the file management interface is displayed. In the interface, set the saving path, filename, the file format "*.cal", then click **Save** to save the file.

Recall CalSet

1. Click **Recall CalSet**, then the file management interface is displayed.



2. Click "Calibration Set Folder" to pop up the "File Management" window, enter the specified path, select the specified *.cal-format calibration set file, and click **Confirm** button, to return to the "Select Calibration Set" window.
3. Select the calibration set, and click Properties button, to pop up the "Calibration Properties" window, displaying the calibration set information.
4. Select the target calibration set and click **Confirm** button, to load it into the currently selected channel.

   If the system detects that the selected calibration set does not match the current channel parameters, it will automatically pop up a prompt asking whether to modify the stimulus parameters to adapt to the calibration set.

13.1.8 Interpolation

Calibration interpolation adjusts calibration error terms to match changes to the stimulus parameters settings (such as frequency and points) that you make after a calibration is performed or a CalSet is applied, keeping the accuracy of the measurement results through the interpolation algorithm, avoiding error calibration failure arising from modifying the settings.

Enable or Disable the Interpolation

By default, interpolation is disabled. When enabled, the interpolation is enabled for the current active measurement, but this does not necessarily mean that the measurement is interpolated. When enabled, if you modify the following stimulus settings, and the system deems that it is necessary to initiate interpolation operation to keep the accuracy of the measurement, then the interpolation will be applied.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

If you modify the stimulus settings while the interpolation is disabled, then interpolation is not applied, but the error correction function (Correction/Correction Properties) is automatically disabled.

Interpolation occurs (if enabled) if you change the following settings in Sweep.

- Start frequency increased
- Stop frequency decreased
- Number of points adjusted

Tip:

Decreasing the start frequency or increasing the stop frequency will force the correction function to be disabled.

Interpolation Calibration Accuracy

When a measurement is interpolated, the accuracy of the measurements cannot be predicted. It may be affected significantly or not at all. Identifying measurement errors in these cases must be determined on a case-by-case basis.

- In general, the magnitude and phase stimulus from the VNA and the response from the DUT need to be smooth and continuous for measurement interpolation to give accurate results.
- Significant measurement inaccuracy occurs when the phase shift response variation between measurement points changes more than 180 degrees.
- In general, the chances of significant inaccuracy increase when interpolating measurements under the following conditions:
    - When the frequency span between measurement points becomes greater
    - When the measurement frequencies are above 10 GHz where the phase changes happen more rapidly
    - When interpolating across frequency band crossings

When the interpolation algorithm encounters an abrupt or large change in the response magnitude or phase, such as can occur at band crossings, large interpolation errors can be included in the displayed data. These errors can be seen as steps or spikes. If this occurs, attempt to use the following methods to avoid it.

- Disable interpolation
- Adjust the sweep measurement parameters such as decreasing the frequency span, decreasing the sweep points, etc.
- Create sweep segments that skip over the band crossings

Display State of Interpolation Calibration

After executing the interpolation correction, the calibration status displayed in System Status Bar below the window changes. The character C following a star "*" appears. For example, "C* 2-Port".

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Interpolation Correction:**

  If you modify the Sweep points, increase the start frequency or decrease the stop frequency, the interpolation correction is enabled, then the correction state displayed in System Status Bar below the window shows "**C***", for example, "C* 2-Port".

- **Configuration Modification**:

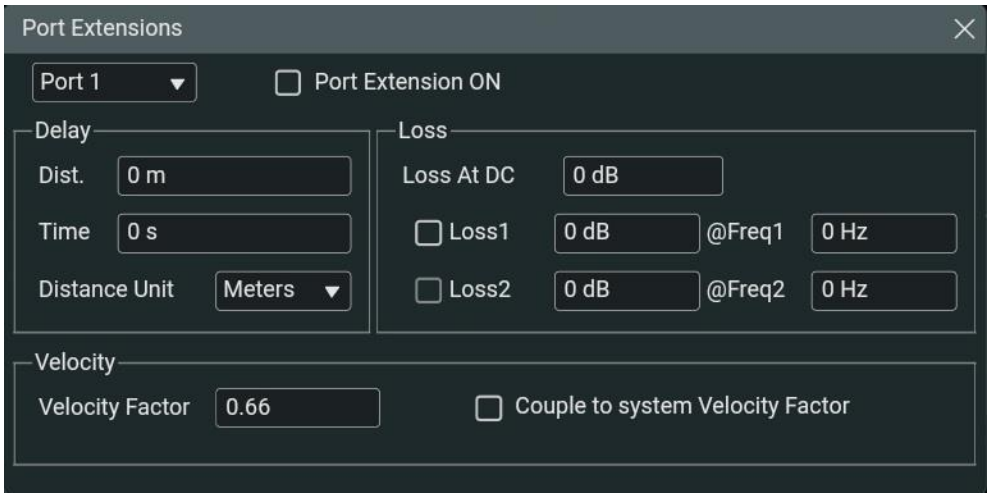
  If one or multiple stimulus parameters such as IF BW, Sweep Time, or Port Power change, the correction status shows "**CΔ**". For example, "CΔ 2-Port".

13.2 Port Extension

Port extensions allow you to electrically move the measurement reference plane after you have performed a calibration. After completing the S-parameter calibration, use the port extension to address the following problems in the test scenarios.

- **Test fixture limitation:** The device is in the test fixture, preventing you from performing a calibration directly on the device. Perform a calibration at a convenient place, then use the port extension function to compensate for the time delay (phase shift) and loss caused by the fixture.
- **Additional transmission line:** If you have performed a calibration, and then you decide to add a length of transmission line in the measurement configuration, then you can use the port extension configuration to extend the length to the specified port of the VNA.

Click **Calibration** > **Port Extension** > **Port Extensions** to enter the Port Extensions interface.



Configure the port extensions parameters

- **Port 1/2:** Selects the measurement port to be used. The settings of the port extension will affect all measurements on the active channel that are associated with the specified port.
- **Enable or disable the port extension:** Check or uncheck the checkbox of **Port Extension ON** to enable or disable the port extension function.
- **Delay**

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Time: indicates the amount of port extension delay in time. Enter a positive or negative value.

Dist.: indicates the amount of port extension delay in physical length. Enter a positive or negative value.

Distance Unit: indicates the available distance units, such as Meters, Feet, and Inches.

- Loss
  - **Loss at DC:** Offsets the entire frequency by this value. It is used to compensate for the loss at DC caused by the transmission line and test fixture. To compensate for loss at DC, enter a positive value which causes the trace to shift in the positive (up) direction.
  - **Loss1@Freq1:** the loss and frequency parameters are used together. After checking the checkbox of "Loss1", input a specific loss value in dB, and specify a frequency value in Hz. Set a loss reference value at the specified frequency point to make the calibration more accurate in the loss at the specified frequency range.
  - **Loss2@Freq2: Similar to the setting of Loss1@Freq1**. Input a specific loss value in dB and specify a frequency value in Hz.

    When "Loss1" and "Loss2" are used, a curved-fit algorithm is used to calculate the loss compensation of other points within the frequency range to accurately calibrate the loss across the entire frequency range.

- **Velocity Factor:**

  For each port, set the velocity factor that applies to the medium of the device that was inserted after the measurement calibration. For details, refer to descriptions about velocity factor in Electrical Delay. By default, the velocity factor is set for only the specified port and only for port extension. After checking the checkbox of **Couple to System Velocity Factor** following **Velocity Factor**, the velocity factor is set for all ports.

  Once the checkbox of **Couple to System Velocity Factor** is checked, modifying the velocity factor value will also change the velocity factor in Electrical Delay, DTF (Option), and Time Domain Setup (TDA Option).

Procedures for performing port extension

1. Select a calibrated S11 measurement trace. Take reflection coefficient of Port 1 as the basic measurement.
2. Set the data format of the measurement trace to "Phase" to better observe and adjust the phase.
3. Connect Open or Short at the calibration reference plane. View and verify the phase readouts across the frequency range. In the ideal state, the phase shall be at or near zero. If the phase deviation is large, check whether there is a problem in the calibration process or hardware connection.
4. Connect the fixture or added transmission line to the measurement system in place of the DUT. In the position where DUT is located, attach an Open or Short.
5. Adjust the delay compensation
   - If the fixture or the "electrical length" of additional transmission line is known, set the "Delay" parameter directly.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- If the fixture or the "physical length" of additional transmission line is known, set the "Distance" parameter directly.
- If the extension length is unknown, increase the value of "Delay" or "Distance" gradually to observe the phase response within the frequency span shown on the screen. Adjust the value continuously until the phase response within the specified frequency span becomes smooth, that is, ensure that the phase variation is within the acceptable range. Then complete the port extension operation.

After performing the above operations, observe whether the phase, reflection coefficient, and other parameters at different frequency points meet the expected values. Compare the results with the results that have not performed port extension to confirm whether the setting is valid. If the measurement results are not ideal, repeat the above steps and readjust the relevant parameters.

13.3 Auto Port Extensions

APE (Auto Port Extension) is an automated reference plane calibration function. By simply connecting an OPEN or SHORT standard, the system can automatically calculate and apply delay and loss compensation parameters, accurately moving the measurement reference plane to the standard interface. This function effectively eliminates errors introduced by additional transmission lines and test fixtures, significantly improving test efficiency and accuracy and avoiding complex manual debugging.

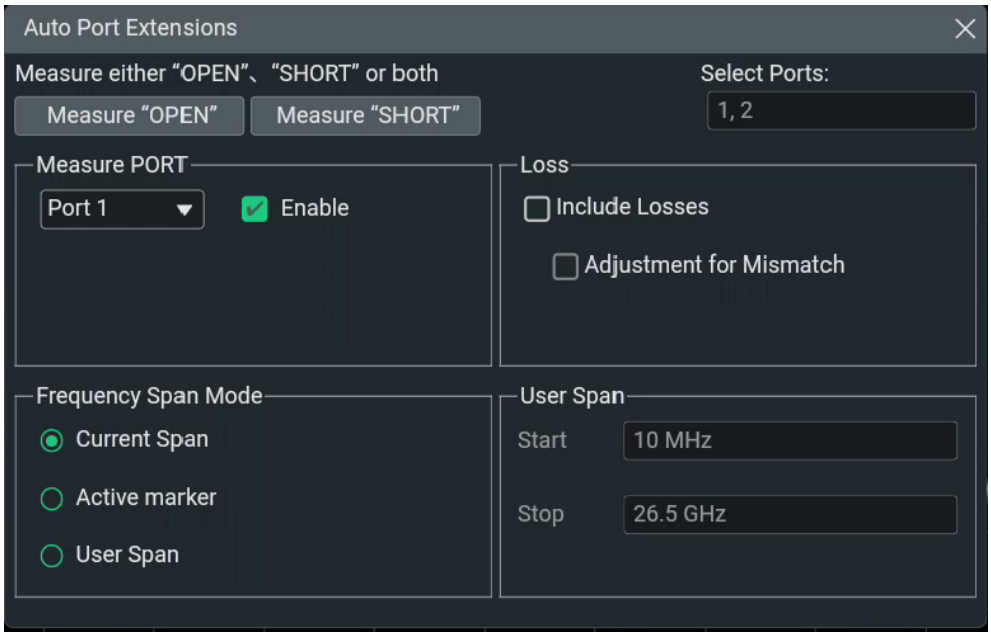
Prerequisites for Using APE

Before executing APE, make sure the system meets the following requirements; otherwise the feature will not be triggered or respond:

- Calibration requirements: The instrument has completed the calibration for the S-parameters of the channel;
- Sweep mode: The current sweep type should be set to a type other than **Power**;
- Feature status: Frequency offset is disabled;
- Media settings: Measurement medium is not configured as Waveguide;
- Trace requirements: No math operation is enabled on the active trace.

Click **Calibration** > **Port Extension** > **Auto Port Extensions** to enter the Auto Port Extensions interface.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



Configure the Auto Port Extensions Parameters

All the settings are configured for the channel, and take effect for all the traces of the current channel.

- **Measure Port**

  Specifies the test port to perform the auto port extension and controls the on/off status of the port, which is updated synchronously as what "Select Ports" displays.

  By default, APE feature is enabled for all the test ports automatically. The Auto Port Extensions settings apply to all measurement items associated with the port on the active channel.

- **Loss**

  - **Include Losses**

    By default, it is enabled and the loss is automatically set to 0; the "Adjustment for Mismatch" is also grayed out and disabled.

    When it is enabled, the system automatically calculates the loss parameters and writes the port extension configuration to compensate for the loss of transmission line/fixtures (including DC loss and frequency-related loss).

  - **Adjustment for Mismatch**

    By default, it is enabled. It is only available to set when "Include Losses" is enabled. When you check the checkbox of "Adjustment for Mismatch", the measurement trace will be adjusted, forcing the amplitudes of all data points ≤ 0 dB, avoiding instability of S-parameter values in the simulation.

- **Frequency Span Mode**

  Selects the frequency span range used in the calculation of APE parameters. Three modes are available to choose:

  - Current Span: Calculated using the current frequency span setting for the channel.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- Active marker: Calculated based on active marker and adjacent high frequency point data. If no marker is currently available, the system automatically creates a marker at the center of the frequency span.
- User Span: The start/stop frequency needs to be set manually and the range shall not exceed the frequency span range of the current channel.

- **User Span**

  It is only available to set when you select "User Span" under the "Frequency Span Mode". It is used to set a user-defined frequency span.

  - Start: indicates start frequency of the user span. Its setting range is as follows: Current Start Frequency of the Channel ≤ Start < Stop Frequency.
  - Stop: indicates stop frequency of the user span. Its setting range is as follows: Start Frequency < Stop ≤ Current Stop Frequency of the Channel.

Operation Procedures for APE

1. **Preparation**

   Complete the S-parameter calibration for the channel to verify that the system meets the requirements for APE.
2. **Configure Parameters**

   Set the APE-related parameters in the APE interface, such as target test port, loss, and frequency span.
3. **Connect the Standard Components**

   Connect Open or Short to the enabled port securely.
4. **Trigger the Measurement**

   Click "Measure OPEN" or "Measure SHORT". The system verifies the pre-conditions before starting the measurement. If the pre-conditions are met, the measurement will be initiated.
5. **System Calculation and Application**

   The system automatically checks the history measurement records and current configuration to select the desired calculation method.

   - With the same parameter configuration, if one of the measurements of the standard components has been performed, the second measurement automatically triggers the OPEN+SHORT joint calculation, making the measurement accuracy higher.
   - If the parameter configurations are different, the system calculates the measurement of the single standard component, and the results will overwrite the history measurement data of different parameters.

   After the system calculation is completed, the system automatically activates the port extension function, and the calculated compensation parameters such as delay, loss etc. are applied to the measurement. The trace is updated in a real-time manner. The compensated measurement results will be presented synchronously.
6. **Verify Compensation Effects**

   Set the data format to **Phase**. If the phase curve in the full range is flat, it indicates that the port

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

extension compensation is completed. The reference plane has been moved to the position where the standard component is connected.

13.4 Fixture

The fixture is used to simulate the electrical characteristics of the fixture with the software-based math model. With the S-parameter (scattering parameters) of the test fixture or the calculated S-parameter based on the physical structure and material parameters of the fixture, you can build the math models. When measuring the DUT, the measured data obtained by the VNA contain the comprehensive characteristics of the DUT and the fixture. The fixture feature applies the established fixture math model to the measured data and removes the effects of the fixture from the measurement results through math operation to obtain the true characteristics of the DUT.

The fixture is commonly used in the testing of semiconductor and other miniature or special encapsulated components, high-precision batch measurements of RF components, and high-frequency signal path compensation in high-speed interconnect testing.

Fixture Functions

- **Eliminate the effects of the fixture:** Through math models, the fixture eliminates the errors from the measurement results, such as loss, reflection, and phase deviation caused by the fixture, to obtain more accurate DUT characteristics data.
- **Circuit embedding and de-embedding:** Adds (embed) or removes (de-embed) the specified circuit to, or from the measurement.
- **Port Z conversion (port reference impedance conversion):** Enables the measurement to be carried out under different impedance standards to meet different test demands.

Test Notice

When using the fixture, avoid the error caused by the fixture to ensure the accuracy of the calibration results.

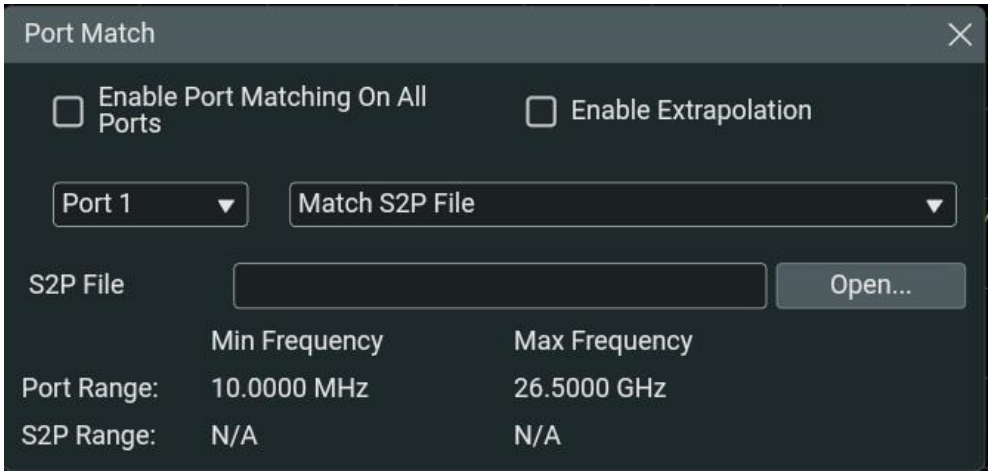
1. **Keep the fixture model parameters accurate:** Deviations of the input S-parameters of the fixture may lead to the distortion of the compensated results.
2. **Keep the consistency between the fixture and the models:** In the batch measurement, if the fixture is replaced, you need to re-measure or adjust the fixture model parameters, rather than using the original model to perform the measurement.
3. **Keep the frequency band matched:** Keep the effective frequency band of the fixture model to cover the working frequency band of the DUT.
4. **Basic calibration not negligible:** The fixture simulator is a compensation tool superimposed on the basic calibration, but it cannot replace the basic calibration (e.g. SOLT calibration) of the analyzer. Thus, basic calibration is still necessary.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

13.4.1 Port Match

The port match feature is used to embed the specific circuit model into the measurement results, simulating the impedance characteristics of the fixture at the VNA port, thus reflecting the performance of the DUT in the actual working environment accurately.

Click **Port Match** to enter the following interface.



1. Check the checkbox of **Enable Port Matching on All Ports** to enable the port matching function.
2. Click the drop-down button to select the desired VNA port to be matched.
3. Click to select "None" or "Match S2P File" from the drop-down list. When "Match S2P File" is selected, you can select the path where to load the "*.S2P" file.

   Click **Open** to enter the file management interface (File Management), select the desired "*.S2P" file to be loaded, then click **Confirm** to complete loading.

   The "*.S2P" file is the S-parameter file of 2-port network, which is used to simulate the representation of component model of the fixture in the actual test circuit.
4. When the frequency range of the loaded S2P file does not match the current measurement range, you can check the checkbox of **Enable Extrapolation**, the system will extend the values for the first and last data points in either direction through algorithm to make it cover the frequency range of the current measurement, ensuring the port match function works normally.

13.4.2 Port Z

The default reference impedance of VNA is 50 Ω, and the characteristic impedance of the actual DUT may have a variation from the default value, e.g. Z_0 = 25Ω/75Ω/100Ω. The port impedance conversion function converts the default reference impedance into the actual characteristic impedance of the DUT through the built-in math modeling algorithm; or converts the performance data under the non-default impedance to the equivalent at the system default impedance, ensuring that the actual electrical performance of the DUT can be accurately obtained under different impedance standards.

Process of port impedance conversion:

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

1. Acquire raw data: Take the system default impedance 50 Ω as the base reference to measure the DUT's raw S-parameters.
2. Configure the target impedance: Set the actual characteristic impedance of the DUT, that is, target impedance.

   The target impedance is configured based on the DUT characteristics settings. The key parameters are as follows:

Parameter	Definition	Application Scenarios
Real	Resistance component	DUT is the pure inductive component, and only real part is required to be configured, and the imaginary part is by default 0.
Imaginary	Reactance component	The DUT contains parasitic capacitance or inductance, and it is necessary to configure the imaginary part. Otherwise, the measurement deviation will increase significantly.

3. Data conversion operation: Convert the raw data into the equivalent S-parameter under the target impedance to output the accurate measurement results.

The Port Z configuration interface is displayed in the following figure.



1. Check the checkbox of "Enable Port Z Conversion (all ports)" to enable the port Z. You should also enable de-embedding. The converted impedance value is only applicable to all the measurement trace of the current active channel.
2. Click the drop-down button to select the desired VNA port (Port 1/Port 2) to be applied to impedance conversion.
3. Set the target impedance parameters: real part (R) and imaginary part (jX).

13.4.3 2-port DeEmbed

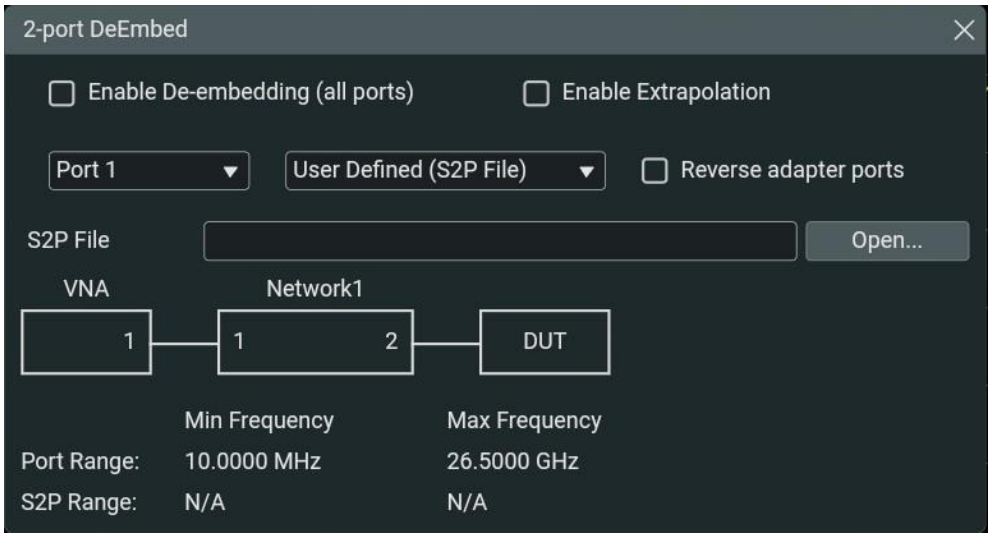
2-port DeEmbed feature is used to optimize the VNA measurement results. After calibration, if a 2-port fixture (e.g. Adapter, attenuator, a long length of cables) is added in the measurement system, they will affect the measurement results if they are connected between the calibration reference plane and the DUT, unable to measure the DUT accurately. The 2-port DeEmbed feature can remove the effects of the 2-port fixture from the measurement results and restore the true performance of the DUT.

The de-embed operation recalls a "*.S2P" file in Touchstone format. This file contains the electrical characteristics of a 2-port fixture or component, such as impedance and phase. By analyzing and

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

calculating the data in this file, in combination with the measured data, the de-embedding feature can eliminate the effects of the fixture on the measurement results.

Click **2-port DeEmbed** to enter the following interface.



1. Check the checkbox of **Enable De-embedding (all ports)** to enable the port de-embedding function.
2. Click the drop-down button to select the desired VNA port (Port 1/Port 2) to be de-embedded.
3. Click to select "None" or "User Defined (S2P File)" from the drop-down list. When "User Defined (S2P File)" is selected, you can select the path where to load the "*.S2P" file.

   Click **Open** to enter the file management interface (File Management), select the desired "*.S2P" file to be loaded, then click **Confirm** to complete loading.

   The "*.S2P" file is the S-parameter file of 2-port network, which is used to simulate the representation of component model of the fixture in the actual test circuit.
4. When you use the 2-port fixture to connect the VNA and the DUT, the connection method of the fixture may affect the accuracy of the measurement results. "Reverse adapter ports" feature is used to adjust the port configuration between the fixture, VNA, and DUT to adapt to the actual connection conditions.

   Generally, Port 1 connects one port of the VNA, and Port 2 connects the DUT. However, in some special test scenarios or in the fixture design, you need to connect Port 2 to the VNA and connect Port 1 to the DUT, at this time, you can check the checkbox of "Reverse adapter ports" to adjust the connection method.

   The reverse of the adapter ports actually changes the signal transmission path and measurement reference direction, enabling the VNA to analyze and process measurement data based on the new port connection configuration, thus removing the effects of the fixture on the measurement results accurately.
5. When the frequency range of the loaded S2P file does not match the current measurement range, you can check the checkbox of **Enable Extrapolation**, the system will extend the values for the first and last data points in either direction through algorithm to make it cover the frequency range of the current measurement, ensuring the port match function works normally.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

14 Power

Sets the power parameters of the stimulus source. In this menu, you can set the output power of the test port within the frequency range to ensure to use the proper stimulus power to make the measurement within the specified frequency range.

Enable or Disable Power

Enables or disables whether the test port outputs the RF stimulus signal. By default, it is enabled. The stimulus source settings are global to all the channels.

Port Power

When you set the sweep type to "Linear Frequency" or "Log Frequency" (click **Sweep** > **Main** > **Type**), you can set the power at the test port.

Start Power/Stop Power

When you set the sweep type to "Power" (click **Sweep** > **Main** > **Type**), you can set the start power and stop power of the sweep signal.

The start power can be greater than the stop power, which indicates that power sweep goes from large values to small values, and no direction is required to be set.

15 Avg BW

The averaging and bandwidth function is used to reduce the random noise and improve measurement accuracy. Modifying the averaging parameters and IF bandwidth will affect the measurement time and measurement accuracy for the current channel.

15.1 Main

The vector network analyzer reduces the random noise on the measurement by the following two methods.

- Reducing IF BW
- Launch sweep averaging

Sweep averaging can be more effective than reducing IF bandwidth. However, comparing with reducing the IF bandwidth, it takes a longer time for sweep averaging in reducing noises, especially if many averages are required. Besides, changing IF bandwidth after calibration will result in uncertain accuracy.

IF BW

Intermediate Frequency Bandwidth (IF Bandwidth) is the width of the intermediate frequency band produced after mixing within the vector network analyzer's receiver. It is the bandpass width of the receiver's internal filter and represents the receiver's ability to filter signal frequency components. Reducing the IF receiver bandwidth reduces the random noise on the measurement. Each tenfold reduction in IF bandwidth lowers the noise floor by 10 dB. However, narrower IF bandwidth results in longer sweep time.

The range of the IF bandwidth is from 1 Hz to 10 MHz. By default, it is 100 kHz. The status bar at the

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

bottom of the screen shows the IF BW value. Here BW refers to IF Bandwidth.

Averaging State/Averaging Times

The averaging feature performs averaging on the measurement results several times, reducing the error arising from uncertain factors in a single measurement, reducing the effect of random noise on the measurement results, making the results more stable and accurate.

Check the checkbox of the **Averaging** menu to enable the averaging function. By default, it is disabled. Each time you enable the averaging function, the measurement restarts to perform the averaging operation again. If you select "Sweep" for the averaging type, then after enabling averaging, the averaging progress (Channel No. Avg = Average Count Completed/Total Average Counts) for the current channel is displayed at the lower-left part of the trace window. If you disable averaging function, the average count progress disappears.

Average count indicates the number of counts for averaging. Its range is from 1 to 65,535 (2^16 - 1). By default, it is 1.

Restart Averaging

Clears the current measurement data and restarts sweep averaging again for a new set of measurements. All the traces of the current active channel will be replotted.

The Restart Averaging function is not applicable to Point averaging. It is only valid when the averaging function is enabled.

Average Type

- **Point Averaging**

  Each data point is measured for a specified number of times, then averaged, before going to the next data point. The point averaging type is generally used in the scenarios where the frequency response is relatively stable or the measurement time is short. The sweep time of the point averaging is increased exponentially based on the average count.

- **Sweep Averaging**

  Computes averaging on subsequent sweeps until the required number of averaging sweeps is performed. This averaging type can reduce the noise in the whole frequency range, and especially effective for the measurement with fast changing frequency response. By default, the averaging type is Sweep averaging.

15.2 Smoothing

Trace smoothing averages a number of adjacent data points to smooth the displayed trace. The number of adjacent data points that get averaged is also known as smoothing aperture. The trace smoothing can reduce the peak-to-peak noise on the broadband measured data. It smooths the trace noise and will not increase measurement time significantly.

Enable Smoothing

By default, smoothing is disabled. When enabled, smoothing is applied to the current active trace.

As trace smoothing follows format in the data processing map, the formatted data is smoothed. If Data Format is Polar or Smith Chart, smoothing is automatically disabled.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Smoothing Aperture

Specifies the percent of the swept stimulus span (X-axis) to smooth. For example, for a trace that contains 101 data points, if the smoothing aperture is 11%, then the number of data points that are averaged is 11. The range of the smoothing aperture is from 0% to 25%. By default, it is 1%.

Smoothing Points

Specifies the number of adjacent data points to average. It can only be set to an odd number. By default, it is 3.

15.3 Group Delay Aperture

Group Delay

Group delay is the time delay of the amplitude envelope of a signal as it passes through a device or network. It represents the rate of change of phase shift with respect to frequency, indicating how different frequency components within a signal experience different transmission delays through the device. The group delay is a measure of phase distortion. It describes the transit time of a signal through a device versus frequency. If the group delay of a system is a constant value for all the frequencies, it indicates that the transmission speed of the frequency components is the same, which will not cause signal distortion.

The phase characteristics of a device typically consists of both linear and higher order (deviations from linear) phase-shift components.

- Linear phase-shift component represents average signal transit time, attributed to electrical length of test device.
- Higher-order phase-shift component represents variations in transit time for different frequencies. It is the source of signal distortion.

For a linear time-invariant (LTI) system, the relationship between input and output can be characterized by its frequency response.

$\tau(\omega) = -d\phi(\omega)/d\omega$, where ω indicates the angular frequency, $\Phi(\omega)$ indicates the frequency response;

$\tau(f) = -d\theta/df$, where θ indicates phase angle; f indicates the frequency.

Tip:

The group delay function is only valid when the data format is Group Delay. The setting is only valid for the current active trace.

Group Delay Aperture

Group delay aperture typically refers to the frequency resolution of the measurement system in group delay measurements.

During a group delay measurement, phase is measured at two closely spaced frequencies and then computes the phase slope. The frequency interval (frequency delta) between the two phase measurement points is called the aperture. Changing the aperture can result in different values of group delay. The computed slope (-delta phase / delta frequency) varies as the aperture is increased. Therefore, when you compare group delay data, you must know the aperture that was used in measurements.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Narrow aperture:** provides more details in phase linearity, making measurement susceptible to noise (smaller signal-to-noise ratio) and phase detector resolution.
- **Wide aperture:** provides less details in phase linearity, because some phase response is averaged out or not measured, making measurement less susceptible to noise (larger signal-to-noise ratio).

If the group delay aperture is set too large, it may lead to over-smoothing of the measurement results, thereby missing some important phase variation details; if set too small, it may result in increased measurement noise, affecting the stability and accuracy of the measurement results.

The number of adjacent data points can be set using any of the following methods. The three options set the aperture from different angles. Modifying one parameter will affect the other two parameters.

Formula: (Aperture Points - 1) / (Sweep Points - 1) = Percent of Span

(Sweep Stop Frequency - Sweep Start Frequency) / (Sweep Points - 1) = Sweep Points Interval

Sweep Points Interval * (Aperture Points - 1) = Aperture Frequency

- **Percent of Span:** The data points within this percentage of the current frequency span are averaged. The range is from (2 points/current number of points) to 100%. The span must contain at least two data points.
- **Points:** indicates the number of adjacent data points to average. By default, it is set to 11. It ranges from 2 to the current number of sweep points in the channel.
- **Frequency:** The data points within this frequency range are averaged. The frequency range must contain at least two data points.

When the frequency span or number of points is reduced so that the current Group Delay Aperture is not attainable, the Aperture is adjusted to the new frequency span or number of points.

16 Trigger

The trigger function causes the analyzer to select the trigger source and make a measurement.

16.1 Main

The settings in this menu are valid for the currently selected channel.

Trigger State

The trigger states include the following types. Check the checkbox of the specified trigger state, and the checkbox turns green, indicating it is the currently selected trigger state.

- **Hold:** The trigger hold state, indicating the channel does not receive any trigger signal and the trace will not update.
- **Single:** The channel receives a single trigger signal, then goes to the Hold state. At this time, the trace updates once and then stops.
- **Groups:** The channel only receives the specified number of triggers, then goes to Hold state. The

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

trace updates for a specified number of times, then stops.

The trigger count ranges from 1 to 2000000. By default, it is 1.

- **Continuous:** By default, continuous trigger is selected. The channel can receive an infinite number of trigger signals. The trace updates continuously.

Restart Trigger

Click **Restart**, then all the channels that stay not in "Hold" state will be triggered in sequence based on the channel ID. Other trigger parameters of the channel remain unchanged.

Trigger Source

Only when the analyzer is not sweeping, can a valid trigger signal be generated.

The following trigger sources are available.

- **External:** receives external signals via the rear-panel TRIG IN connector as the trigger source.
- **Internal:** It is the default setting for trigger source. When you select internal trigger source, the continuous trigger signals are generated by the analyzer.
- **Manual:** When you select "Manual" trigger source, click **Manual Trigger** to generate a trigger signal. You can also click the manual trigger icon on the quick operation toolbar at the top of the screen to generate a trigger signal.

  Only when you select "Manual" trigger source, can the **Manual Trigger** menu be enabled and the manual trigger icon on the quick operation toolbar at the top of the screen be valid.

Trigger Scope

Sets the scope where the trigger signal is sent. The available scopes include:

- **Global:** All channels that are not in "Hold" state will receive the trigger signal.
- **Channel:** default setting. Based on the channel ID, each time only one channel that stays not in "Hold" state can receive the trigger signal in sequence. It is recommended when you set the trigger source to Manual or External.

Trigger Mode

This option is enabled when you set **Scope** to "Channel" and set **Source** to "Manual" or "External".

The available trigger modes include:

- **Channel:** each trigger causes all the traces in the channel to be swept in the specified order.
- **Point:** each manual or external trigger causes one data point to be measured. Subsequent triggers go to the same trace until all the data points of the trace have been completely measured. Then other traces in the same channel are swept in the specified order.

  In the Point trigger mode, when you select "Single" or "Groups" trigger state, the trigger count is decremented by one after all data points on all traces in the channel are measured.
- **Sweep:** Each manual or external trigger signal causes all traces that share a source port to be swept in the specified order.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

When you select "Single" or "Groups" trigger state, the trigger count is decremented by one after all traces in all directions are swept.

16.2 Input

Only when the trigger source is set to "External", can the input configuration be valid. The external trigger signal is input from the rear-panel TRIG IN connector.

Global Trigger Delay

Sets the time interval during which the instrument waits to start the sweep operation after the trigger signal that meets the trigger conditions is received. Its range is from 0 s to 3 s. By default, it is 0 s.

Level/Edge

Sets the trigger condition. You can select the condition where the input trigger signal is triggered. The available choices include "High Level", "Low Level", "Positive Edge", and "Negative Edge".

16.3 Output

This configuration takes effect when the trigger source is set to **Internal** or **Manual**. It configures the output trigger signal, which is output via the TRIG OUT port on the rear-panel.

Enable or Disable the Trigger Output

Enables or disables outputting trigger signals from the rear-panel TRIG OUT connector. By default, it is disabled.

Polarity

- Positive Pulse: the output pulse is the positive pulse.
- Negative Pulse: the output pulse is the negative pulse. By default, it is Negative pulse.

Position

- Before Acquisition: sends a pulse immediately before starting the data acquisition.
- After Acquisition: sends a pulse immediately after completing the data acquisition. This is the default setting.

Per Point

- OFF: sends the trigger signal at each sweep. This is the default setting.
- ON: sends the trigger signal when acquiring each data point.

Pulse Duration

Indicates the pulse width of the output trigger signal. Its range is from 1.00 μs to 1 s. By default, it is 1.00 μs.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

17 Scale

The **Scale** menu mainly includes the vertical scale and related settings. You can set the scale, reference level, and reference position (and format) to determine the data trace on the VNA screen.



17.1 Main

Auto Scale

Click **Auto Scale** to set value of the vertical divisions and reference value to fit the active data trace within the grid area of the screen.

- The analyzer determines the smallest possible scale factor that will allow all the displayed data to fit onto 80 percent of the vertical grid.
- The reference value is auto adjusted to center the trace on the screen.

The stimulus values and reference position are not affected when performing the auto scale operation.

Auto Scale All

Click **Auto Scale All** to automatically scale all data traces in the active window to fit vertically within the grid area of the screen.

Scale

Click **Scale** to set the vertical scale manually. That is, set the value of the vertical divisions of a rectangular display format.

The unit of the scale varies according to the trace data format. In Polar and Smith formats, scale sets the value of the outer circumference.

Reference Level

Click **Reference Level** to set the value of the reference line in the vertical axis in a rectangular format. The reference level varies based on the trace data format. In Polar and Smith formats, reference level is invalid.

Reference Position

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Adjusts the position of the reference line to move it up and down in the vertical direction. Click **Reference Position** to set the position of the reference line in the vertical axis in a rectangular format manually. 0 is the bottom line of the screen and 10 is the top line. By default, it is set to 5 (middle). In Polar and Smith formats, reference position is invalid.

17.2 Electrical Delay

Electrical Delay refers to the linear phase delay caused by the physical length and dielectric constant when a signal transmits in the device under test. The electrical delay of the VNA specifies the value of delay added or removed, in Time or Distance. It can be used to compensate for the linear phase shift through a device to make the measurement results more accurate. You can set the electrical delay independently for each measurement trace.

Set the Electrical Delay Time/Distance

- Click **Electrical Delay Time** to set the electrical delay time, expressed in s.
- Click **Electrical Delay Distance** to set the electrical delay distance.

  Click **Distance Units** to set the unit of the electrical delay distance. The available units are meter, feet, and inch.

The "Electrical Delay Time" and "Electrical Delay Distance" are correlative. Modifying one parameter will affect the other.

Velocity Factor

Velocity Factor (VF) is used to describe the ratio of the transmission speed of electromagnetic waves in the medium relative to the speed of light in the vacuum. VF = v/c. Here, v indicates the transmission speed in the medium; c indicates the light speed.

The velocity factor is smaller than or equal to 1, which is related to the dielectric constant (εr) of the medium. It directly affects the transmission line electrical length calculations, time delay compensation, and calibration accuracy.

Click the input field of **Velocity Factor** to set the velocity factor that applies to the medium of the device that was inserted after the measurement calibration. The velocity factor of the common transmission medium is as follows:

- Polyethylene dielectric cable: 0.66
- PTFE dielectric: 0.7
- Speed of light in a vacuum: 1.0

17.3 Constants

System Impedance

The actual impedance of the VNA hardware test port is generally 50 Ω. The system impedance can be changed for measuring device with an impedance other than 50 Ω, such as waveguide devices.

VNA mathematically transforms and displays the measurement data as though the VNA ports were the

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

specified impedance value. Physically, the test ports are always about 50 Ω.

Phase Offset

Click input field of **Phase Offset** to set the phase offset. By default, it is 0°.

- **Optimize the display effects of the phase measurement:** The method is the same as modifying the reference level in magnitude measurement. Adjust phase response to make it in the middle of the screen or aligned.
- **Simulate the expected phase offset in the measurement**: If a cable is connected to the instrument and the cable length may cause some phase offset for the measurement setup, then you can set the phase offset to correct the offset.

Magnitude Offset

Click **Magnitude Offset** to offset the magnitude data of the current active trace by a fixed value in dB in the vertical coordinate (Y-axis).

Magnitude Offset Slope

The magnitude offset slope shows that the magnitude offset varies with the frequency. The offset slope begins at 0 Hz, in dB/GHz.

Click **Magnitude Offset Slope** to offset the data trace by a value that changes with the frequency, making the current active trace to move in the vertical coordinate (Y-axis) based on the set magnitude offset slope. On the screen, you can see the slope varies with the frequency dynamically while the trace moves up and down, making it vary linearly within the frequency range.

18 Math Operation

The VNA supports saving the current data trace to the internal memory, and performing math operations on the current data trace and memory trace. The data analysis function includes the data transform of S-parameters, enabling statistics for the data of the active trace, computing the mean, standard deviation, and peak-to-peak values, and enabling the limit test.

18.1 Memory

Data->Memory

Click **Data->Memory** to save the measurement data of the active trace to the memory. Click this menu for multiple times, and the latest data of the current active trace will be saved to the memory. You can only store one memory trace data for each trace.

Data Math

Before performing the data math operation, perform "Data->Memory" operation first to ensure that there are available active trace data in the memory. Then perform the specified type of math operation on the current measurement data of the active trace and the trace data stored in the memory and then display them in the specified format.

The math operation types supported include:

- **Off:** By default, it is disabled. No math operation is performed on the current measurement data.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Data / Memory:** Current measurement data is divided by the trace data in the memory. It is used for ratio comparison of two traces.
- **Data - Memory:** Data in the memory is subtracted from the current measurement data.

  It can be used for storing a measured vector error, then subtract this error from the DUT measurement to gain a more accurate result.
- **Data + Memory:** Current measurement data is added to the data in the memory. It can be used to add measurement results.
- **Data * Memory:** Current measurement data is multiplied by the data in the memory. It can be used for some special analysis requirements.

Normalize

The **Normalize** menu functions the same as performing "Data->Memory" first, and then "Data / Memory". Current measurement data is divided by the trace data in the memory. The normalized operation results will be shown on the current active trace, and the data math will be automatically switched to "Data / Memory".

The Normalize operation aims to perform one reference measurement in the preset condition and saves the measurement data to the memory as the reference data. Then perform the actual measurement, compare the actual measurement data with the reference data, and perform data math operation. The Normalize operation can eliminate the error caused by the system, making the measurement results more accurate.

Display

Sets the data display mode of the currently selected trace.

- **Data Trace:** displays the current measurement data of all the created traces.
- **Memory Trace:** displays the trace that was put in memory. If no memory data of a certain trace is found, then the current measurement data of the trace will not be displayed.
- **Data and Memory:** displays both the current measurement data of the trace and the trace that was put in memory.

18.2 Analysis

Parameter Conversion

The vector network analyzer supports parameter conversion function. You can convert the measurement results of the S-parameter to the desired parameter mode. After conversion, the trace curve will change. Engineers can better observe the characteristics of the components.

Table 18.1 Conversion Type

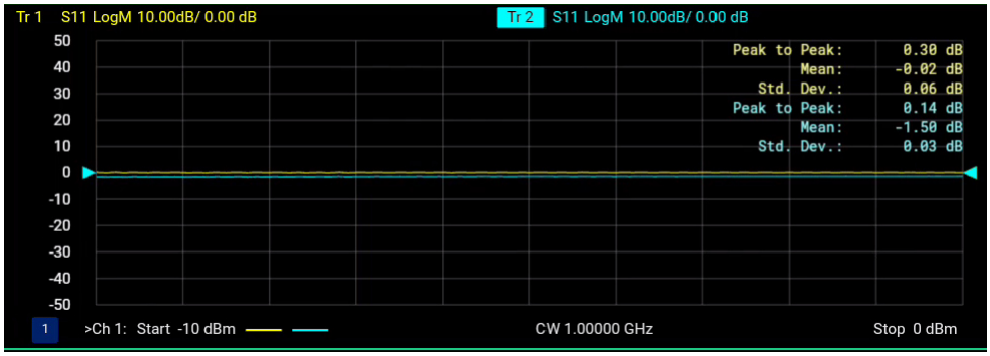
Menu Function Key	Conversion Function	Formula
Off	Conversion disabled	-

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Menu Function Key	Conversion Function	Formula
Z-Reflect	Impedance (Zr) in reflection measurement	$Z_r = Z_{0a} \times \frac{1 + S_{aa}}{1 - S_{aa}}$
Z-Transmit	Impedance (Zt) in transmission measurement	$Z_t = \frac{2 \times \sqrt{Z_{0a} \times Z_{0b}}}{S_{ab}} - (Z_{0a} + Z_{0b})$.
Z-Trans-Shunt	Impedance (Zt) transmission shunt	$Z_t = \frac{1}{Y_t}$
Y-Reflect	Admittance (Yr) in reflection measurement	$Y_r = \frac{1}{Z_r}$
Y-Transmit	Admittance (Yt) in transmission measurement	$Y_{0a} = \frac{1}{Z_{0a}} Y_{0b} = \frac{1}{Z_{0b}}$ $Y_t = \frac{2 * \sqrt{Y_{0a} * Y_{0b}}}{S_{ab}} - (Y_{0a} + Y_{0b})$
Y-Trans-Shunt	Admittance (Yt) in transmission measurement	$Y_t = \frac{2 * \sqrt{Y_{0a} * Y_{0b}}}{S_{ab}} - (Y_{0a} + Y_{0b})$
1/S	Inverse S-Parameter	$\frac{1}{S_{ab}}$
Conjugation	Complex conjugate number	-

Statistics

Check the checkbox of **Statistics**, and the checkbox is filled in green. The statistical function for the current active trace is enabled. The statistics information is displayed at the upper-right corner of the window, as shown in the figure below. Uncheck the checkbox of **Statistics**, then the statistical function for the current active trace is disabled. The statistics information will not be displayed at the upper-right corner of the window.



Click the input field of **Statistics** to set the statistics range for the current trace.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Full Span:** By default, it is full span. That is the whole X-axis, which is the same as the sweep range of the trace itself.
- **User 1 to User 16:** makes statistics within the user-defined span. These settings are the same as what you set for the **Search Domain** sub-menu under **Search**.

  At most, 16 user-defined range can be set. You need to set **Domain Start** and **Domain Stop** for each user-defined range. The user-defined ranges are independent of each other and can overlap with each other.

The statistics information includes the following items:

- Peak to Peak: indicates the peak-to-peak value of the trace within the statistical range.
- Mean: indicates the mean value of the trace within the statistical range.
- Std. Dev.: indicates the standard deviation of the trace in the statistical range.

18.3 Limit Test

Limit test compares the measured data with the defined limit value to judge whether the response data of the measurement meet the PASS/FAIL conditions to obtain the test result.

To configure the limit test parameters, click **Limit Test** in the "Math" menu. The measurement interface of the limit test is as shown in the figure below.



1. Limit Table

   Used to define whether the response data of measurement complies with the pass standards. Each line has a judging standard. By default, the limit table is disabled. When enabled, the limit table displaying the current active trace is displayed below the window.
2. Limit Line

   The judge standards defined for each line in the limit table include begin stimulus (X-axis) and end stimulus (X-axis); begin response (Y-axis) and end response (Y-axis). They form a limit line on the screen. The limit line is a dotted line, and its color is the same as the trace color. For each measurement trace, you can set multiple discrete limit lines to test the characteristics of the response of the test component.
3. Test Result

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

When you enable the limit line, the result is displayed in the window. The data points that fail are displayed in red.

Tip:

The limit line and limit test are not applicable to **Smith Chart** or **Polar Chart** display format. If the limit line and limit test are both enabled, when you select Smith or Polar format, the system exits the limit test automatically.

18.3.1 To Enable or Disable the Limit Test

Enable/Disable

By default, the limit test is disabled. When enabled, the measurement data trace will compare with the set limit line to display PASS or FAIL result.

Limit Line

By default, the limit line is disabled. When enabled, the limit line is displayed on the screen. When you enable the limit test and disable the limit line, the limit test will still continue.

Test Fail Sound

By default, sound on test failure is disabled. When enabled, the beep sounds when a point on the data trace fails the limit test.

18.3.2 Limit Table

Configure the Limit Table

You can add and edit the limit line information in the limit table. Each line in the limit table is a test rule for the limit line. To add a limit line, modify the "Type" parameter to a value other than "OFF", then a new line will be added automatically in the table. The "Type" parameter in the new line turns out to be "OFF".

- Click the drop-down button of **Type** to select OFF, MAX, or MIN.
  - OFF: disables the limit line. That is, the limit line is invalid.
  - MAX: The MAX value will fail measurements above this limit.
  - MIN: The MIN value will fail measurements below this limit.
- Begin Stimulus: a value for the signal source to begin outputting stimulus signal. Click the input field under Begin Stimulus parameter to input the value.
- End Stimulus: a value for the signal source to stop outputting stimulus signal. Click the input field under End Stimulus parameter to input the value.
- Begin Response: a value for the signal source to begin measuring DUT output response. Click the input field under Begin Response parameter to input the value.
- End Response: a value for the signal source to stop measuring DUT output response. Click the input field under End Response parameter to input the value.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Load/Save Limit Table

Click **Load Table**, then the file management interface is displayed. Select the specified limit table file suffixed with "*.csv", then click **Confirm** to load the specified file to the limit table of the current active trace.

Click **Save Table**, then the file management interface is displayed. Select the path to save the file, then click **Save** to save the limit table of the current active trace as a file suffixed with "*.csv". If the file with the same filename already exists in the current path, then overwrite the original file.

For details about the file management method, refer to File.

Example of Limit Table File (*.csv)

The limit table file suffixed with "*.csv" must contain the header in the following format.

```
"# VNA Limit Test"
"# Revision: 1.00"
TYPE,BEGIN STIMULUS,END STIMULUS,BEGIN RESPONSE,END RESPONSE
```

1. The first line indicates the limit test type of the instrument.
2. The second line indicates the limit test version.
3. The third line indicates the title of the output parameters starting to output from the fourth line.
4. The fourth line indicates outputting segment parameters and data.

18.3.3 Global Pass/Fail

Enable or Disable Global Pass/Fail

The Global Pass/Fail feature enables or disables monitoring the status of all measurements.

- **ON:** when you enable Global Pass/Fail, the status of all the measurements is displayed in the system status area. If any one of the traces fails in the limit line test, the result shows "FAIL"; if no failed measurement exists, it displays "PASS".
- **OFF:** when you disable Global Pass/Fail, the status of all the measurements is not displayed in the system status area.

Global Pass/Fail Policy

Sets the policy to determine the global pass/fail status.

- **All Tests:**

  If all the limit tests pass, then the Global Pass/Fail status shows PASS.

- **All Measurements:**

  If all the measured data points fall within the established test limits and limit test is ON, the Global Pass/Fail status will show PASS; if one measurement does not have limit test ON, the Global Pass/Fail status will show FAIL.

  This setting is stricter. If all the test pass, it shows PASS; otherwise, it shows FAIL.

  In this mode, if one measurement does not have Limit Test ON, the Global Pass/Fail status will show FAIL.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

18.4 DTF (Option)

The DTF (Distance To Fault) feature identifies fault location and distance by analyzing reflected signals in the frequency domain. It calculates the distance from the fault point to the reference plane based on reflection characteristics and frequency response. Its working principle is as follows:

- Signal reflection mechanism:

  When electromagnetic waves encounter impedance discontinuities (such as short circuit, open circuit, or loose connector) during transmission, a portion of the energy is reflected back to the source. The amplitude and phase of the reflected signal are related to the characteristics of the fault point (such as the variation of impedance) and its location.

- Conversion between time domain and frequency domain:

  The VNA instrument measures the reflection coefficient (e.g., S□□ parameter) in the frequency domain, then transforms the frequency domain data into time domain response through IFFT (Inverse Fast Fourier Transform). At this point, the horizontal axis of the time domain waveform represents time, which can be further converted to distance through the known velocity factor of signal transmission in the medium.

- Distance calculation:

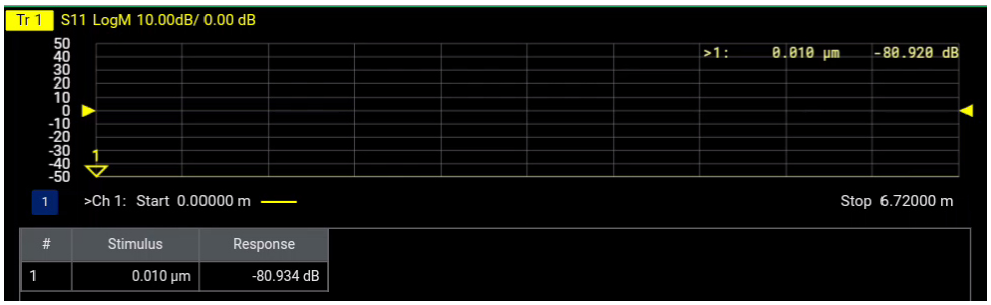
  DTF (L) can be obtained with the following formula: ***L = (c*t*VF)/2***. Here, c indicates speed of light, t indicates round-trip of the reflection signal, and VF indicates the velocity factor.

You can configure the following DTF-related parameters:

DTF State

By default, DTF state is disabled. Only when the sweep type is set to Linear Frequency for the current trace, can you enable the DTF function. Otherwise, the DTF function is unavailable to use. DTF and TDA are mutually exclusive. When you enable either one of them, the other will be disabled automatically.

When DTF is enabled, the window presents the relationship between the transmission loss (Y-axis) and distance (X-axis). The measurement readout is displayed at the upper-right corner of the window, and the marker indicates the fault position. The unit of the X-axis can be selected from **Distance Unit**. By default, the unit is Meter. The available units include Meter, Feet, and Inch.



Distance Unit

After enabling DTF, the trace X-axis becomes distance display; the distance unit defaults to meters, but

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

feet or inches can also be used.

Start Distance

Sets the start position of the X-axis after enabling DTF; default 0, supported setting range 0 to the maximum alias-free distance.

Stop Distance

After enabling DTF (Distance-To-Fault) measurement, the stop distance of the X-axis set by default to the maximum measurable distance under the current configuration. The supported setting range is 0 to the maximum alias-free distance.

If the set stop distance exceeds the current maximum alias-free distance, the system will automatically pop up a prompt asking whether to meet this setting by modifying the number of scan points. If "Auto Set Scan Points" is selected, the system will automatically calculate and apply the minimum number of scan points that satisfies the condition (the number of points ranges from 201 to 100001).

Alias-Free Distance

The theoretical maximum measurable distance of the instrument under the current test settings. To ensure the accuracy of measurement data, the actual physical length of the cable under test must not exceed this range. It can be calculated by the following formula:

$$D_{max} = \frac{c\ Vf}{2\ddot{A}f} = \frac{c\ Vf}{2} = \frac{N-1}{f_{stop} - f_{start}}$$

Where:

c: Speed of light

VF: Velocity factor

f_{stop} - f_{start}: Frequency span

N: Number of scan points

Δf: Frequency step

Distance Resolution

Under the current test configuration, the minimum distance interval at which the system can distinguish two adjacent targets. This indicator is related to the frequency span of the test signal; the larger the frequency span, the higher the distance resolution (i.e., the smaller the value), and the test accuracy improves accordingly.

$$D_{res} = \frac{c\ Vf}{2f_{stop} - f_{start}}$$

Where:

c: Speed of light

VF: Velocity factor

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

f_{stop} - f_{start}: Frequency span

Auto Set Scan Points

This function can automatically calculate and apply the minimum number of scan points that satisfies the test conditions based on the currently set stop distance (the system lower limit is 201 points). This mechanism aims to minimize test time and improve work efficiency while ensuring test accuracy.

Cable Loss

During cable transmission, electromagnetic waves will result in attenuation due to loss. Therefore, the longer the distance, the smaller the signal amplitude. If the reflection signal of the distance faults is too weak, it may cause misjudgment of the fault. Therefore, you need to compensate the signal attenuation caused by cable transmission.

Cable loss is the attenuation characteristics parameters of the signal in the cable transmission, expressed in dB/m. Configuring the cable loss can compensate the attenuation of stimulus signal in different positions of the cable, making the peak amplitude of the reflection signal at fault points of different distances in the transmission cable more precise, ensuring that the fault points can be identified accurately.

Velocity Factor

Velocity factor is the ratio of the transmission velocity of the signal in a medium to the speed of light in vacuum. For details, refer to Electrical Delay.

Through the velocity factor and the X-axis coordinate (time) where the specified measurement marker is located, you can obtain the distance (length) between the test port and the fault point.

Window Function

The available window types are Rectangular, Hanning, Hamming, Gaussian, and Flattop. You can select a proper filter type by referring to the following table according to the actual measurement requirements.

Window Function	Spectral Leakage	Amplitude Accuracy	Frequency Resolution
Gaussian	Moderate	Good	Medium
Flattop	Good	Excellent	Poor
Rectangular	Poor	Poor	Excellent
Hanning	Good	Medium	Good
Hamming	Good	Medium	Good

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

18.5 Time Domain (TDA Option)

The VNA instrument is usually used to analyze the characteristics of the frequency domain of the device, but also can use the frequency domain parameters to calculate the inverse Fourier transform (IFFT) to obtain the characteristics of time-domain of the device.

TDA (Time Domain Analysis) feature enables the measured frequency domain data to be converted to the time domain data through math calculation, and performs an analysis of the time domain characteristics of the DUT.

The basic principle of the TDA feature is: Send a signal on the transmission cable, and if an impedance mismatch (such as Open Circuit, Short Circuit, and impedance variation) occurs during transmission, some signals will be reflected. Through measuring the delay between the transmission pulse and reflection pulse, in combination with the transmission rate of the signal on the transmission cable, you can determine the impedance distribution and fault location on the transmission cable.



- **Transform State:**

  By default, it is disabled. When it is enabled, the coordinate in the window switches to the time-domain mode. The measurement trace is displayed in the window, with the X-axis coordinate representing time.

- **Start Time:**

  Sets the transform start time that is displayed on the analyzer screen, as ① indicated in the above figure.

- **Stop Time:**

  Sets the transform stop time that is displayed on the analyzer screen, as ② indicated in the above figure.

- **Center Time:**

  Sets the transform center time that is displayed in the center of the analyzer screen, as ③ (a T icon) in the above figure.

- **Span Time:**

  Sets the transform span time that is split on either side of the center value. It is the difference

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

between the transform stop time and transform start time in the window.

The start time, stop time, center time, and span time are interdependent. Changing the start time or stop time will also change the values of center time and span time. The start time and stop time will also be changed when modifying the center time or span time.

Transform Type

Three transform types are available. They are three variations on how the time domain transform algorithm is applied to the frequency domain measurement.

- **Band Pass:** applicable to scenarios where resolution requirement is not high and require rapidly locating problems in the wideband.

  It features wide pulse and low resolution, applicable for measuring band limited devices.

- **Low Pass Impulse**

  Narrow pulse, high resolution, easy to identify the tiny impedance variations in devices that pass low frequencies.

- **Low Pass Step:**

  Step signal, sensitive rising edge, easy to identify inductive and capacitive discontinuities in devices that pass low frequencies.

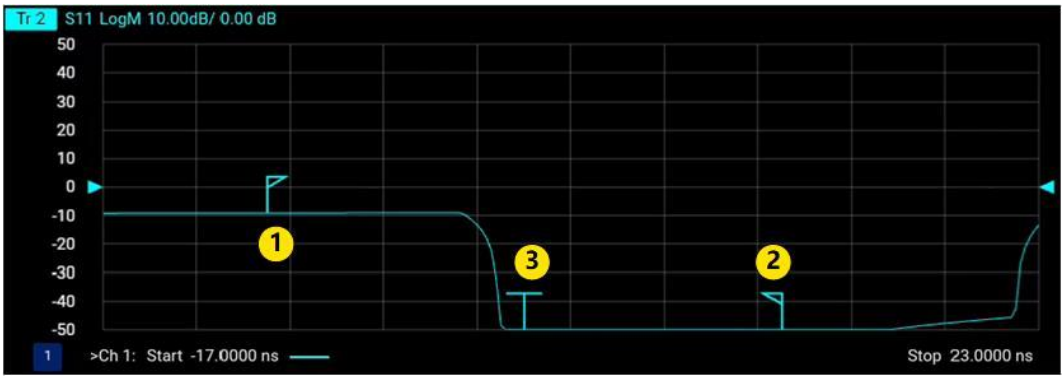
Tip:

If you choose Low Pass Impulse or Low Pass Step, configure the parameters as follows:

Start frequency = Frequency step; Stop frequency = Start frequency × Number of sweep points.

18.6 Time Gating (TDA Option)

The time domain gating feature converts the raw measurement data from the frequency domain to the time domain. In the time domain, data are filtered according to filtering conditions. The filtered data are then subjected to Fourier transform to convert them back into the corresponding frequency domain characteristic curve for further analysis. Time domain gating is a commonly used method to filter the frequency domain signal.



You can set the following parameters:

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Gating State:**

  By default, gating is disabled. When enabled, the measurement data will be filtered according to the set time gating condition. Before enabling the gating, first enable Time Domain (TDA Option) to convert the current measurement trace to be displayed in the time domain coordinate system.

- **Start Time:**

  Sets the start time for the gate. It is indicated as ① (a flag icon) in the above figure.

- **Stop Time:**

  Sets the stop time for the gate. It is indicated as ② (a flag icon) in the above figure.

- **Center Time:**

  Sets the center time for the gate. It is indicated as ③ (a T icon) in the above figure.

- **Span Time:**

  Sets the range to either side of the center value of the area that is affected by the gating function. It is the difference between the stop gate time and start gate time in the window.

The start time, stop time, center time, and span time are interdependent. Changing the start time or stop time will also change the values of center time and span time. The start time and stop time will also be changed when modifying the center time or span time.

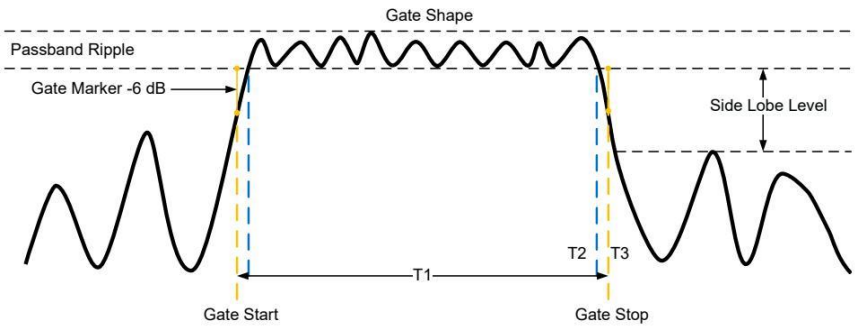
Gate Type

Defines the type of filtering that will be performed for the gating function.

- Band Pass: keeps the response data within the gate span.
- Notch: removes the response data within the gate span.

Gate Shape

Defines the filter characteristics of the gate function. The available gate shapes include Maximum, Wide, Normal, and Minimum. The definitions are as follows:



Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Gate Shape	Passband Ripple	Sidelobe Level	Cut-off Time	Minimum Gate Span
Min.	±0.1 dB	-48 dB	1.4/Frequency Span	2.8/Frequency Span
Normal	±0.1 dB	-68 dB	2.8/Frequency Span	5.6/Frequency Span
Width	±0.1 dB	-57 dB	4.4/Frequency Span	8.8/Frequency Span
Max.	±0.01 dB	-48 dB	12.7/Frequency Span	25.4/Frequency Span

Cut-off Time indicates the time between the stop time (-6 dB on the filter skirt) and the peak of the first sidelobe. The diagram above shows the overall gate shape and lists the characteristics for each gate shape.

- T1 is the gate span, which is equal to the stop time minus the start time.
- T2 is the time between the edge of the passband and the 6 dB point, representing the cut-off rate of the filter.
- T3 is the time between the 6 dB point and the edge of the gate stopband.
- For all the filter shapes, T2 is equal to T3, and the filter is the same on both sides of the center time.

Minimum gate span is twice the cut-off time. Each gate shape has a minimum recommended gate span for proper operation. This is a consequence of the finite cutoff rate of the gate. If you specify a gate span that is smaller than the minimum span, the response will show the following effects:

- Distorted gate shape that has no passband
- Distorted shape
- Incorrect indications of start and stop time
- May result in increased sidelobe level

18.7 Time Domain Setup (TDA Option)

Channel Coupling

Enables or disables the coupling state for the specified parameters of all the traces for the same channel.

- **Coupling State**:
  - ON: By default, it is enabled. When it is enabled, modifications of the specified parameters for the current active trace are global to all the traces of the current channel.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- OFF: When it is disabled, modifications of the parameters for the current active trace will not affect the other traces of the current channel.

- **Transform Parameters:** selects the time domain transform related parameters that support channel coupling.
  - Stimulus: the "Start Time", "Stop Time", "Center Time", and "Span Time" settings in the **Time Domain** menu.
  - State (On/Off): the "Transform State" setting in the **Time Domain** menu.
  - Window: the "Kaiser Beta" and "Impulse Width" settings in the **Time Domain Setup** menu.
  - Mode: the "Transform Type" setting in the **Time Domain** menu.

- **Gating Parameters:** selects the time domain gating related parameters that support channel coupling.
  - Stimulus: the "Start Time", "Stop Time", "Center Time", and "Span Time" settings in the **Time Gating** menu.
  - State (On/Off): the "Gating State" setting in the **Time Gating** menu.
  - Shape: the "Gate Shape" setting in the **Time Gating** menu.
  - Type: the "Gate Type" setting in the **Time Gating** menu.

Marker Mode

Sets cursor distance-calculation mode to Reflection or Transmission; defaults to auto based on active-trace measurement parameters.

Units

Distance display unit; selectable: Meters, Inches, Feet.

Velocity Factor

Indicates the ratio of the transmission velocity of the signal in a medium to the speed of light in vacuum. For details, refer to Electrical Delay.

Through the velocity factor and the X-axis coordinate (time) where the specified measurement marker is located, you can obtain the distance (length) between the test port and the destination point.

Window Function

In the TDA test, you can modify **Kaiser Beta** and **Impulse Width**. Beta parameter is inversely proportional to impulse width, and they are interdependent.

- The greater the Beta value, the narrower the impulse width and the higher the time resolution, applicable for detecting multiple impedance discontinuities at close range, but sidelobe suppression is relatively weak.

- The smaller the Beta value, the wider the impulse width and the stronger the sidelobe suppression, applicable for detecting weak reflection or long distance fault, but the time resolution is

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

relatively low.

18.8 Skew Measure (TDA Option)

In high-speed differential systems, the time difference or length difference between two identical source signals in the transmission path is called signal skew.

The skew test is based on Time Domain Reflectometry (TDR) technology and accurately quantifies the skew value by transmitting a low-pass step signal to the DUT, comparing the step response time of the reference trace with the measured trace, which can be used for the relative deviation calibration of differential signals.

To use the skew test feature, you need to install the TDA option first.

To configure the skew test parameters, click **Skew Measure** in the "Math" menu.

- **Skew State**

  By default, it is disabled. When it is enabled,

  - Click **Time Domain** > **Transform State** to enable the transform state.
  - Click **Time Domain** > **Transform Type** to select "Low Pass Step".
  - Set the trace data format for the skew test to Real.
  - Start the skew test for the current trace. The skew test result for the trace is displayed at the top part of the window.

- **Measure Threshold**

  It is used to set the measurement point for step voltage. By default, it is 50%. It means measurements are taken at 50% of step voltage by default. Its range is from 10% to 90%. It will be automatically adjusted to its boundary value when the value you set is out of its range.

- **Skew Limit**

  By default, it is 0 s. Its range is from -1.0e+15 s to 1.0e+15 s.

  If the absolute value of the time skew exceeds the limit value, the test fails, displaying "Fail" following the test result; otherwise it shows "Pass".

- **Reference Trace**

  The reference trace is used to compare with the current trace, which must be in the same channel as the current trace. To select the reference trace, you need to enable the skew state.

18.9 Delay Measure (TDA Option)

Signal delay refers to the total time for a single signal to transmit from the transmitter to the receiver. It directly reflects the length of the transmission path, media characteristics, or delay characteristics of the component. Delay testing is based on the low-pass step mode of Time Domain Reflectometry (TDR). By measuring the time difference between the stimulus of a step signal and the corresponding response of the Device Under Test (DUT), it enables **absolute delay measurement** without requiring a reference trace. This method is particularly suitable for characterizing single-path transmission characteristics.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

To use the delay test feature, you need to install the TDA option first.

To configure the delay test parameters, click **Delay Measure** in the "Math" menu.

1. Delay State

   By default, it is disabled. When it is enabled,

   - Click **Time Domain** > **Transform State** to enable the transform state.
   - Click **Time Domain** > **Transform Type** to select "Low Pass Step".
   - Set the trace data format for the delay test to Real.
   - Start the delay test for the current trace. The delay test result for the trace is displayed at the top part of the window.

2. Measure Threshold

   It is used to set the measurement point for step voltage. By default, it is 50%. It means measurements are taken at 50% of step voltage by default. Its range is from 10% to 90%. It will be automatically adjusted to its boundary value when the value you set is out of its range.

18.10 Automated Test (TAS Option)

This option is supported only by the DNA6000, DNA6000-R and DNA5000 series products.

Refer to *Automated Test Operation Guide* for detailed operating instructions.

19 Setup

This menu allows you to perform quick operation for some functions, save, recall, and manage files.

19.1 Layout

In the Layout menu, you can quickly add, delete, and select the trace, channel, window, and sheet. For detailed information about the trace, channel, window, and sheet, refer to descriptions in "Trace" and "Channel".

- Click **Add Trace**, **Add Channel**, **Add Window**, or **Add Sheet** to quickly add a trace, a channel, a window, or a sheet. They are set to be activated.
- Click **Delete**, and the sub-menu shows the existing sheet, window, channel, and trace. If a green dot exists at the left side of the specified item, it indicates that the item is activated. Click any of the items (trace, channel, window, or sheet) from the drop-down list of **Delete** to delete the desired one.
- Click **Select**, and the sub-menu shows the existing sheet, window, channel, and trace. If a green dot exists at the left side of the specified item, it indicates that the item is activated. Click any of the items (trace, channel, window, or sheet) from the drop-down list of **Select** to select the desired one to be activated. Once selected, a green dot will be highlighted at the left part of the selected item.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

19.2 Sweep Setup

The sweep setup menu has the same function as the **Main** sub-menu of **Sweep**. For detailed operations, refer to Main in "Sweep".

19.3 S-Parameter

The sub-menus of the **S-Param** in the S-Parameter menu are the same as that in **Measure** >**S-Param**. For detailed descriptions, refer to S-Parameter in the "Measure" menu.

19.4 Display Setup

Window Max

By default, Window Max is disabled. When you have created multiple windows, multiple windows are displayed on the screen simultaneously. When Window Max is enabled, the currently selected window will be maximized, and the "Window Max" label is displayed at the upper-right corner of the window.

In multi-pane windowing state, you can double-click the specified window to maximize the selected window; double-click it again to exit the "Window Max" state.

Grid Line

Sets the grid line in the window to "Solid" or "Dotted". By default, it is Solid.

Show Table

- None: default setting. The table is not displayed below the window.
- Marker: a marker table is displayed below the window.
- Limit: a limit test table is displayed below the window.
- Segment: a segment sweep table is displayed below the window.

Layout Mode

In multi-pane windowing, you can select the layout mode.

- Auto: default layout mode. The window size can be adjusted automatically based on the number of windows. Different windows can be different in size. The windows cover the whole window display area.
- Fixed: Adjusts the size of the window automatically based on the number of windows. Each window size is the same.

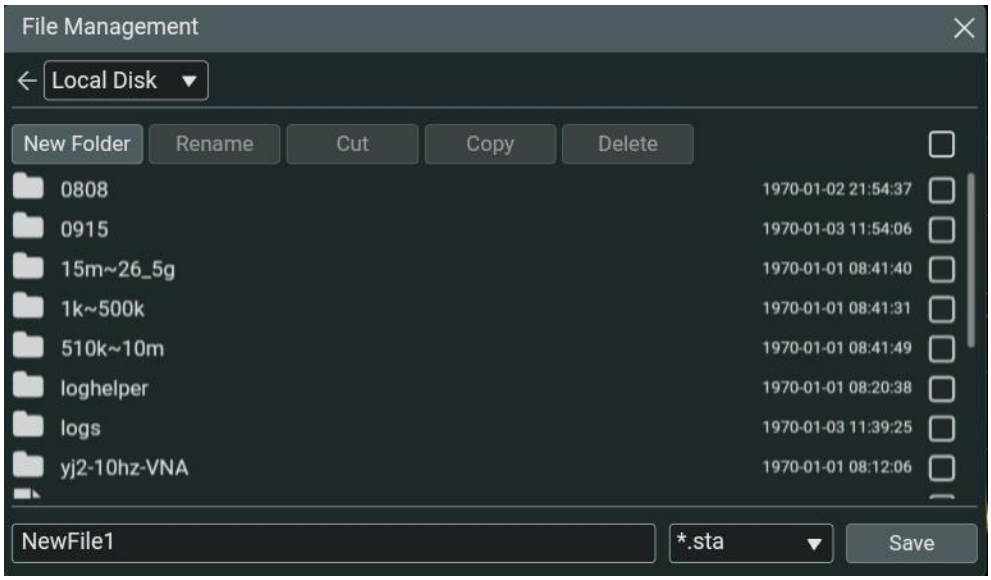
Max Window Count

By default, up to 9 windows are supported to be displayed in the window display area. The available max. window count can be set to 1, 2, 4, 6, and 9. When the number of created windows exceeds the max. number of windows that can be displayed in the window display area, click **Pg Up** or **Pg Dn** to page up or page down the display interface to find the specified window.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

19.5 Save

The Save operation is performed in the file management interface, as shown in the figure below.



For the file management operation, refer to File. The procedures for saving the file are as follows:

- **Set the filename:** click the input field of the filename, then the virtual keypad is displayed. Input the filename with the virtual keypad.
- **Select the file format:** click the drop-down button of the file format to select the desired format.
- **Save:** click **Save** to save the file with the specified filename and format to the current selected path.

Save State

Click **Save State**, then the file management interface is displayed. Save the current state information of the instrument to the specified path with the filename suffixed with ".sta".

Save CalSet

Click **Save CalSet**, then the file management interface is displayed. Save the current calibration sets information of the instrument to the specified path with the filename suffixed with ".cal".

State+CalSet

Click **State+CalSet**, then the file management interface is displayed. Save the current state and calibration sets information of the instrument to the specified path with the filename suffixed with ".csa".

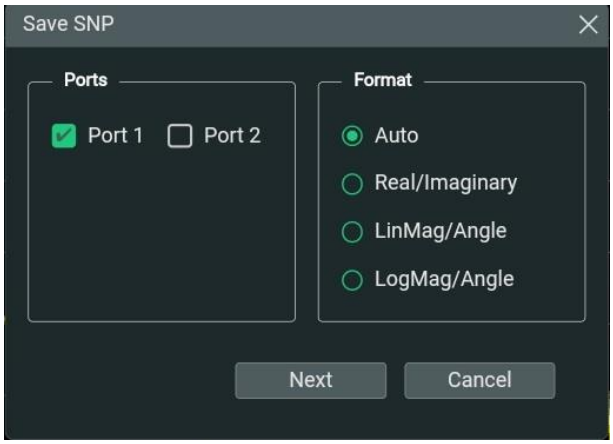
Save SNP

The *.snp file format is also known as Touchstone format. The "*.snp" data are usually used to gather all S-parameters for a fully corrected measurement. The file in Touchstone format is suffixed with "*.snp". Here, n indicates the number of ports. For example, "*.s2p" indicates a 2-port network; "*.s4p" indicates a 4-port network. The "*.snp" file contains the header information, stimulus data, and a response data pair for each S-parameter measurement. The file is a text file, and can be opened with

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Notepad.

Click **Save SNP**, then the "Save SNP" interface is displayed.



Select the specified port and format of the data to be saved, then click **Next**. The file management interface is displayed. Save the current S-parameter information of the instrument to the specified path, with the filename "*.s1p" (single port data of Port1 or Port 2) or "*.s2p" (Dual-port data of both Port1 and Port2).

Save CSV

Click **Save CSV**, then the "Save CSV" interface is displayed. Select the specified scope and format of the data to be saved, then click **Next**. The file management interface is displayed. Save the current S-parameter information of the instrument to the specified path with the filename suffixed with "*.csv".

Screenshot

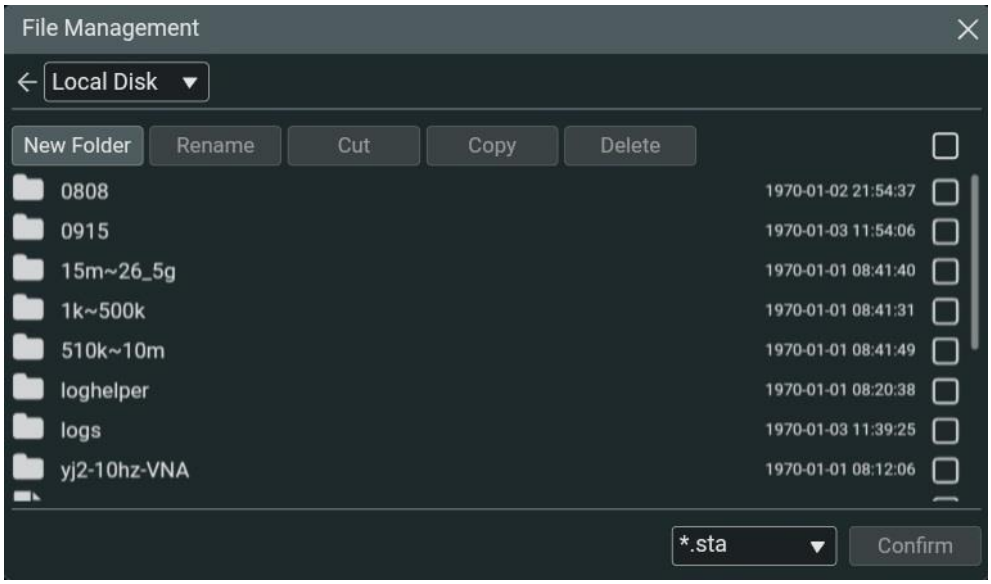
In the screenshot interface, configure the settings for Color, Invert, Header, Overlay, File Name, Format and File Path. Click **Screenshot** button to complete the screenshot capture.



19.6 Recall

The Recall operation is performed in the file management interface, as shown in the figure below.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



For the file management operation, refer to File.

Recall State

Click **Recall State**, then the file management interface is displayed. Select the specified "*.sta" file from the specified path, click **Confirm** to complete the recalling operation.

State+CalSet

Click **State+CalSet**, then the file management interface is displayed. Select the specified "*.csa" file from the specified path, click **Confirm** to complete the recalling operation.

Recall CalSet

Click **Recall CalSet**, then the file management interface is displayed. Select the specified "*.cal" file from the specified path, click **Confirm** to complete the recalling operation.

Recall SNP

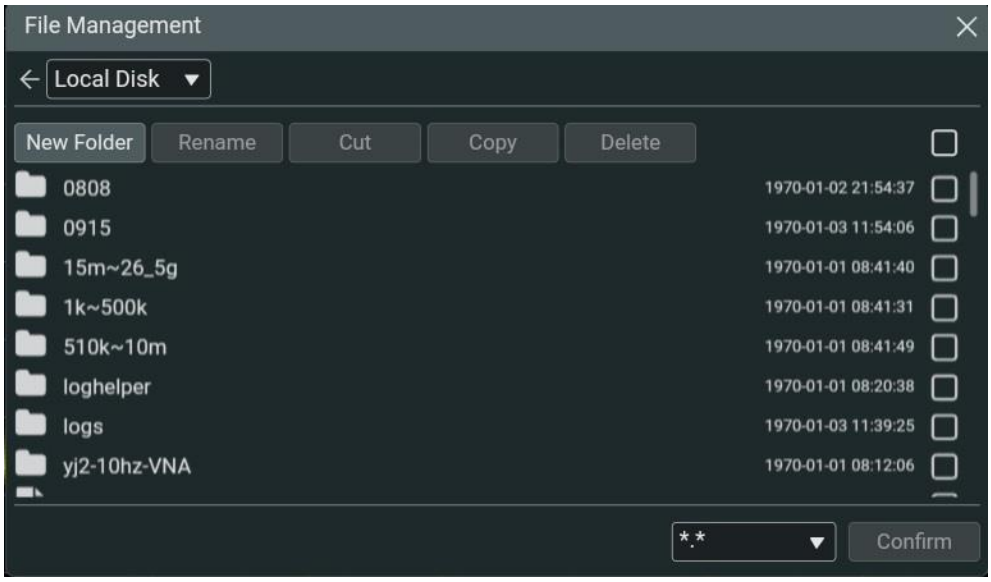
Click **Recall SNP**, then the file management interface is displayed. Select the specified "*.snp" file from the specified path, click **Confirm** to complete the recalling operation.

19.7 File

File Management

Click **Manage Files**, then the file management interface is displayed.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



You can perform the following operations in the file management interface.

- **Select the Disk**

  Before using the external storage device, please ensure that the USB storage device (FAT32 format, flash memory) is properly connected. By default, contents in the "Local Disk" are displayed in the file management interface. If an external storage device is inserted, "Removable USB Disk" is added to the available disks. Select "Removable USB Disk", then the contents of the USB Disk (D) are displayedin the file management interface.

- **Create a Folder**

  Click **New Folder**, then the virtual keypad is displayed, input the folder name to create a new folder.

- **Enter or Exit the Folder**

  Double-click the specified folder to open the folder. The contents in the folder are displayed. The current path can be seen in the disk selection area. Click **<-** to exit the current folder and go back to the previous level of folder.

- **Select a File or Folder**

  Before operating on a file or folder, first select the file or folder. Check the checkbox of the specified file or folder to select it. You can select multiple files or folders.

- **Rename a File or Folder**

  Select the specified file or folder, click **Rename**, then the virtual keypad is displayed. Input the filename or folder name. Click **OK** to complete the input.

- **Delete a File or Folder**

  Select a file or folder to be deleted. Click **Delete**, then a prompt message that prompts you to confirm whether to delete the file or folder is displayed. Click **Confirm** to confirm deleting it. Click **Cancel** to cancel deleting operation.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Copy, Cut, and Paste a File or Folder:** Select a file or folder, then click the specified button such as cut, copy, or paste to perform the specified operation.

Safe Clear

Click **Safe Clear**, then a prompt message "Are you sure to execute secure memory wipe?" is displayed. Click Confirm to clear the internal memory safely. Click **Cancel** to cancel safe clear operation.

Upgrade

This instrument supports local upgrade.

1. Save the upgrade file suffixed with "*.GEL" to the specified path of the instrument.
2. Click **Upgrade**, then the file management interface is displayed.
3. Select the upgrade file from the destination path. Then click **Confirm** to start local upgrade.

20 Save & Recall

This menu allows you to perform quick operation for saving, recalling, and managing files.

Save

The saving operation is the same as that described in **Save** menu under **Setup**. For details, refer to the Save section under Setup.

Recall

The recalling operation is the same as that described in **Recall** menu under **Setup**. For details, refer to the Recall section under Setup.

File

The file management operation is the same as that described in **File** menu under **Setup**. For details, refer to the File section under Setup.

21 System

In the **System** menu, you can set the system-related parameters, I/O settings, etc. To enter the System menu, perform the following operation.

- Click the notification area at the lower-right corner of the screen to enter the **System** menu.
- Press 

to enter the **System** menu.

21.1 I/O Setting

In the **System** menu, click **IO** to enter the I/O interface settings. You can configure the following parameters.

Network Status

Different prompts will be displayed according to the current network connection status.

- Network Config Succeeded!

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- Acquiring IP...
- IP Conflict!
- DISCONNECTED!
- DHCP Config Failed
- Read Status Fail!
- CONNECTED
- Invalid IP
- IP lost
- Please wait...

MAC Address

The MAC address of each instrument is unique. When assigning the IP address for the instrument, the system uses the MAC address to identify the instrument.

VISA Address

Displays the VISA address currently used by the instrument.

IP Configuration Type

The configuration type of the IP address can be DHCP, Auto IP, or Static IP. In different IP configuration types, the configurations for IP address and other network parameters are different.

- **DHCP**

  If "DHCP" is selected, the DHCP server in the current network will assign the network parameters (e.g. IP address, Subnet, Gateway, and DNS) for the instrument.

- **Auto IP**

  When "Auto IP" is selected, the instrument will acquire the IP address ranging from "169.254.0.1" to "169.254.255.254" and the subnet mask (255.255.0.0) automatically based on the current network configuration. The "Auto IP" works only when "DHCP" is not selected or connection is failed.

- **Static IP**

  If "Static IP" is selected, the instrument is configured with static IP. In this case, you need to disable DHCP and Auto IP manually. At this time, you need to set the IP address, Subnet, Gateway, and DNS manually. At this time, you can self-define the network parameters (e.g. IP address) of the instrument.

  - **Set the IP address**

    The format of the IP address is nnn.nnn.nnn.nnn. The range of the first segment (nnn) of the address is from 0 to 255 (except 127); where the valid range is from 0 to 223. The range for the other three segments is from 0 to 255. You are recommended to ask your network administrator for an available IP address.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Set the subnet mask**

  The format of the subnet mask is nnn.nnn.nnn.nnn. Here, the range of "nnn" is from 0 to 255. You are recommended to ask your network administrator for an available subnet mask.

- **Set the default gateway**

  You can set this parameter in Static IP mode. The format of the gateway is nnn.nnn.nnn.nnn. The range of the first segment (nnn) is from 0 to 223 (except 127), and the range for the other three segments is from 0 to 255. You are recommended to ask your network administrator for an available gateway address.

- **Set the DNS address**

  You can set this parameter in Static IP mode. The format of the DNS address is "nnn.nnn.nnn.nnn". The range for the first segment (nnn) of the address is from 0 to 223 (except 127); and the range for the other three segments is from 0 to 255. You are recommended to ask your network administrator for an available address.

  Generally, you do not need to set the DNS, therefore this parameter setting can be ignored.

Tip:

- When the three IP configuration types are all turned on, the priority of the parameter configuration from high to low is "DHCP", "Auto IP", and "Static IP".
- The three IP configuration types cannot be all turned off at the same time.

mDNS

Click on the **ON/OFF** tab for mDNS to enable or disable the multicast Domain Name System (mDNS). This system is used to provide the function of DNS server for service discovery in a small network without a DNS server.

Host Name

The host name can be up to 26 characters.

GPIB[6]

When controlling the instrument via the GPIB, first use the USB-GPIB module to extend a GPIB interface, then use the GPIB cable to connect it to the PC. Configure the GPIB address. Its settable range is from 1 to 30. By default, it is 1.

Apply the Network Parameter Setting

Click **Apply** to validate the current network parameter setting. Click **Reset** to restore to the default settings.

The interface settings are saved to the non-volatile memory. Each time the instrument is restarted, it will load the last interface settings automatically.

6 GPIB interface is supported by the DNA6000 and DNA6000-R series.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

21.2 Basic Settings

In the **System** menu, click **Setting** to enter the basic setting menu.

Language

This product supports menus in multiple languages. The help information, prompt messages, and interface can be displayed in multiple languages. Click the drop-down button of **Language** to select the specified system language from the drop-down list.

Power Switch

- OFF: After the instrument is connected to power, you need to press the Power key on the front panel to power on the instrument.
- ON: After the instrument is connected to power, it will be powered on automatically.

Beeper

Click the ON/OFF tab for **Beeper** to enable or disable the beeper. When the beeper is enabled, you can hear the sound of the beeper when you perform the following operations:

- Operate on the touch screen
- When a prompt message is displayed

Screen Brightness

Drag the slide to set the screen brightness. Its range is from 0% to 100%.

Show Time

Click the ON/OFF tab for the **Show Time** menu to enable or disable the display of the system time. When enabled, the system time (date and time) is displayed in the Notification Area at the lower-right corner of the screen. The date is displayed in "yyyy/mm/dd" format, and the time is displayed in "hh:mm:ss" format. You can enable or disable the display of system time when saving the waveform. When enabled, the saved file will contain the system time information.

- **Date:** Click the "Date" area, then the date setting interface is displayed. Drag the year, month, and day section up and down respectively to set the date. Click **Confirm** to confirm the date modification. Click the close window icon  to cancel the date modification and exit the menu. You can also click any place other than the date setting interface to exit the date modification menu.
- **Time:** Click the "Time" area, then the time setting interface is displayed. Drag the hour and minute section up and down respectively to set the time. Click **Confirm** to confirm the time modification. Click the close window icon  to cancel the time modification and exit the menu. You can also click any place other than the time setting interface to exit the time modification.

21.3 About this Instrument

In the **System** menu, click **About** to view the model of the instrument, its serial number, firmware version, etc.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

- **Model:** indicates the product model.
- **Serial number:** indicates the serial number of the instrument. It is the unique identity of the instrument.
- **Firmware:** indicates the firmware version of the instrument.
- **Hardware:** indicates the hardware version of the instrument.
- **Build:** indicates the build time of the software version of the instrument.
- **Android Build:** indicates the build time of the Android operating system.
- **Android Version:** indicates the version number of the Android operating system. For example, 7.1.0.
- **Launcher:** indicates the GUI version number of the Android operating system.
- **WebControl:** indicates the version number of the web control module.

Upgrade

This instrument supports local upgrade.

1. Save the upgrade file suffixed with "*.GEL" to the specified path of the instrument.
2. In the **System** menu, click **About** > **Upgrade**, then the file management interface is displayed.
3. Select the upgrade file from the destination path. Then click **Confirm** to start local upgrade.

21.4 Options

In the **System** menu, click **Options** to view all the options. For the procedures of installing the option, refer to descriptions in To View the Option and the Option Installation.

21.5 SMB

The SMB menu supports configuring the connection between the instrument and the computer to enable file sharing.

Connect State:

Connection state between the instrument and the PC where the shared directory is located

Boot Auto_Connect:

You may also enable Auto Login with Saved Password, so the instrument will automatically reconnect using the preconfigured server path, username and password after powering on next time.

Drive Letter:

The drive letter assigned to the SMB shared folder in the instrument's internal storage is I:. You may enter this directory via the File Management menu.

Server Path:

The server path format is \\xxx.xxx.xxx.xxx\name, where "xxx.xxx.xxx.xxx" is the IP address and "name" is the name of the shared folder. For example, the server path can be \\172.16.25.77\Share.

Username/Password:

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

You can press Windows key + R to open the Run window. Enter "netplwiz" and click OK to check the user account. The user name and the password are case sensitive.

Operation Procedures

Take Windows 10 Pro as an example. Follow the steps below to use SMB to share files.

1. Enable the SMB file sharing function in the computer.
   a) Open Control Panel > Programs and Features > Turn Windows features on or off.
   b) Locate SMB 1.0/CIFS File Sharing Support in the pop-up window, check all its sub-items, then click OK.
   c) Restart the computer to finish enabling SMB file sharing.
2. Create a shared directory
   a) Create a new folder (the file path must only contain English characters).
   b) Right-click the folder, select Properties > Share > Advanced Sharing > Permissions. You may specify a particular user or the Everyone group, click Add, and then assign the corresponding permission level.
   c) Click Share > Done to complete the creation of the shared directory.
3. Configure the instrument
   a) Access the SMB configuration menu, then set the server path, username and password.
   b) Click the **Connect** button. The setup is finished once the connection status shows Connected.
4. File Sharing

   After the SMB server is connected, you can access and manage the contents inside the shared folder through the disk management system on both the computer and the instrument.

22 Remote Control

This instrument supports Web Control remote operation. Web Control is Web-based remote control operation. With Web control, you can access and operate the LAN-connected instrument via the web page on any smart terminals such as PC, mobile, and iPad, without needing to install any software. The operation procedures are as follows:

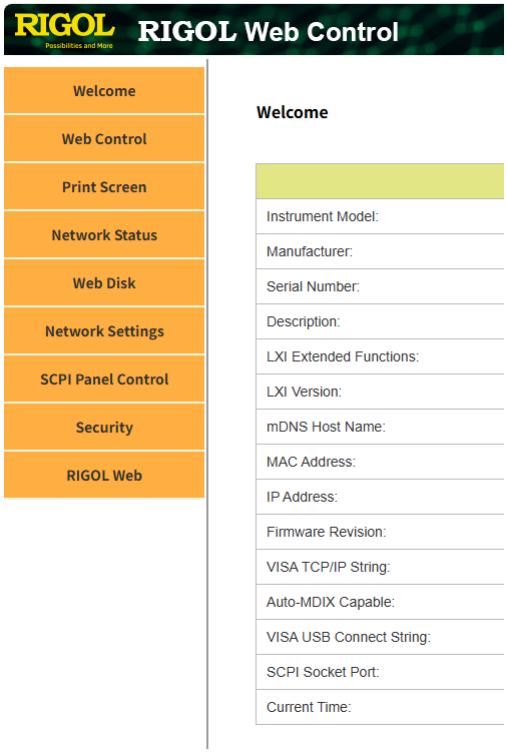
1. Connect the instrument to the network

   Ensure that the rear-panel LAN interface is connected to the network. Note that the instrument must be connected to the network where the control terminal is located. Then you can operate the instrument in remote way by accessing the network.
2. Obtain the IP address

   In the System menu, click or tap IO to view the IP address of the instrument.

   Input the IP address of the instrument into the browser address bar, then press Enter to access the web page, as shown in the following figure.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.



3. Click Web Control on the left side of the screen to enter the instrument remote control interface. You can use the mouse to remotely control the instrument in real time, with the same effect as operating the instrument directly.
4. Click Print Screen, and you can select "Take Screenshot" or "Record Screen" to capture the current screen shot.
5. Click Web Disk to manage files stored in the instrument's internal memory, and upload or download files via the network.
6. Click Network Settings to configure the network. Note that login is required when changing the network configuration. When you first log in to the Web Control, use the user name "admin" and the password "rigol".
7. The SCPI Panel Control function allows the user to send SCPI commands through the web interface for remote programming control of the instrument. Click SCPI Panel Control to enter the commands into the SCPI Command input field. After inputting the commands, click the Send&Read button to send the command and read the returned value.

   You can program and control the instrument by using the SCPI (Standard Commands for Programmable Instruments) commands. For details about the SCPI commands and programming, refer to *Programming Guide* of this series of product.
8. Close the browser to exit the instrument remote control interface.

Only one user can access the instrument IP address for remote control operation at a time. First come, first served. Concurrent logins are not allowed. If the connection is interrupted, you can refresh the browser to load the page.

23 Preset

Press Preset on the front panel to recall the preset settings and restore the system settings of the analyzer to factory defaults.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Trace	
Selected Trace	Trace 1
Trace Title	Off
Trace Hold	Off
Channel	
Selected Channel	Channel 1
Selected Window	Window 1
Window Title	Off
Selected Sheet	Sheet 1
Sheet Title	Sheet 1
Frequency	
Start Freq	10 MHz
Stop Freq	Related to the instrument model
Center Freq	Related to the instrument model
Span	Related to the instrument model
Sweep	
Sweep Points	201
Sweep Type	Linear Frequency
Delay Time	0 s
Auto Sweep State	Auto
Auto Dwell Time	0 s
Measure	
S-Parameter	S11
Parameter Conversion	Off
Data Format	Log Magnitude
Marker	
Selected Marker	Marker 1
Delta Marker	Off
Discrete	Off

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Marker Table	Off
Marker Type	Normal
Show Marker Readout	On
Readouts Per Trace	5
Symbol	Triangle
Coupling Mode	Channel Coupled
Search	
Search Domain	Full Span
Search Tracking	Off
Peak	
Peak Threshold	-100 dB
Peak Excursion	3 dB
Peak Polarity	Positive
Target	
Target Value	0 dB
Target Transition	Both
Multi Peak & Target	
Multi Peak Threshold	-100 dB
Multi Peak Excursion	3 dB
Multi Peak Polarity	Positive
Multi Target Value	0 dB
Multi Target Transition	Both
Power	
State	On
Signal Output Port	Port1
Port Power	-5 dBm
Avg BW	
IF BW	100 kHz
Averaging	

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Averaging Enable	Off
Avg Number	1
Average Type	Sweep
Smoothing	
Smoothing Enable	Off
Smoothing Aperture	1%
Smoothing Points	3
Group Delay Aperture	
Percent of Span	5%
Number of Points	11
Frequency	424.5 MHz
Trigger	
Trigger Mode	Continuous
Trigger Source	Internal
Trigger Scope	Channel
Input	
Global Trigger Delay	0 s
Trigger Level/Edge	High Level
Output	
Enable	Off
Polarity	Negative pulse
Position	After Acquisition
Per Point	Off
Pulse Duration	1 μs
Scale	
Scale	10 dB
Reference Level	0 dB
Reference Position	5
Electrical Delay	

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Electrical Delay Time	0 s
Electrical Delay Distance	0 m
Distance Unit	Meter
Velocity Factor	0.66
Constant	
System Impedance	50 Ω
Phase Offset	0°
Magnitude Offset	0 dB
Magnitude Offset Slope	0 dB/GHz
Math Operation	
Data Math	Off
Display	Data Trace
Parameter Conversion	Off
Statistics	Off
Limit Test	
Enable	Off
Limit Line	Off
Limit Table	Off
Global Pass/Fail	Off
Global Pass/Fail Policy	All Tests
DTF	
DTF State	Off
Distance Unit	Meter
Stop Distance	6.72 m
Cable Loss	0
Velocity Factor	0.66
Window Function	Rectangular
Time Domain	
Transform State	Off

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Start Time	-10 ns
Stop Time	10 ns
Center Time	0 s
Span Time	20 ns
Transform Type	Band Pass
Time Gating	
Gating State	Off
Start Time	-10 ns
Stop Time	10 ns
Center Time	0 s
Span Time	20 ns
Gate Type	Band Pass
Gate Shape	Normal
Time Domain Setup	
Coupling State	On
Marker Mode	Auto
Unit	Meter
Velocity Factor	0.66
Kaiser Beta	6.00
Impulse Width	227.701 ps
Calibration	
Correction	Off
Interpolation	Off
Port Extension	
Enable	Off
Delay	0 s
Distance	0 m
Distance Unit	Meter
Velocity Factor	0.66

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

Parameter Name	Parameter Value
Fixture	
Apply Fixture	Off
Port Match	Off
Port Z	Off
2-port DeEmbed	Off
Save/Recall	
Path	C:/data/UserData
System	
Language	English

24 Troubleshooting

1. **When I power on the instrument, the instrument stays black and does not display anything.**
    a. Check whether the power is correctly connected.
    b. Check the power key on the front panel, and ensure that LED light of at least one test connector is illuminated in green. This proves that the power key is enabled and works normally.
    c. Check whether the fuse is blown. If you need to replace the fuse, use only the specified fuse that conforms to the product.
    d. Restart the instrument after finishing the above inspections.
    e. If the problem still persists, contact RIGOL.
2. **The USB storage device cannot be recognized.**
    a. Check whether the USB storage device can work normally when connected to other instruments or PC.
    b. Make sure that the USB storage device is FAT32 format and flash type. The instrument doesn't support hardware USB storage devices.
    c. After restarting the instrument, insert the USB storage device again to check whether it can work normally.
    d. If the problem still persists, contact RIGOL.
3. **The touch functions cannot be used normally.**
    a. Check whether you have locked the touch screen. If yes, unlock the touch screen.
    b. Check whether the screen or your finger is stained with oil or sweat. If yes, please clean the screen or dry your hands.
    c. Check whether there is a strong magnetic field around the instrument. If the instrument is close to the strong magnetic field (e.g. a magnet), please move the instrument away from the magnet field.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

d. If the problem still persists, contact RIGOL.

4. **The user interface has not refreshed for a long time.**

   a. Check whether the screen is locked or not. If yes, press Esc to unlock it.

   b. Check whether all the trigger conditions have been met and whether there is a valid trigger signal.

   c. Check whether the analyzer is in Stop state or in single trigger state.

5. **The performance verification test failed.**

   a. Check whether the instrument is within the calibration period.

   b. Ensure that you have warmed up the instrument for at least 60 minutes before the test.

   c. Check whether the instrument is at the specified temperature.

   d. Check whether the test is in a strong magnetic environment.

   e. Check whether the power supplies of the instrument and the test system have strong interference.

   f. Check whether the performance of the test device used meets the requirement.

   g. Make sure that the test device used is within the calibration period.

   h. Check whether the test devices used meet the required conditions of the manual.

   i. Check whether all the connections are tight.

   j. Check whether the instrument has connected other devices, cables, and connectors. If yes, check if they are properly connected and work normally.

   k. Make sure that the operations conform to settings and processes specified in the performance verification manual.

   l. If the instrument does not work well, press Preset to restore the system to its preset settings.

   m. Check whether the error calculation has faults.

   n. Correctly understand the definitions of "Typical" and "Nominal" values for this product.

      • Typical (typ.): typical performance, which 80 percent of the measurement results meet at room temperature (approximately 25℃). The data are not warranted and do not include the measurement uncertainty.

      • Nominal (nom.): the expected mean or average performance or a designed attribute (such as the 50Ω connector). The data are not warranted and are measured at room temperature (approximately 25℃).

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

25 Appendix

25.1 Appendix A: Options and Accessories

Accessories and Options	Description	Order No.
Standard Accessory	Power Cord	-
Measurement Application Option	TDA (Time-Domain Analysis)	DNA-TDA10
	DTF (Distance to Fault)	DNA-DTF10
	TAS VNA Automation Test Software	DNA-TAS10[7]
	TDR Enhanced Time-Domain Reflectometry	DNA-TDR10[7]
Optional Accessories	Electronic Calibration Module	Please refer to "Electronic Calibration Module" manual
	Mechanical calibration kit	Refer to Mechanical Calibration Kits Datasheet
	RF Cable	Refer to RF-Cable Datasheet
	Rack Mount Kit	RM6051

Note:

For all the mainframes, accessories, and options, please contact the local office of RIGOL.

25.2 Appendix B: Warranty

RIGOL TECHNOLOGIES CO., LTD. (hereinafter referred to as RIGOL) warrants that the product mainframe and product accessories will be free from defects in materials and workmanship within the warranty period. If a product proves defective within the warranty period, RIGOL guarantees free replacement or repair for the defective product.

To get repair service, please contact your nearest RIGOL sales or service office.

There is no other warranty, expressed or implied, except such as is expressly set forth herein or other applicable warranty card. There is no implied warranty of merchantability or fitness for a particular purpose. Under no circumstances shall RIGOL be liable for any consequential, indirect, ensuing, or special damages for any breach of warranty in any case.

7 This option is supported only on the DNA6000, DNA6000-R, and DNA5000 series products.

Copyright ©RIGOL TECHNOLOGIES CO., LTD. All rights reserved.

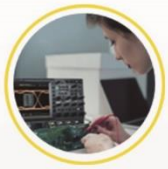
Boost Smart World and Technology Innovation

Industrial Intelligent Manufacturing

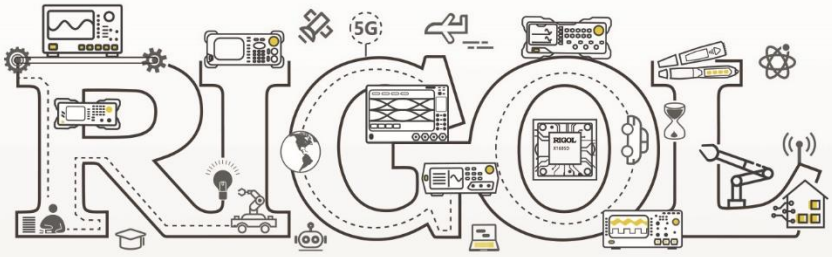




Semiconductors



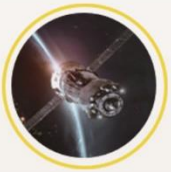
Education& Research





Communication

System Integration





New Energy

- Cellular-5G/WIFI
- UWB/RFID/ ZIGBEE
- Digital Bus/Ethernet
- Optical Communication

- Digital/Analog/RF Chip
- Memory and MCU Chip
- Third-Generation Semiconductor
- Solar Photovoltaic Cells

- New Energy Automobile
- PV/Inverter
- Power Test
- Automotive Electronics

Provide Testing and Measuring Products and Solutions for Industry Customers

HEADQUARTER

RIGOL TECHNOLOGIES CO., LTD.
No.8 Keling Road, New District,
Suzhou,JiangSu, P.R.China
Tel: +86-400620002
Email: info-cn@rigol.com

EUROPE

RIGOL TECHNOLOGIES EU GmbH
Friedrichshafener Str. 5c
82205 Gilching
Germany
Tel: +49-(0)8105-27292-22
Email: johannes.kroiher@rigol.com

NORTH AMERICA

RIGOL TECHNOLOGIES, USA INC.
10220 SW Nimbus Ave.
Suite K-7
Portland, OR 97223
Tel: +1-877-4-**RIGOL**-1
Email: sales@rigol.com

JAPAN

RIGOL JAPAN CO., LTD.
5F,3-45-6,Minamiotsuka, Toshima-Ku,
Tokyo,170-0005,Japan
Tel: +81-3-6262-8932
Fax: +81-3-6262-8933
Email: info.jp@rigol.com

KOREA

RIGOL KOREA CO,. LTD.
5F, 222, Gonghang-daero,
Gangseo-gu, Seoul, Republic of Korea
Tel: +82-2-6953-4466
Fax: +82-2-6953-4422
Email: info.kr@rigol.com

For Assistance in Other Countries

Email: info.int@rigol.com

RIGOL® is the trademark of **RIGOL** TECHNOLOGIES CO., LTD. Product information in this document is subject to update without notice. For the latest information about **RIGOL**'s products, applications and services, please contact local **RIGOL** channel partners or access **RIGOL** official website: **www.rigol.com**