



3657 Series Vector Network Analyzer User's Manual



Ceyear Technologies Co., Ltd.

This manual applies to the following models of vector network analyzer:

- 3657 Series Vector Network Analyzers:
 - 3657A/B Vector Network Analyzers
 - 3657AM/BM Vector Network Analyzers
 - 3657AS/BS Vector Network Analyzers

Options other than standard accessories are as follows:

- English options
- 2-port 9kHz low frequency expansion
- Four-port measurement
- 4-port 9kHz low frequency expansion
- AFR
- Advanced time domain measurement

Version: A.1 March 2023, Ceyear Technologies Co., Ltd.
Address: No. 98, Xiangjiang Road, Huangdao District, Qingdao, Shandong Province
Free customer service number: 800-868-7041
Phone number: 0532-86889847
Fax: 0532-86889056
Website: www.ceyear.com
Email: techbb@ceyear.com

Foreword

Thank you for choosing and using 3657 series vector network analyzers developed and produced by Ceyear Technologies Co., Ltd.! Please read carefully this manual before use.

We will take the responsibility to maximally meet your needs and provide you with high-quality measuring instruments and first-class after-sales service. With the consistent tenet of "high quality and customer satisfaction", we are committed to providing you with satisfactory products and services.

Manual No.

YQ2.733.1253SSCN

Version

A.1 2023.3

Ceyear Technologies Co., Ltd.

Manual Authorization

The contents of this manual are subject to change without notice. The contents and terms used in this manual are interpreted by Ceyear Technologies Co., Ltd.

The copyright of the manual belongs to Ceyear Technologies Co., Ltd, no modification can be made to the manual contents by any unit or person without approval of the Institute, and no reproduction or propagation of the manual can be made for profits, otherwise, Ceyear Technologies Co., Ltd reserves the right of pursuing legal responsibilities from any infringer.

Product Warranty

The warranty period of this product is 18 months from the date of shipment. The instrument manufacturer will repair or replace the damaged components as required within the warranty period. For this purpose, the user shall return the product to the manufacturer and prepay the mailing fee. The manufacturer will return the fee to the user together with the product after maintenance.

Product Quality

Certificate

This product is guaranteed to meet the specifications in this manual from the date of shipment. The calibration and measurement are completed by measuring bodies with national qualification, with relevant data to be provided for reference by users.

Quality/Environmental Management

This product complies with the quality and environmental management systems during R&D, manufacturing and testing. Ceyear Technologies Co., Ltd. is qualified and has passed ISO 9001 and ISO 14001 management systems.

Safety Precautions



The symbol "Warning" indicates a hazard. It reminds the user to pay attention to a certain operation process, operation method or the like. In case of any failure of observing the rule or maloperation, personal injury can occur. Proceed to the next step only after fully understanding and meeting the warning conditions indicated.

Notice

Notice mark, indicating some important information which will not cause danger. It reminds the user to pay attention to a certain operation process, operation method or the like. Failure to observe the rules or operate correctly may cause damage to the instrument or loss of important data. Proceed to the next step only after fully understanding and meeting the note conditions indicated.

Table of Contents

1. Manual Navigation.....	11
1.1. About the Manual.....	11
1.2 Related Documents	12
2 Overview	15
2.1 Product Overview	15
2.2 Safe Operation Guide	20
2.2.1 Safety Marks	20
2.2.2 Operation Status and Locations	22
2.2.3 Electrical Safety	23
2.2.4 Operation Precautions.....	24
2.2.5 Maintenance.....	25
2.2.6 Batteries and Power Modules.....	25
2.2.7 Transportation	26
2.2.8 Waste Disposal/Environmental Protection	26
3 Quick Start	29
3.1 Get Prepared	29
3.1.1 Preparations Before Operation.....	29
3.1.2 System Recovery and Installation Procedures for the Analyzer.....	42
3.1.3 Routine Maintenance	46
3.2 Description of Front and Rear Panels	47
3.2.1 Front Panel Description.....	47
3.2.2 Rear Panel Description	51
3.3 Analyzer Software Interface.....	54
3.4 Analyzer Traces, Channels and Windows	55
3.4.1 Trace.....	55
3.4.2 Channel.....	56
3.4.3 Window	56
3.5 Analyze Data	57
3.5.1 Cursor	57
3.5.2 Trace Operation and Statistics	66

Table of Contents	
3.5.3 Limit Test	69
3.5.4 Ripple Test.....	72
3.5.5 Bandwidth Test	74
3.5.6 Formula Editor.....	77
3.6 Data Output	84
3.6.1 Save and Callback Files	84
3.6.2 Printer displaying measurement results.....	90
4 Measurement Settings	93
4.1 Reset Analyzer	93
4.1.1 Default Resetting Status.....	93
4.1.2 User Define State	96
4.1.3 Reset Analyzer	97
4.2 Measurement Parameter Selection.....	97
4.2.1 S-parameter	98
4.2.2 Differential Balance S-parameter.....	100
4.2.3 Arbitrary Ratio	104
4.2.4 Non-ratio Power Measurement.....	104
4.2.5 Change Trace Measurement Type	105
4.3. Set Frequency Range	106
4.4 Set the Power Level of Signal.....	106
4.5 Set Sweep.....	109
4.5.1 Overview of Sweep Type.....	109
4.5.2 Sweep Type Settings	110
4.5.3 Sweep Time	113
4.5.4 Sweep Settings	114
4.6 Trigger Mode	115
4.6.1 Simple Trigger Settings	115
4.6.2 Detailed Trigger Settings.....	115
4.6.3 Trigger Dialog Box	116
4.7 Set Data Format and Scale.....	119
4.7.1 Data Format	120
4.7.2 Set Data Format.....	122
4.7.3 Scale	123

4.8 Observe Multiple Traces and Open Multiple Channels	124
4.8.1 Automatically Set to Multi-window Display	124
4.8.2 Manually Add Measurement Traces and Windows	125
4.9. Set the Analyzer Display	125
4.9.1 Status Bar	125
4.9.2 Toolbar	126
4.9.3 List	126
4.9.4 Display Contents	127
5 Menus	129
5.1 Menu Structure	129
5.1.1 Meas	130
5.1.2 Trace	131
5.1.3 Chan	132
5.1.4 Freq	133
5.1.5 Power	134
5.1.6 Sweep	135
5.1.7 Trigger	136
5.1.8 Cal	137
5.1.9 Mrk	138
5.1.10 Format	139
5.1.11 Scale	140
5.1.12 Average	141
5.1.13 Display	142
5.1.14 Analysis	146
5.1.15 System	149
5.2 Menu Description	152
5.2.1 Meas	152
5.2.2 Trace	154
5.2.3 Channel	155
5.2.4 Frequency	157
5.2.5 Power	159
5.2.6 Sweep	161
5.2.7 Trigger	163

Table of Contents

5.2.8 Cal	164
5.2.9 Mrk	169
5.2.10 Format	172
5.2.11 Scale	174
5.2.12 Average	176
5.2.13 Display	179
5.2.14 Analysis	180
5.2.15 System	183
6 Optimization & Measurement	187
6.1 Reduce the Impact of Accessories	187
6.2 Improving Reflection Accuracy of Low-loss Two-port Devices	188
6.3 Add Dynamic Range	190
6.3.1 Increase DUT Input Power	191
6.3.2 Decrease the Receiver Noise Base	191
6.4 Improve Measurement Results of Electrical Length Devices	193
6.5. Improve the Accuracy of Phase Measurement	194
6.5.1 Electrical Delay	195
6.5.2 Port Extension	195
6.5.3 Phase Offset	198
6.5.4 Spacing of Frequency Points	198
6.5.5 Specific Operations	199
6.6. Reduce Trace Noise	201
6.6.1 Sweep Averaging	201
6.6.2 Trace Smoothing	202
6.6.3 IFBW	203
6.7 Increase Data Points	203
6.8. Improve Measurement Stability	205
6.9 Increase the Sweep Speed	206
6.10 Improve Multi-state Measurement Efficiency	210
6.10.1 Improve Measurement Efficiency via Measurement Settings	210
6.10.2 Automatically Change Measurement Setting	212
6.10.3 Quickly Call Measurement	212
6.11 Accelerating Data Transmission	212

6.11.1 Use Single Trigger Mode.....	212
6.11.2 Transmit As Little Data As Possible.....	213
6.11.3 Use Real Number Format.....	213
6.11.4 Use LAN.....	213
6.11.5 Use COM Programs.....	213
6.12 Use Macro	213
6.12.1 New Macro.....	213
6.12.2 Macro Settings Dialog Box	214
7 Calibration	217
7.1 Overview of Calibration.....	217
7.1.1 Definition of Calibration	217
7.1.2 Significance of Calibration	218
7.1.3 Calibration Applications.....	218
7.1.4 Simple Procedures for Calibration	218
7.2 Select Calibration Type.....	219
7.3 Calibration Wizard	222
7.3.1 Run Calibration Wizard	222
7.3.2 Perform Calibration Settings.....	223
7.3.3 Modify Calibration Method.....	225
7.3.4 Perform Source Calibration Settings	227
7.3.5 Measurement Standard.....	228
7.3.6 Finish Calibration	230
7.3.6 Save as User Calibration Set	230
7.4 Calibration Set	231
7.4.1 Definition of Calibration Set.....	232
7.4.2 Display Calibration Set Properties.....	233
7.4.3 Save Calibration Set As	234
7.4.4 Delete Calibration Set	234
7.4.5 Delete All Calibration Sets.....	234
7.4.6 Apply/Unapply Calibration Set.....	234
7.5 Measurement Error.....	236
7.5.1 Drift Error	236
7.5.2 Random Error.....	236

Table of Contents

7.5.3 System Error	237
7.5.4 High-accuracy Measurement Calibration	240
7.6 Edit Calibration Kit	242
7.6.1 Definition of Calibration Kit	243
7.6.2 Self-define Calibration Kit	243
7.6.3 Create Calibration Kit	243
7.6.4 Edit Calibration Kit.....	244
7.6.5 Edit Calibration Kit Dialog Box.....	246
7.6.6 Edit Connector Dialog Box	247
7.6.7 Class Information Dialog Box	248
7.6.8 Add Std Dialog Box	249
7.6.9 Open Circuit Breaker Dialog Box.....	250
7.6.10 Short Circuit Breaker Dialog Box	251
7.6.11 Load Dialog Box	252
7.6.12 THRU/TL Dialog Box.....	253
7.6.13 Databased Standard	254
7.7. Calibration Standard	254
7.8 TRL Calibration.....	256
7.9 Fixture Compensation Calibration.....	258
7.9.1 Fixture Simulation Terminologies	259
7.9.2 Fixture Enable	259
7.9.3 Fixture Processing Sequence.....	260
7.9.4 Port Extension.....	261
7.9.5 2-port De-embedding	265
7.9.6 Matching Circuit Embedding.....	267
7.9.7 Port Impedance Conversion	271
7.9.8 4-port De-embedding/Embedding.....	272
7.9.9 AFR.....	275
7.9.10 Cal Plane Manager.....	280
8 Network Measurement Fundamentals	295
8.1 Reflection Measurement.....	295
8.1.1 Expression of Reflection Measurement	296
8.1.2 Summary of Reflection Measurement Expression	298

8.2 Phase Measurement.....	298
8.2.1 What is Phase Measurement	298
8.2.2 Why Phase Measurement is Made.....	299
8.2.3 Use the Analyzer's Phase Format	300
8.2.4 Type of Phase Measurement	300
8.2.5 Linear Phase Offset and Group Delay	301
8.3 Amplifier Parameter Description	302
8.3.1 Gain	302
8.3.2 Gain Flatness	302
8.3.3 Reserve Isolation	302
8.3.4 Gain Drift Change (Temperature, Bias) with Time	302
8.3.5 Linear Phase Deviation	302
8.3.6 Group Delay	303
8.3.7 Return Loss (VSWR, ρ).....	303
8.3.8 Complex Impedance	303
8.3.9 Gain Compression	303
8.3.10 AM-PM Conversion Coefficient.....	304
8.4 Complex Impedance	305
8.4.1 What is Complex Impedance.....	305
8.4.2 Improve the Accuracy of Impedance Measurement.....	305
8.4.3 Measurement Steps of Complex Impedance	306
8.5 Group Delay.....	307
8.5.1 What is Group Delay	307
8.5.2 Reason for Measuring Group Delay	308
8.5.3 What is Group Delay Aperture.....	309
8.5.4 Improve the Measurement Accuracy of Group Delay	310
8.5.5 Measurement Steps of Group Delay	310
8.6. Absolute Output Power	311
8.6.1 What is Absolute Output Power.....	312
8.6.2 Why do We Measure Absolute Output Power.....	312
8.6.3 Measurement Steps for Absolute Output Power.....	312
8.7 AM-PM Conversion.....	313
8.7.1 What is AM-PM Conversion	313

Table of Contents

8.7.2 Reason for Measuring AM-PM Conversion	313
8.7.3 Factors Related to Measurement Accuracy	314
8.7.4 Measurement Steps for AM-PM Conversion.....	315
8.8 Linear Phase Deviation.....	316
8.8.1 What Is Linear Phase Offset.....	316
8.8.2 What Is Linear Phase Offset.....	316
8.8.3 Reason for Measuring Linear Phase Deviation.....	316
8.8.4 Use Electrical Delay Function.....	316
8.8.5 Factors Related to Measurement Accuracy	317
8.8.6 Measurement Steps For Linear Phase Deviation	317
8.9 Reverse Isolation	318
8.9.1 What is Reverse Isolation.....	318
8.9.2 Reasons for Measuring Reverse Isolation	318
8.9.3 Factors Related to Measurement Accuracy	318
8.9.4 Measurement Steps of Reverse Isolation	319
8.10 Small Signal Gain and Flatness.....	320
8.10.1 What is Gain.....	320
8.10.2 What is Flatness.....	320
8.10.3 Reasons for Measuring Small Signal Gain and Flatness	320
8.10.4 Factors Related to Measurement Accuracy	320
8.10.5 Measurement Steps for Small Signal Gain and Flatness.....	321
9 Remote Control	323
9.1 Remote Control Basics	323
9.1.1 Programmed Control Interface	323
9.1.2 Message	326
9.1.3 SCPI	326
9.1.4 Command Sequence and Synchronization.....	336
9.1.5 Status Reporting System.....	337
9.1.6 Programming Precautions.....	340
9.2 Instrument Programmed Control Port and Configuration	340
9.2.1 LAN.....	340
9.3 Basic VISA Interface Programming	343
9.3.1 VISA Library	344

9.3.2 Initialize and Set Default State	344
9.3.3 Send setting command.....	346
9.3.4 Read the Status of Measuring Instrument	348
9.3.5 Read Frequency Marker.....	349
9.3.6 Reading Trace Data	349
9.3.7 Command Synchronization	350
9.4 I/O library	351
9.4.1 Overview of I/O Library.....	351
9.4.2 Installation and Configuration of I/O Library.....	353
10 Troubleshooting and After-sales Services	355
10.1 Working Principles	355
10.1.1 Response of DUT to the RF signal	355
10.1.2 Analyzer Principles.....	356
10.2 Fault Diagnosis and Troubleshooting	357
10.2.1 System Failure	357
10.2.2 Abnormal Curve Display.....	358
10.2.3 Sweep Failure	358
10.2.4 Display Failure	359
10.2.5 No Response From Front Panel Keys	359
10.2.6 Menu Operation Failure.....	359
10.3 Error Message	359
10.3.1 Local Error	360
10.3.2 Programmed Control Error Information.....	361
10.4 Method to Obtain After-sales Services.....	380
10.4.1 Contact Us	380
10.4.2 Package and Mailing	380
Annex.....	383
Appendix 1 Introduction to Typical Measurements	383
1.1 After 30 Minutes of Preheating, Reset the Analyzer	383
1.2 Set Frequency and Power	383
1.3 Select a Measurement and Create a Trace	384
1.4 Calibration.....	385
1.5 Connect the DUT	386

Table of Contents

1.6 Adjust Scale and Data Analysis.....	386
1.7 Record or Save Data.....	387
Appendix 2 Time domain measurement	388
2.1 Time Domain Measurement Principle.....	388
2.2 Time Domain Measurement Resolution and Range	390
2.3 Windows Filtering.....	395
2.4 Time Gate Filtering.....	397
2.5 Time Domain Measurement Data.....	402
2.6 Bandpass and lowpass time domain modes.....	408
2.7 Setting of Time Domain Conversion Measurement	411
2.8 Time Domain Reflectometer (TDR) Impedance Test.....	413
Annex 3 Advanced Time Domain Analysis	417
3.1 Overview	417
3.2 Measurement Settings	419
3.3 Measurement Process	426
3.4 Eye Diagram and Eye Diagram Template Measurements	430
3.5 Advanced Analysis of Eye Diagram Waveforms.....	436

1. Manual Navigation

This chapter introduces the user's manual functions, chapter structure and main contents of 3657 series vector network analyzer, as well as the instrument-related documents provided to users.

- [About the Manual.....11](#)
- [Related Documents.....12](#)

1.1. About the Manual

This manual introduces the applications, performance indicators, basic working principles, operation, precautions and other information of the 3657 series vector network analyzers made by Ceyear Technologies Co., Ltd., so as to help you master the operation methods and key points of the analyzers as soon as possible. Please read this manual carefully and follow its instructions for correct operation.

Due to tight schedule and limited ability of the author, there might be some inevitable errors or omissions in this guide, so please do not hesitate to give your comments if you find such problems! We apologize for any mistake that may cause inconvenience to you.

This manual contains the following chapters:

- **Overview**

It outlines the features and precautions of 3657 series vector network analyzer, including a product overview and a guide for safe use.

- **Quick Start**

This chapter introduces the preparations before using the 3657 series vector network analyzer, routine maintenance of the system and instrument, front panel overview, rear panel overview, operation interface, analyzer menus, traces, channels and windows and how to analyze and save measurement data.

- **Measurement setting**

This chapter introduces various settings during the use of the vector network analyzer, including resetting analyzer, selecting measurement parameters, setting frequency range, setting signal power level, setting sweep, selecting trigger mode, selecting data format and scale, observing multiple traces and opening multiple channels, and setting analyzer display.

- **Menus**

This part introduce the menu structure and menu items according to the functions to facilitate the users to query for reference.

- **Optimization and Measurement**

This chapter introduces how to optimize the measurement accuracy through

1.2 Related Documents

reasonable settings and adjustment. The methods cover reducing influence of accessories, improving reflection accuracy of low-loss two-port devices, increasing dynamic range, improving measurement results of electrical length devices, improving phase measurement accuracy, reducing trace noise, reducing receiver crosstalk, increasing data points, improving measurement stability, improving sweep speed, improving multi-state measurement efficiency, accelerating data transmission, using macros, etc.

- **Calibration**

It introduces the calibration types and methods of 3657 series vector network analyzer to improve the accuracy of the measurement process.

- **Network Measurement Fundamentals**

This chapter introduces the basic concepts and related theoretical knowledge of advanced network parameter measurement.

- **Remote Control**

The methods for remote control of the instrument are summarized to make users get familiar with remote control quickly. It is divided into four parts: programmed control fundamentals, introducing concepts related to programmed control, software configuration, programmed control ports, SCPI commands, etc; port configuration methods, introducing the connection methods and software configuration methods for program-controlled ports of 3657 series vector network analyzers; basic programming methods for VISA interfaces, giving basic programming examples in the form of text description and sample codes to enable users to quickly master the programming methods; I/O function library, introducing the basic concepts of instrument drivers and the basic installation and configuration instructions of IVI-COM/IVI-C drivers.

- **Troubleshooting and Repair**

This part includes the introduction of the working principles of the instrument, troubleshooting, error message description and repair methods.

- **Technical Indicators and Testing Methods**

It introduces the main technical indicators of 3657 series vector network analyzer and the test method instructions recommended for users.

- **Annex**

This part contains operation instructions of options for 3657 series vector network analyzer.

1.2 Related Documents

Product documentation for 3657 series vector network analyzers includes:

- User's Manual

- Programmed Control Manual
- Quick Start Guide
- Online Help

User's Manual

This manual describes the functions and operation methods of the analyzer in detail, including configuration, measurement, programmed control and maintenance, etc. The purpose is to guide users to fully understand the functional characteristics of the product and master common testing methods of the instrument. Main chapters include:

- Manual Navigation
- Overview
- Quick Start
- Measurement setting
- Menus
- Optimization and Measurement
- Calibration
- Network Measurement Fundamentals
- Remote Control
- Troubleshooting and Repair
- Technical Indicators and Measurement Methods
- Annex

Programmed Control Manual

This manual introduces remote programming basics, SCPI basics, SCPI, programming examples and I/O driver function library in detail. The purpose is to guide users to quickly and comprehensively master the programmed control commands and methods of the instrument. Main chapters include:

- Remote Control
- Programmed Control Commands
- Programming Examples
- Error Description
- Annex

Quick Start Guide

This manual introduces the basic methods for configuration and start-up measurement of the instrument to enable users to quickly understand the characteristics of the instrument, and master the basic settings and basic operation methods. Main chapters include:

- Get Prepared
- Typical applications
- Get Help

Online Help

Online help is integrated in the instrument, providing fast text navigation help to make it convenient for users in local and remote control operation. Both the hard keys on the

1.2 Related Documents

front panel of the instrument or the user interface tool bar offer corresponding shortcut keys to activate this function. Main chapters are identical to those of the User's Manual.

2 Overview

This chapter introduces the main performance characteristics, main applications and main technical indicators of 3657 series vector network analyzers. It also gives introductions on correct operation of the instrument and precautions such as electrical safety.

- [Product Overview.....15](#)
- [Safe Operation Guide.....20](#)

2.1 Product Overview

The 3657 series vector network analyzer is an upgraded economical product developed by Ceyear Technologies Co., Ltd. In terms of hardware, new design concepts and technical solutions are applied, greatly improving the key performance specifications including sweep speed and dynamic range; in terms of software, the platform environment adapted to the Intel i3-6300 processor and based on the Windows operating system is adopted, greatly enhancing the interconnection and availability of the network analyzer.

The 3657 series vector network analyzer supports various calibration methods including frequency response calibration, single port calibration, response isolation calibration, enhanced response calibration, full dual-port calibration and electrical calibration, supports multiple display formats including LOG MAG, Linear MAG, SWR, phase, group delay, Smith chart and polar coordinates, and is designed with a variety of external standard interfaces including USB, LAN and DP, so that it is able to accurately measure the magnitude-frequency characteristics, phase-frequency characteristics and group delay characteristics of microwave network.

3656 series vector network analyzers can quickly and accurately measure the S parameters of the DUT, including amplitude, phase and group delay, featuring efficient and strong error correction capability, which is widely used in such military and civilian fields as components, radar, aerospace, electronic jamming and countermeasures, communications, radio and television.

Product features:

- 12.1-inch high-resolution touch screen;
- Windows OS, and menu languages including Chinese and English;
- Frequency response calibration, single port calibration, response isolation calibration, full dual-port calibration, TRL calibration, electrical calibration and other calibration methods supported;

2.1 Product Overview

- Up to 16 display windows, each of which displays up to 8 trace curves simultaneously; 64 separate measurement channels available to quickly implement complex test scenarios;
- One-button Record/Run selection, greatly simplifying the measurement setting steps and improving work efficiency;
- Multiple display formats such as LOG MAG, Linear MAG, SWR, phase, group delay, Smith chart and polar coordinate supported;
- Display interfaces including USB, LAN, HDMI and DP available;
- Single-source stimulus dual-port vector network analyzer and dual-source stimulus four-port vector network analyzer optional;
- Functions of time domain measurement and advanced time domain measurement.

1) User-friendly user interface, which is simple, intuitive, easy to operate and thus able to improve test efficiency

Touch screen, panel buttons and mouse are provided to effectively guide the user to correctly operate and use this product, and the Windows operating system allows the user to operate fast and intuitively, greatly improving the test efficiency.

The 3657 series vector network analyzer is equipped with a 12.1-inch 1280*800 touch screen (as shown in Figure 2.1), making the operation easier.



Figure 2.1 12.1-inch 1280*800 touch screen

2.1 Product Overview



Figure 2.2 Soft panel interface

2) Multi-window and multi-channel measurement display

The 3657 series vector network analyzer supports up to 64 channels, and can display up to 16 measurement windows and 8 test traces in each window at the same time, so that multiple parameters of the DUT can be measured at the same time without multiple instrument status calling, and the test process is simplified.

2.1 Product Overview

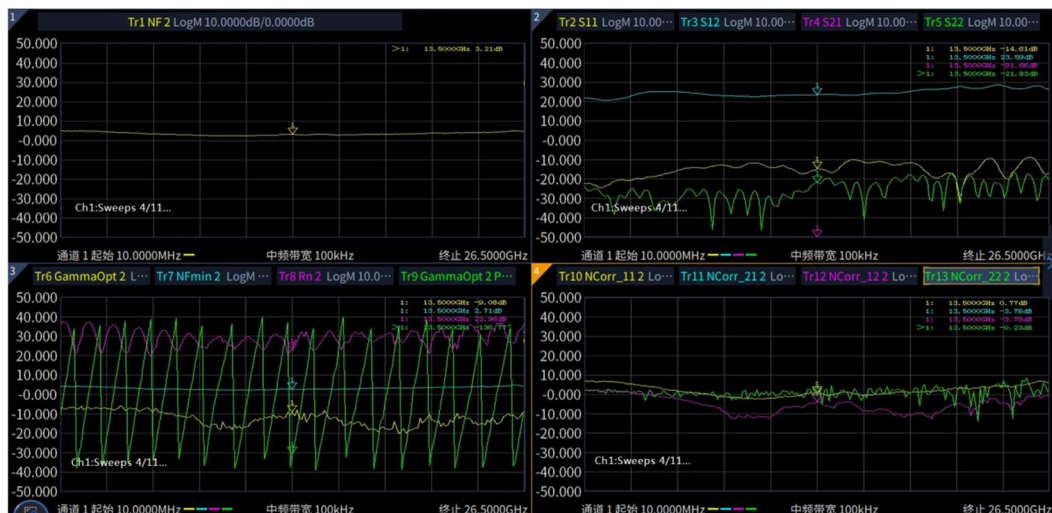


Figure 2.3 Multi-window measurement display

3) Wide dynamic range

The 3657 series vector network analyzer is designed based on the fundamental wave mixing and receiving concept, effectively expanding the test dynamic range of the analyzer and meets the growing demand of users for large dynamic range testing.

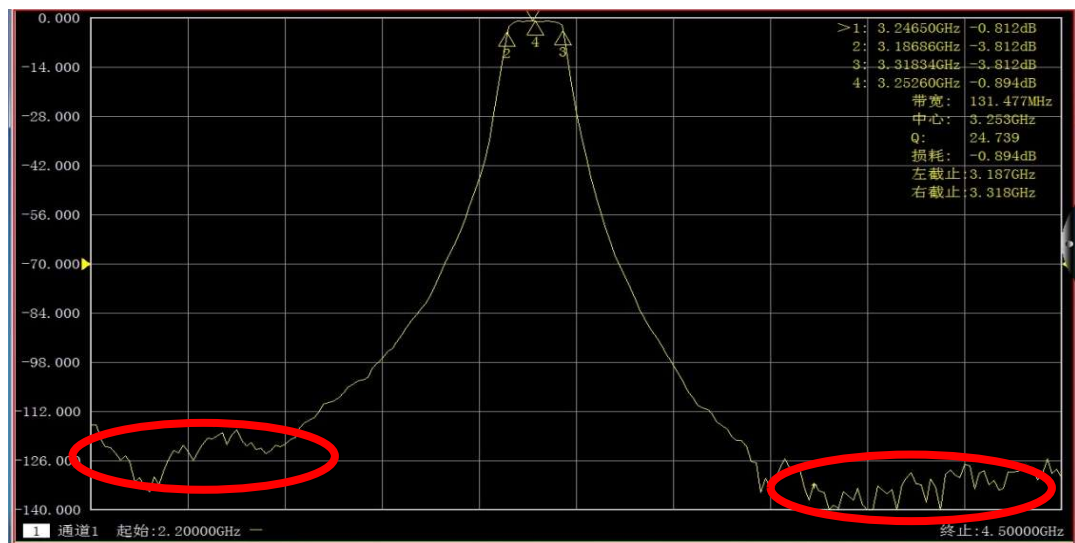


Figure 2.4 Filter measurement results

4) Auto test

Auto test can save a lot of time required for testing, and the use of a flexible automation environment can effectively reduce the cost of testing:

- 1) The SCPI command is used to control the vector network analyzer and complete the automated testing;
- 2) Execute code directly from the vector network analyzer or an external PC via the LAN interface;
- 3) The application can be directly run without the need to connect an external



Figure 2.5 Auto test

5) USB port

The 3657 series vector network analyzer provides six USB interfaces for easy connection to keyboards, mice, printers, electronic calibration kits, and other peripheral devices with USB interfaces.

6) Printing

The 3657 series vector network analyzer provides a powerful printing function that allows the measurement display to be output through a printer or printed to a specified file. The printer may be a local or network printer. It may be a LAN interface or a USB interface printer. Measurement printing can be realized as long as the printer is added in the operating system.

2.2 Safe Operation Guide

Please read carefully and strictly observe the following precautions!

We will spare no effort to ensure that all production processes meet the latest safety standards and provide users with the highest safety guarantee. The design and testing of our products and the auxiliary equipment used meet relevant safety standards, and a quality assurance system has been established to monitor the product quality and ensure the products to always comply with such standards. In order to keep the equipment in good condition and ensure operation safety, please observe the precautions mentioned in this manual. If you have any questions, please feel free to consult us.

In addition, the correct use of this product is also your responsibility. Please read carefully and observe the safety instructions before starting to use this instrument. This product is suitable for use in industrial and laboratory environments or field measurement. Always use the product correctly according to its restrictions to avoid personal injury or property damage. You will be responsible for problems caused by improper use of the product or noncompliance with the requirements, and we will not be held responsible.

Therefore, please always observe the safety instructions to prevent personal injury or property damage caused by dangerous situations. Please keep the basic safety instructions and the product documentation properly and deliver them to end users.

●	Safety Marks.....	20
●	Operation Status and Locations.....	22
●	Electrical Safety.....	23
●	Operation Precautions	2 4
●	Maintenance.....	25
●	Batteries and Power Modules	25
●	Transportation.....	26
●	Waster Disposal/Environmental Protection.....	26

2.2.1 Safety Marks

2.2.1.1 Product-related Marks

Safety marks on the products are described as follows (Table 2.1):

Table 2.1 Products safety marks

Symbol	Meaning	Symbol	Meaning
	Notice, reminding users of information to be paid special attention to. It reminds users of the operation information or		Power ON/OFF

2.2 Safe Operation Guide

	instructions to be paid attention to.		
	Notice, handling heavy equipment.		Standby indication
	Danger! Hazard of electric shock.		DC
	Warning! Hot surface.		AC
	Protective conductive end		DC/AC
	Ground		Reinforced insulation protection of the instrument
	Ground terminal		EU mark for batteries and accumulators. Please refer to Item 1 of "2.2.8 Waste Disposal/Environmental Protection" in this section for specific instructions.

Table 2.1 (Continued)

	Notice, please handle classical sensitive devices with care.	 	EU mark for separate collection of electronic devices. Please refer to Item 2 of "2.2.8 Waste Disposal/Environmental Protection" in this section for specific instructions.
	Warning! Radiation. Please refer to Item 7 of "2.2.4 Operation Precautions" in this section for specific instructions.		

2.2 Safe Operation Guide

2.2.1.2 Manual-related Marks

In order to remind users to operate the instrument safely and pay attention to relevant information, the following safety warning marks are used in the product manual, which are explained as follows:



Danger mark, personal injury or equipment damage may be caused if not avoided.



Warning mark, personal injury or equipment damage may be caused if not avoided.



Caution mark, slight or medium personal injury or equipment damage may be caused if not avoided.



Notice mark, indicating some important information which will not cause danger.



Tips on information about the instrument and its operation.

2.2.2 Operation Status and Locations

Please note before operating the instrument:

- 1) Unless otherwise specified, the 3657 series vector network analyzer should be operated in an environment that allows for smooth placement and indoor operation of the instrument. The maximum altitude for operating the instrument shall not exceed 2000m, and the maximum altitude for transporting the instrument shall not exceed 4500m. The range of actual supply voltage is $\pm 10\%$ of the marked voltage, and the range of supply frequency is $\pm 5\%$ of the marked frequency. The overvoltage level is 2 and the contamination intensity is 2.
- 2) Do not place the instrument on surfaces with water, vehicles, cabinets, tables and other objects that are not fixed and do not meet the load conditions. Please place the instrument securely and fix it on the surface of a solid object (e.g., an ESD workbench).
- 3) Do not place the instrument on the surface of a heat-dissipating object (e.g., a radiator). The operating environment temperature shall not exceed the value specified in the description of relevant indicators of the product. Overheating of the product will lead to electric shock, fire and other risks.

2.2.3 Electrical Safety

Precautions for electrical safety of the instrument:

- 1) The instrument shall be separately supplied and grounded, and shall never share power grid with other high-power apparatus.
- 2) Qualified regulated power supply shall be selected according to the voltage, frequency, power requirements of the analyzer, and shall be reliably grounded.
- 3) Separate and reliable grounding means shall be taken for the analyzer.
- 4) Before the instrument is powered on, the actual supply voltage should match the supply voltage marked on the instrument.
- 5) According to the power requirements of the real panel of the instrument, a three-core power cord should be adopted while ensuring reliable grounding of the ground wire during operation. Either floating ground or poor grounding may cause damage to the instrument and even cause injury to operators.
- 6) Do not damage the power cord, otherwise electric leakage will be caused, resulting in damage to the instrument and even injury of the operators. If an external power cord or extension socket is used, it should be checked before use to ensure electrical safety.
- 7) If the power supply socket does not provide an on/off switch, to cut the power of the instrument, you can just directly unplug the instrument, and therefore, it should be ensured that the power plug can be inserted or drawn conveniently.
- 8) Do not use damaged power cords. Before connecting the instrument to the power cord, check the integrity and safety of the power cord, and properly place the power cord to avoid the impact due to human factors, such as, too long power cord that may trip the operator.
- 9) The TN/TT power supply network is required for the instrument, and the maximum rated current of its fuse is 4A (if a fuse with higher rated current is used, it shall be discussed and determined with the manufacturer).
- 10) Keep the socket clean and tidy, and ensure the plug and the socket in good contact and reliable engagement.
- 11) Neither the socket nor the power cord can be overloaded, otherwise fire or electric shock will be caused.
- 12) If the instrument is tested in a circuit with the voltage $V_{rms} > 30\text{ V}$, sound protections shall be taken to avoid instrument damage (e.g. by using appropriate test instruments, adding fuses, limiting current value, electrical isolation and insulation, etc.).

2.2 Safe Operation Guide

- 13) The instrument shall comply with IEC60950-1/EN60950-1 or IEC61010-1/EN 61010-1 standards to connect with PC or IPC.
- 14) Unless otherwise allowed, do not open the housing of the instrument, which may expose internal circuits and devices of the instrument and cause unnecessary damage.
- 15) If the instrument needs to be fixed at the test site, a qualified electrician is required to install the protective earth wire between the test site and the instrument first.
- 16) Take appropriate overload protections to prevent overload voltage (caused by lightning, for instance) from damaging the instrument or causing personal injury.
- 17) When opening the housing of the instrument, do not place objects not belonging to the interior of the instrument, otherwise, short circuit, damage to the instrument and even personal injury may be caused.
- 18) Unless otherwise stated, the instrument has not received any waterproof treatment, so keep the instrument from contacting with liquid to prevent damage to the instrument or even personal injury.
- 19) Do not place the instrument in an environment where fog is easily formed, for example, moving the instrument in a environment where cold and heat are in alternation, where water droplets formed on the instrument may cause electric shock and other hazards.

2.2.4 Operation Precautions

- 1) Instrument operators need to have certain professional and technical knowledge, good psychological quality, and certain emergency response capabilities.
- 2) Before moving or transporting the instrument, please refer to the relevant instructions in "2.2.7 Transportation" of this section.
- 3) The inevitable use of substances (e.g. nickel) in the production process of the instrument may cause allergy to personnel. If an operator of the instrument has allergic symptoms (e.g. rash, frequent sneezing, ophthalmia or dyspnea) during the operations, please seek medical care in time to find out the reason and solve the symptoms.
- 4) Please refer to the relevant instructions in "2.2.8 Waste Disposal/Environmental Protection" of this section before disassembling this instrument for disposal.
- 5) RF instruments will generate high electromagnetic radiation, during which period, pregnant women and operators with cardiac pacemakers need special protection.

2.2 Safe Operation Guide

If the radiation level is high, corresponding measures may be taken to remove the radiation sources to prevent personal injury.

- 6) In case of fire, the damaged instrument will release toxic substances. Therefore, the operators should wear appropriate protective equipment (e.g. Protective masks and exposure suits) for safety.
- 7) Laser products shall have different warning signs according to the laser category, because the radiation characteristics of laser and such equipment have high-intensity electromagnetic power characteristics, which will cause harm to human body. If the product is integrated with other laser products (e.g. CD/DVD drive), it will not provide other functions except the settings and functions described in the product manual in order to prevent the injury of the laser beam to the human body.
- 8) Electromagnetic compatibility level (in accordance with EN 55011/CISPR 11, EN 55022/CISPR 22 and EN 55032/CISPR 32 standards)
 - Class A equipment:

The equipment can be used except in residential areas and low-voltage power supply environment.

Note: Class A equipment is suitable for industrial operation environment, because it produces wireless communication disturbance in residential areas. Therefore, operators need to take relevant measures to reduce the impact of such disturbance.
 - Class B equipment:

Equipment suitable for residential areas and low-voltage power supply environment.

2.2.5 Maintenance

- 1) Only authorized and specially trained operators are allowed to open the casing of the instrument. Before such operations, it is required to disconnect the power cord to prevent damage to the instrument or even personal injury.
- 2) The repair, replacement and maintenance of the instrument should be performed by dedicated electronic engineers of the manufacturer, and the parts subject to replacement and maintenance should receive safety tests to ensure safe use of the product in the future.

2.2.6 Batteries and Power Modules

Before using batteries and power modules, carefully read the relevant information to avoid explosion, fire and even personal injury. In some cases, disused alkaline batteries (e.g. lithium batteries) shall be disposed of in accordance with **EN 62133** standard.

Precautions for use of batteries include the following:

- 1) Do not damage the battery.

2.2 Safe Operation Guide

- 2) Do not expose batteries and power modules to heat sources such as open fire; avoid direct sunlight and keep them clean and dry; clean the connection port of the battery or power module with a clean and dry soft cotton cloth.
- 3) Do not short circuit the battery or power module. Do not store multiple batteries or power modules in cartons or drawers because the batteries are likely to cause short circuit due to being in contact with each other or other conductors; Do not remove the original outer packaging of the battery and power module before use.
- 4) Batteries and power modules must not be subjected to mechanical impact.
- 5) If the battery fluid leaks, please do not touch the skin and eyes, otherwise wash it with a large amount of water and get medical treatment in time.
- 6) Please use the manufacturer's original batteries and power modules. Any incorrect replacement and charging of alkaline batteries (such as lithium batteries) is likely to cause an explosion.
- 7) Discarded batteries and power modules shall be recycled and disposed of separately from other wastes. Due to the toxic substances inside the battery, they shall be properly discarded or recycled according to local regulations.

2.2.7 Transportation

- 1) If the instrument is heavy, please handle it with care. If necessary, use tools (a crane, for instance) to move the instrument so as to prevent damaging the body.
- 2) The handle of the instrument is suitable for personal handling of the instrument and cannot be fixed on the transportation equipment when during the transportation of the instrument. To prevent property loss and personal injury, please follow the manufacturer's safety regulations on the transportation of the instrument.
- 3) When operating the instrument on the vehicle, the driver should drive carefully to ensure transportation safety, and the manufacturer is not responsible for any emergencies during the transportation. Therefore, please do not use this instrument during the transportation, and reinforcement and preventive measures should be taken to ensure the transportation safety of the product.

2.2.8 Waste Disposal/Environmental Protection

- 1) Do not dispose of devices marked with batteries or accumulators together with unclassified waste; Instead, such devices should be collected separately and disposed of in a suitable collection location or through the customer service center of the manufacturer.
- 2) Do not dispose of waste electronic devices together with unclassified waste; Instead, such devices should be collected separately. The manufacturer has the right and responsibility to help end users dispose of waste products. If necessary, please contact the customer service center of the manufacturer for corresponding disposal so as not to damage the environment.

2.2 Safe Operation Guide

- 3) During mechanical or thermal processing of the product or its internal components, toxic substances (dust of heavy metals, such as lead, beryllium, and nickel, etc.) may be released. Therefore, specially trained technicians with relevant experience are required to disassemble the product to avoid personal injury.
- 4) During the reprocessing, please refer to the safety operation rules recommended by the manufacturer to dispose of toxic substances or fuel released from the product with specific methods to avoid causing personal injury.

3 Quick Start

This chapter introduces the pre-operation precautions, front/back panel browsing, basic measuring method, and data file management of 3657 series vector network analyzers, so that users can have a preliminary understanding of the instrument itself and its measurement processes.

●	Get Prepared.....	29
●	Front and Real Panels Description.....	47
●	Analyzer Interface.....	54
●	Analyzer Traces, Channels and Windows.....	55
●	Data Analysis.....	57
●	Data Output.....	84

3.1 Get Prepared

●	Preparation Before Operation.....	29
●	System recovery and installation procedures for the analyzer.....	42
●	Routine Maintenance	46

3.1.1 Preparations Before Operation

The chapter introduces precautions for 3657 series vector network analyzers before initial settings and operation.



Warning

Avoid damaging the instrument

Pay attention to the follow to avoid electric shock, fire and personal injury:

- Do not open the cabinet arbitrarily;
- Do not try to disassemble or refit any part of the instrument not mentioned in the manual. Otherwise, such consequences as reduced electromagnetic shielding property and internal component damage can occur, affecting product reliability. In this case, we will not provide any free maintenance any more even if the product is still in the warranty period.
- Read carefully relevant contents in “2.2 Safety operation instruction” of the Manual as well as the following precautions on safety operation, and pay attention to the requirement on specific operation environment involved in the data pages.

Notice

Electrostatic Protection

Take electrostatic protection measures at workplaces to avoid any damage caused by the instrument. For details, see relevant contents in “2.2 Safety operation instruction” of the User’ Manual.

Notice

Pay attention to the following when operating the instrument:

Improper operating or measuring position can damage the instrument or the one connected to it. Pay attention to the following before powering on the instrument:

- Keep the fan blade and heat emission hole unblocked, with a distance of at least 10cm away from the wall;
- Keep the instrument dry;
- Keep the instrument level, and arrange it properly;
- Make sure the environment temperature meets with the requirement noted in the data page;
- Make sure the port input signal amplitude is within the range specified;
- Make sure the signal output port is connected properly, without any overload.

Tips

Effect of electromagnetic interference (EMI)

Since EMI can affect the measurement result, pay attention to the following:

- Select proper shield cables. For example, to use the double-shielded RF/network connection cable;
- Please close any cable connection port that is enabled but temporarily unused or connect a matching load to the connection port in time;
- Please refer to the EMC level labels in the Data Page.

●	Unpacking.....	30
●	Environmental Requirements.....	33
●	Turning On/Off the Power.....	35
●	Correct Use of Connectors.....	38

3.1.1.1 Unpacking

1) Visual examination

Step 1: check whether the outer package and the shockproof packing of the instrument are damaged. If there is any damage, keep the outer package for standby and

proceed with the following examination steps.

Step 2: unpack, and check the mainframe and articles provided in the package for any damage;

Step 3: check carefully the articles mentioned above as per Table 3.1 for any problem;

Step 4: in case of any outer package damage, or damage or problem to the instrument or articles provided in the package, never power the instrument on or start it up! Please contact our service consultation center

with the service hotline provided on the cover, and we will repair or change it as soon as possible accordingly.

Notice

Handling: since the instrument and packing box are heavy, it shall be handled by two person at the same time, and placed with care.

2) Model confirmation

Name	Function
Host:	
✧ 3657 series vector network analyzers	—
Standard:	
✧ 3-core power cord	—
✧ USB mouse	—
✧ Packing List	—
✧ Product Certificate of Conformity	—
Options	
✧ H03 English option	For setting language of front and rear panels and operating system to English.
✧ H63 User's Manual (hardcopy)	Providing detailed User Manual of hardcopy version.
✧ S10 Time domain measurement function	For time domain measurements to locate and analyze breakpoints in devices, fixtures or cables.
✧ S11 Advanced time domain analysis function	For TDR impedance test, eye diagram analysis, etc.

Table 3.1 List of articles provided together with 3657

3.1.1.2 Environmental Requirements

The operation sites of 3657 series vector network analyzers should meet the following environmental requirements:

1) Operating environment

The operating environment should meet the following requirements:

Table 3.2 Environmental requirements of 3657

Temperature	0°C ~ 40°C
Temperature range during adjustment error	23°C ±5°C (allowable temperature deviation during error adjustment < 1 ° C)
Humidity	Hygrometer measurement range at <+29 °C: 20% ~ 80% (non-condensed)
Elevation	0 ~ 2,000 m (0 ~ 6,561 ft)
Vibration	Max. 0.21 G, 5 Hz ~ 500 Hz

Notice

The above environmental requirements are only applicable to the operating environment factors of the instrument, and are not with the scope of technical specifications.

2) Heat dissipation requirements

In order to ensure that the working environment temperature of the instrument is within the temperature range required by the operating environment, the following heat dissipation space requirements of the instrument shall be met:

Table 3.3 Heat dissipation requirements of 3657

Instrument part	Heat dissipation distance
Back	≥60 mm
Left and right sides	≥180 mm

3) Electrostatic Protection

Static electricity is extremely destructive to electronic components and equipment. Usually we take two anti-static measures: conductive table mat and wrist strap; Conductive floor mat and ankle strap. Using the above two anti-static measurements at the same time can provide good antistatic protection. If using one of them, only the former can provide antistatic protection. 1MΩ earth isolation resistor must be provided for the antistatic components at least for ensuring user safety.

Correctly take the following antistatic measures to techniques to reduce electrostatic damages:

- Ensure all instruments are grounded properly, so as to avoid any static electricity;
- Let the internal/external conductor of the cable contact the ground shortly before connecting the coaxial cable with the instrument;
- Operators must wear anti-static wrist straps or take other antistatic measures before touching the joints, core or conducting any assembly.

Warning

Voltage range

The above-mentioned anti-static measures cannot be applied when the voltage exceeds 500V.

3.1.1.3 Power ON/OFF

1) Precautions before turning on the power

Pay attention to the following when turning on the power of the instrument:

a) Confirming power supply parameters

3657 series vector network analyzers adopt three-core power cord interfaces, which conform to international safety standards. Before the analyzer is powered up, it must be confirmed that the protective ground of the power supply socket has been reliably grounded before plugging the power cord into a standard three-prong outlet. Poor grounding or floating grounding may damage the instrument and even cause personal injury. Never use the power cord without protective ground. Table 3.4 shows the external power supply requirement for normal operation of VNA.

Table 3.4 Working power parameter requirements of 3657

Power supply parameter	Applications
Voltage, frequency	220V/110V adaptive; 50Hz-60Hz
Rated output current	>3A
Power consumption	150W

Tips

Prevent mutual interference of power supplies

To prevent mutual interference among multiple devices through power supplies, especially spike pulse interference caused by high-power devices which may cause damage to instrument hardware, it is recommended to use a 220V/110V AC stabilized power supply to supply power to the analyzer.

b) Confirm and connect the power cord

3657 series signal generators adopt three-core power cord interfaces, which conform to national safety standards. Before powering on the network analyzer, it is necessary to confirm **reliable grounding of the ground wire of the network analyzer power line**. Either floating ground or poor grounding may cause damage to the analyzer and even cause injury to operators. Using a power cord without protective grounding is strictly prohibited. When the instrument is connected to a suitable power outlet, the power cord connects the housing of the instrument to the ground.

When connecting the instrument to the power supply:

Step 1. Confirm that the working power cord is not damaged;

3.1 Get Prepared

Step 2. Connect the power plug of the rear panel of the instrument to a well-grounded three-core power socket with the power cord.



Grounding

Poor or wrong grounding may cause damage of the instrument or personal injury. Before powering on the network analyzer, make sure that the ground wire is in good contact with the ground wire of the power supply.

Please use a power outlet with grounding protection. Do not use any external cable, power cord or autotransformer without any protective grounding as the protective grounding line. If an auto-transformer is necessary, it is required to connect the common terminal to the protective ground of the power connector.

2) Initial power-up

Precautions for turning on/off the power of the instrument are as follows:

a) Connecting the Power supply

Please confirm the power supply parameters and the power cord before power-on for the first time. For details, please refer to contents on “Precautions before Power-on” in 3.1.1.3 of the User's Manual.

Step 1. Confirm switch state: As shown in Figure 3.1, check whether the power button on the rear panel (the upper part in Figure 3.1) is off. If so, proceed to the next step. If not, press down the side of button marked with "O";

Step 2. Connect the power cord: Connect one end of the power cord matched with the network analyzer in the package or a three-core power cord that meets the requirements to the power socket of the rear panel of the network analyzer (lower part in Figure 3.1). The required voltage parameter of the network analyzer is marked beside the power socket to remind the user that the voltage used should meet the requirement. The other end of the power cord is connected to a compliant AC power source.



Figure 3.1 Power socket (lower) and power button (upper)



Figure 3.2 Front panel power button

b) Turning on/off the power**i. Start up**

Step 1. Turn on the power button on the rear panel (press down the side of button marked with "|"). Then, the button on the front panel (as shown in Figure 3.2) shall turn yellow;

Step 2. Press the power button in the upper right corner of the front panel. At this time, the color of the power indicator above the power button changes from yellow to white.

Step 3. The network analyzer will take approximately 1 minute to boot the operating system, and after performing a series of self-tests and program adjustment, it starts running the main program of measurement.

The instrument is in the operable status.

Tips**Instrument cold-start warm-up**

When the 3657 series vector network analyzer is cold started, the network analyzer should be allowed to warm up for at least 30 minutes or more before measurements in order for the instrument to meet the specified performance specifications.

Tips**Running the analyzer applications**

The network analyzer automatically runs the application when it is powered on. If you exit the application, you can re-run the measurement program by:

Method 1 Click [Start] in the taskbar at the bottom left corner of the screen, point to [Programs] in the Start menu,

point to [VNA] in the Programs submenu, click [VNA] in the pop-up submenu, and the analyzer will start running the measurement application, or you can double-click the shortcut on the desktop to run the program.

Method 2 Press [RST] in the Function keypad and the analyzer starts running the 3657 series vector network analyzer application.

ii. Shutdown

Step 1. Turn the power button in the upper right corner of the front panel (as shown in Figure 3.2). At this time, the instrument enters the shutdown process (the software and hardware need to go through some processing before the power is turned off). After more than ten seconds, the instrument is powered off, when the color of the power indicator above the power button changes from white to yellow;

3.1 Get Prepared

Step 2. Turn off the power button on the rear panel ("O") or disconnect the power supply of the instrument. The instrument is turned off.

Notice

Power cut of the instrument

When the instrument is in normal operation, it can only be shut down by operating the power button on the front panel. **Do not directly operate the power button of the rear panel or directly disconnect the power connection with the instrument.** Otherwise, the instrument cannot enter the normal shutdown state, which may cause damage to instrument or loss of the current instrument status/measurement data. Please shut down the instrument with the correct method. If the analyzer cannot be shut down normally due to the operating system or an application exception, you can shut down the analyzer by pressing and holding the [PWR On/SBY] key for at least 4 seconds.

c) Power cut

In case of emergency, in order to avoid personal injury, the network analyzer needs to be powered off immediately. In this case, just pull up the power cord (from the AC outlet or from the power outlet on the rear panel of the instrument). Therefore, sufficient operating space should be reserved when operating the instrument to facilitate direct shutdown when necessary.

3.1.1.4 Proper Operation of Connectors

Connectors are often used in various tests of network analyzer. Although the connectors of calibration kits, test cables and analyzer measurement ports are designed and manufactured according to the highest standards, the service life of all these connectors is still limited. Due to the inevitable wear and tear during normal use, the performance specifications of the connectors will decrease or even be unable to meet the measurement requirements. Therefore, correct maintenance and measurement connection of the connectors can not only ensure accurate and repeatable measurement results, but also prolong the service life of the connectors and reduce the measurement costs. In actual use, the following aspects should be paid attention to:

1) Connector check

When conducting connector inspection, anti-static wrist band should be worn. It is recommended to use a magnifier to check the following items:

- 1) Whether the electroplated surface is worn or not and whether there are deep scratches;
- 2) Whether the thread is deformed;

3.1 Get Prepared

- 3) Whether there are metal particles on the threads and the joint plane of the connector;
- 4) Whether the inner conductor is bent or broken;
- 5) Whether the screw sleeve of the connector rotates improperly.

Caution
Check the connector to prevent damaging ports of the instrument

Any damaged connector may damage the good connector connected to it even when measuring the connection for the first time. In order to protect each interface of the analyzer itself, the connector must be checked before connector operation.

2) Connector cleaning

When cleaning the connectors, always wear antistatic wrist straps and observe the following steps:

- 1) Remove loose particles on the thread and joint plane of the connectors with clean low-pressure air, and thoroughly inspect the connectors. If further cleaning treatment is required, proceed as follows:
- 2) Soak (but not thoroughly soak) a lint-free cotton swab with isopropyl alcohol;
- 3) Remove the dirt and debris from the joint plane and threads of the connectors with cotton swabs. When cleaning the inner surface of a connector, be careful not to apply external force to the central inner conductor and not to leave the fibers of cotton swabs on the central conductor of the connector.
- 4) Let the alcohol volatilize, then blow the surface clean with compressed air;
- 5) Check the connector to make sure that it is free of particles and residues.
- 6) If any defects of the connector is still obvious after cleaning, it indicates that the connector may have been damaged and should not be used again. Make clear the cause of the connector damage before connection.

3) Connection method

Before the connection, the connectors should be inspected and cleaned to ensure cleanness and intactness. Anti-static wrist straps should be worn before connection. The correct connection method and steps are as follows:

Step 1. As shown in Figure 3.3, align the axes of the two interconnecting devices to ensure that the pin of the male connector slides concentrically into the



socket of the female connector.

Figure 3.3 Axes of interconnected devices in a straight line

3.1 Get Prepared

Step 2: as shown in Figure 3.4, move the two connectors leveled together so that they can be smoothly engaged. Rotate the screw sleeve of the connector (note, not the rotating connector itself) until it is tightened, and there can be no relative rotational movement between the connectors during the connection.

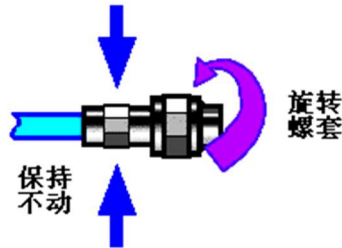


Figure 3.4 Connection method

Step 3: As shown in Figure 3.5, tighten the connectors with a torque wrench to complete the connection. Note that the torque wrench should not exceed the initial turning point, and an auxiliary wrench can be used to prevent the connector from rotating.

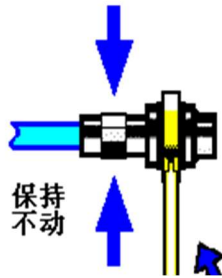


Figure 3.5 Finishing the connection with a torque wrench

4) Disconnection

Step 1: support the connectors to prevent any connector from being twisted, shaken or bent;

Step 2: an open-ended wrench can be used to prevent the connector body from rotating;

Step 3: loosen the screw sleeve of the connector with another wrench;

Step 4. Rotate the screw sleeve of the connector by hand to complete the disconnection;

Step 5. Pull the two connectors levelly apart.

5) Usage of a torque wrench

The use of a torque wrench is shown in Figure 3.6. The following points should be paid attention to when using it:

- Confirm that the torque of the torque wrench is correct set before use;
- Ensure that the angle between the torque wrench and another wrench (used to support a connector or a cable) is within 90° before applying force.

3.1 Get Prepared

- Grasp the end of the torque wrench handle gently, and apply force in the direction perpendicular to the handle until reaching the folding point of the wrench.

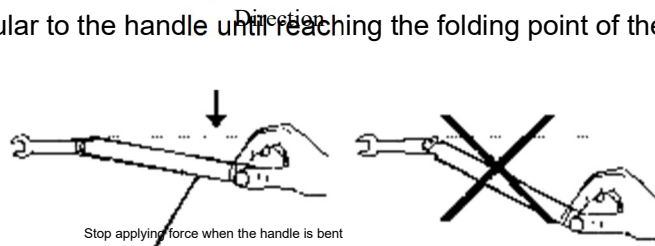


Figure 3.6 Usage of a torque wrench

6) Use and storage of connectors

- 1) The connectors should be covered by protective sleeves when not in use.
- 2) Do not mix various connectors, air lines and standard calibration pieces in a box because this is one of the most common causes of connector damage.
- 3) Keep the connectors and the analyzer at the same temperature. Holding a connector by hand or cleaning a connector with compressed air will significantly change its temperature. The connectors should be calibrated after its temperature is stable.
- 4) Do not touch the joint plane of the connectors because the grease and dust particles on the skin are difficult to be removed from the joint plane;
- 5) Do not put the contact surface of a connector downward on a hard table surface. Contact with any hard surface may damage the electroplated layer and the joint surface of the connector.
- 6) Always wear anti-static wrist straps and work on a grounded conductive workbench pad, which can protect the analyzer and the connectors from electrostatic discharge.

7) Use of adapters

When the measuring port of the analyzer and the connector type used are different, adapters must be used for the connection before measurement. In addition, even if the measuring port of the analyzer and the connector type of port of the DUT are the same, it is also advisable to use adapters. Both cases can protect the measurement port, prolong its service life and reduce the maintenance cost. Before connecting an adapter to the measurement port of an analyzer, it is required to carefully check and clean the adapter. And a high-quality adapter should be used to reduce the influence of mismatching on measurement accuracy.

8) Joint plane of connectors

An important concept in microwave measurement is reference plane. And an analyzer, it is the benchmark reference plane for all measurements. During the calibration, the reference plane is defined as the plane where the measurement port and the

3.1 Get Prepared

calibration standard are engaged. Good connection and calibration depend on thorough and level contact between the connectors on the joint plane.

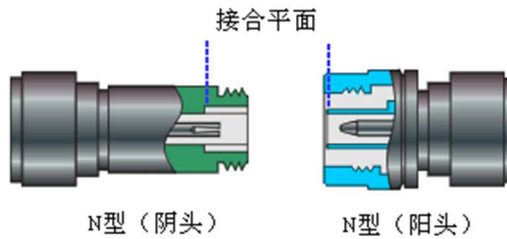


Figure 3.7 Calibration plane

3.1.2 System Recovery and Installation Procedures for the Analyzer

3.1.2.1 System Recovery for the Network Analyzer

The operating system of the analyzer may not work normally due to abnormal shutdown of the analyzer by the user, virus infection or self installation of other software on the analyzer. So it is necessary to restore the analyzer to the default state at the factory. The system recovery steps are as follows:

- 1) Turn off the analyzer and connect the keyboard to the USB interface;
- 2) Press **[PWR On/SBY]** button in the upper right corner of the front panel, and when the indicator on the external keyboard goes on and the interface shown in Figure 3.8 pops up, immediately press the **[↑]/[↓]** button on the keyboard, or directly touch Recovery and then press the **[Enter]** button on the keyboard to automatically restore the system of the analyzer;



Figure 3.8 Startup interface

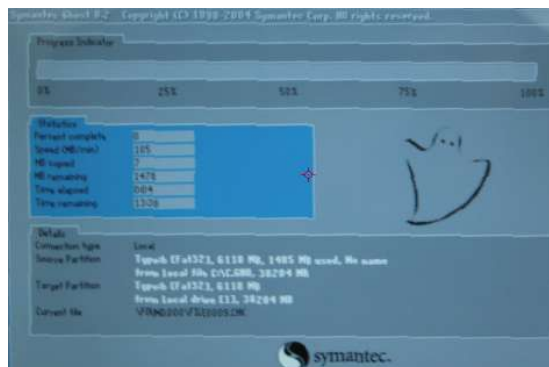


Figure 3.9 System recovery interface

3.1 Get Prepared

- 3) After the system recovery starts, the interface shown in Figure 3.9 pops up, indicating that the system recovery is in progress. The system recovery process takes about 10min, and after the system recovery is completed, the analyzer needs to be restarted.

3.1.2.2 Updating and Installation of Network Analyzer Program

If program update or installation of new program is required for the 3657 series vector network analyzer, double click **D:\ CNA-Windows-x64-XXX.exe**, directly press **[Next]** following the Menu Wizard of **[Ceyear Vector Network Analyzer Installer]**, select **[Complete]** when **[SEL Instl TYP]** appears, and if any other menu appears in this process, directly click **[Cancel]**. Then, double-click the **CNA** icon on the desktop or press **[Reset]** to start the program.

The operation steps are shown in Figure 3.10~3.16.



Figure 3.10 Installer boot interface



Figure 3.11 Installation directory selection interface



Figure 3.12 Installation type selection interface



Figure 3.13 License agreement interface



Figure 3.14 Shortcut creation interface



Figure 3.15 Installation preparation interface

3.1 Get Prepared



Figure 3.16 Installation in-process interface

3.1.3 Routine Maintenance

This section introduces the routine maintenance of the 3657 series vector network analyzers.

3.1.4.1 Cleaning Method

1) Cleaning instrument surface

Please follow the steps below when cleaning the surface of the instrument:

Step 1: Shut down the instrument and disconnect the power cord connected to it;

Step 2: Wipe the surface gently with dry or slightly wet soft cloth, and do not wipe the inside of the instrument.

Step 3: do not use chemical cleaners, such as alcohol, acetone or dilutable cleaners.

2) Cleaning the display

After a period of use, the display needs to be cleaned. Please follow the steps below:

Step 1: Shut down the instrument and disconnect the power cord connected to it;

Step 2: Dip a piece of clean and soft cotton cloth into the cleaner and then gently wipe the display panel;

Step 3: Dry the display with a piece of clean and soft cotton cloth;

Step 4: Connect the power cord only after the cleaner is completely dried.

Notice

Display cleaning

There is an antistatic coating on the surface of the display. Do not use cleaners containing fluoride, acid and alkaline. Do not spray the cleaner directly onto the display panel, otherwise it may penetrate into the instrument and damage the instrument.

3.2 Description of Front and Rear Panels

This section introduces the composition and functions of the front and rear panels as well as operation interface elements of 3657 series vector network analyzers.

- [Front Panel Description](#).....47
- [Real Panel Description](#).....51

3.2.1 Front Panel Description

This section introduces the composition and functions of the front panels of 3657 series vector network analyzers. The front panel is shown in Figure 3.17.



Figure 3.17. Front panel

3.2 Description of Front and Real Panels

1) **Adjustment keypad**

It includes navigation keys and adjustment knobs, as shown in Figure 3.18.



Figure 3.18 Adjustment keypad

a) **Adjusting knob**

Turn the knob to adjust the setting value of the currently active input box.

b) **[←Tab] and [→Tab]**

- Move the selection menu to the left or right.
- Toggle the activated option in the dialog box.

c) **[↑] and [↓]**

Move up and down in the menu to select menu items. At the same time, the dialog box has the following functions: change the value, select the item in the drop-down list, and select the desired option from a set of option buttons.

2) **Setting keypad**

The function keypad contains four common setting keys, as shown in Figure 3.19.



Figure 3.19 Setting keypad

a) **[CBK]**

It is used to call back the status.

b) **【Save】**

It is used to save the user setting status.

c) **【System】**

d) This button is for opening the main menu of the system, with the first-level menus listed in the auxiliary menubar, including Configure, Record/Run, SPSP, Windows Taskbar, Reset, Define User Reset State, Language.

e) **[Help]**

Open the user manual.

f) **【Macro/LCL】**

This button is for switching to macro when the analyzer is under programmed control, or opening the micro menu when the analyzer is not under programmed control.

g) **【Reset】**

This is the shortcut key for resetting of the analyzer. If the user has saved the reset state and checked Enable User Reset State, the state saved by the user will be restored when this button is pressed, otherwise, the system reset state will be restored.

3) **USB interface**

The USB interface can be used to connect keyboard, mouse or other USB devices.

Four USB interfaces are provided on the rear panel, complying with the USB3.0

3 Quick Start

3.2 Description of Front and Rear Panels

specification, and two USB interfaces are provided on the front panel, complying with the USB2.0 specification.



Figure 3.20 USB interface (rear panel)

4) Display screen

As shown in Figure 3.21, the screen display of the analyzer uses TFT LCD, with the following technical indicators:

- 12.1-inch touch screen
- Resolution: 1280 × 800

For more information on the functions and settings of each display element on the screen, refer to "4.9 Setting analyzer display".



Figure 3.21 Display screen of the analyzer

1) [PWR On/SBY] button and indicator

The [PWR On/SBY] button and indicator are shown in Figure 3.22. The power button is used to turn on the analyzer or switch the analyzer to standby mode.

- The indicator turns white when the analyzer is turned on.

3.2 Description of Front and Rear Panels

- The indicator is orange when the analyzer is in standby.
- By pressing the power button when the analyzer is turned on, the analyzer automatically runs the operating system and the measurement application program.
- When the power button is pressed with analyzer on, the analyzer will automatically exit the application, the power supply will be turned off, and the analyzer will enter the standby mode.
- This switch is only a standby switch, which is not directly connected to the external power supply, and cannot cut off the connection between the instrument and the external power supply. The external power supply of the analyzer can be cut off via the power button on the rear panel, and the connection between the analyzer and the external power supply can be completely cut off by removing the power cord.



Figure 3.22 [PWR On/SBY] key and indicator

2) Test port

As shown in Figure 3.23, the analyzer is provided with N type (female) ports with a resistance of 50Ω, which supports change-over between RF source and receiver so as to make measurements to the DUT in two directions possible. The green indicator is used to indicate the source output port.



Figure 3.23 Test port of analyzer

3.2.2 Rear Panel Description

This section introduces the composition and functions of the rear panels of 3657 series vector network analyzers. The rear panel is shown in Figure 3.24.



3.2 Description of Front and Real Panels

Figure 3.24 Real panel of network analyzer

1) 10MHz reference connector



Figure 3.25 10MHz reference connector

a) 10 MHz reference input

As shown in Figure 3.25, this BNC (female) connector allows the analyzer to be mated with an external reference signal. The 10MHz external reference signal, if detected on this port, will be taken to replace the internal frequency reference of the network analyzer and serve as the frequency reference. The characteristics of the 10MHz reference input port are described as follows:

- Input frequency: $10\text{MHz} \pm 1\text{ppm}$
- Input level: $-15 \sim 20\text{dBm}$
- Input impedance: 200ohm

b) 10 MHz reference output

Reference signals with the following characteristics are provided externally via the BNC (female) connector:

- Output frequency: $10\text{MHz} \pm 1\text{ppm}$
- Signal type: Sine wave
- Output level: $13\text{dBm} \pm 4\text{dB}$ into 50ohm
- Output impedance: 50ohm

2) LAN connector

As shown in Figure 3.26, this is a 10/100/1000BaseT Ethernet connector with a standard 8-pin structure that allows for automatic selection between three data rates.



Figure 3.26 LAN connector

3) USB connector

As shown in Figure 3.27, the connector's jacks are of type A configuration (4 contacts embedded: contact 1 is on the left), which can be connected to USB mouse, keyboard, or

3.2 Description of Front and Rear Panels

other USB interface device, with 4 on the rear panel and 2 on the front panel. The port features are as follows:

- Contact 1: Vcc, 4.75V to 5.25V, max. 900mA
- Contact 2: Data-
- Contact 3: Data+
- Contact 4: Earthing

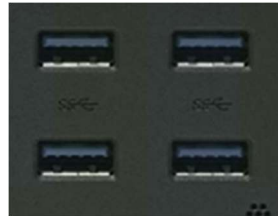


Figure 3.27 USB connector

4) Display port (DP) output connector

The DP connector, as shown in Figure 3.28, is intended for connecting to an external DP display of the corresponding resolution so that the user can observe both the internal and external displays.



Figure 3.28 Display port (DP) output connector

5) Trigger input/output interface connector

As shown in Figure 3.29, this interface is an external and auxiliary trigger input and output interface, and its functions are described as follows:

- External trigger input - When this function is enabled, the vector network analyzer will be triggered by the signal of this connector;
- External trigger readiness - When this function is enabled, the vector network analyzer will send a "ready" signal to the external device through this connector;
- Auxiliary trigger input 1/2 - When this function is enabled, the external device will send a "confirmation" signal to the vector network analyzer through this connector (the external device is ready to receive the trigger signal);
- Auxiliary trigger output 1/2 - When this function is enabled, the vector network analyzer transmits a "confirmation" signal before (or after) a measurement.



Figure 3.29 Trigger input/output connector

3.3 Analyzer Software Interface

As shown in Figure 3.30, the following operations can be performed by using the mouse/touch screen:

- **Menubar:** the menubar contains 14 sub-menus including measurement, trace, channel, frequency, power, sweep, trigger, calibration, marker, format, scale, average, display and analysis, and each sub-menu allows for setting, ON/OFF switching and other operations of the corresponding function;
- **Dialog box:** the dialog box will pop up once the corresponding button is clicked for you to perform the operations of a certain function; as shown, after the New Trace button is clicked, the New Trace dialog box pops up, and you can create a new trace herein.
- **System button:** when the system button is clicked, the system menu will pop up for system settings, including file operation, macro, user configuration and others;
- **Quick access toolbar:** you can click the quick access toolbar to perform corresponding quick operations, such as reset, file saving and recall;
- **Trace title:** you can click the trace title bar to set the current trace as the active trace, and can also right click the trace bar to display the right click menu to modify the current active trace such as the title, scale, measurement, etc;
- **Menu search:** you can type the name of the menu to be searched in the menu search input box, and then the corresponding menu will be called with one click;
- **Right click menu:** you can right click the window area to display the right click menu, including window right-click menu, coordinate right-click menu, etc., so that you can perform corresponding operations including opening/closing of window, settings of color and display of scale.

3.4 Analyzer Traces, Channels and Windows

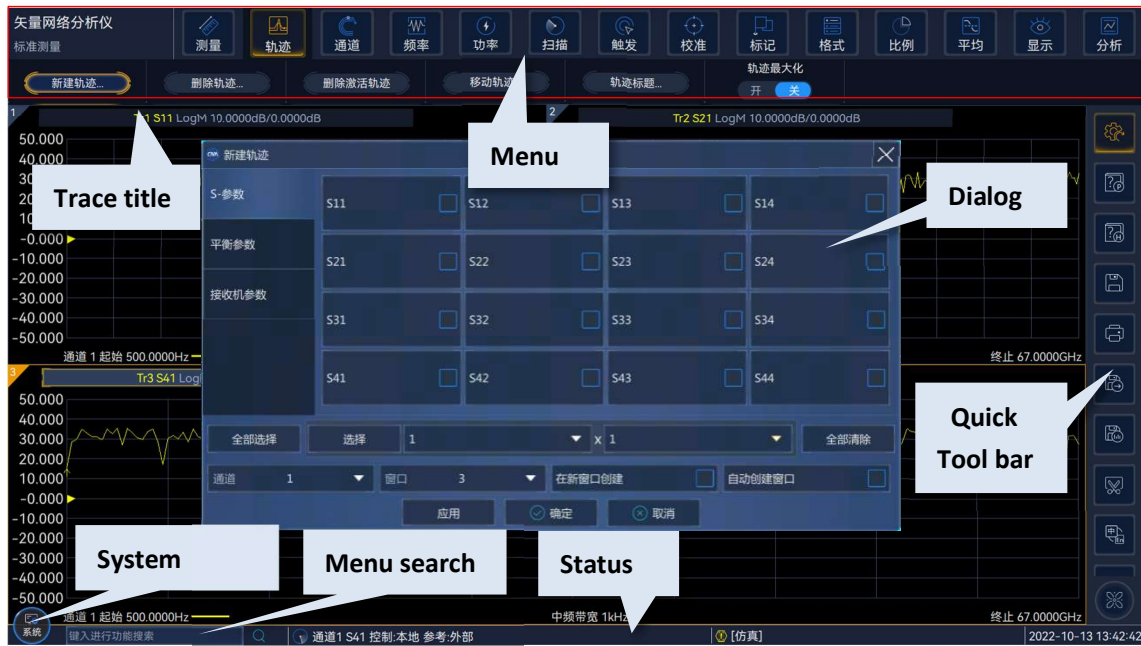


Figure 3.30 Analyzer display screen

3.4 Analyzer Traces, Channels and Windows

- [Trace](#).....55
- [Channel](#).....59
- [Window](#).....56

3.4.1 Trace

Trace is a series of measurement data points. The trace settings will affect the mathematical operation and display of measurement data. Its settings can be changed only when the trace is active. Click the corresponding trace status button to activate the trace. For detailed settings, see "Changing Trace Activation State" in "4.2 Selecting Measurement Parameters". The trace settings include

- Measurement parameters
- Display Format
- Scale
- Trace Calculation
- Cursor
- Electrical delay
- Phase Offset
- Smoothing
- Time Domain Conversion

3.4 Analyzer Traces, Channels and Windows

3.4.2 Channel

Multiple traces can be included in each channel, and the analyzer supports up to 64 channels. The channel settings determine how the traces in the channel are measured, and traces in the same channel have the same channel settings. Different channel settings are available for different channels. A channel can only change its settings when it is active. Whenever a trace is activated in a channel, the channel is also activated at the same time. For details on how to activate the trace, see "Changing Trace Activation State" in "4.2 Selecting Measurement Parameters". 【Channel Setting】 includes:

- Frequency span
- Power
- Standard data
- IFBW
- Sweep Points
- Sweep settings
- Avgs
- Trigger (some settings)

3.4.3 Window

Windows are used to observe measurement traces. The analyzer supports up to 32 windows, with up to 8 traces displayed in each window. The [View] menu is used to set the window display. For details, see "4.9 Setting the analyzer display".

1) Create a new window

Menu path: click **[Display]** → **[Window]** → **[New Window]**, and then the analyzer will create a new window with no default settings for traces. Or, you can also create a new window by operating as follows: [Right click] in the main interface → click **[New Window]**.



Figure 3.31 Creating a new window

2) Use the full-screen view window

When too many windows are opened at the same time, the trace will be unclear because the window is too small. At this time, a window can be displayed in full screen to better observe the trace in the window.

When using the mouse, there are four ways to make a window full screen display:

- With the title bar ON, double-click on the window to be maximized.
- With the title bar ON, right-click [Window] → [Trace Maximize]].
- With the title bar ON, right-click the interface → [Maximize].
- Menu path: click [Display] → [Window] → [Maximize]].

3.5 Analyze Data

●	Cursor.....	57
●	Trace Operation and Statistics.....	66
●	Limit Test.....	69
●	Ripple Test.....	72
●	Bandwidth Test.....	74
●	Formula Editor.....	77

3.5.1 Cursor

Use the cursor to read measurement data, search for specific types of values, or change stimulus settings. Up to 9 normal cursors and 1 reference cursor can be used for each trace.

●	Create a Cursor.....	57
●	Move a Cursor.....	63
●	Cursor Search.....	59
●	Cursor Functions	63
●	Advanced Cursor Options Settings.....	64
●	Cursor Table.....	65

3.5.1.1 Create a Cursor

1) Cursor menu

Menu path: click **[Marker]** to display the **cursor** toolbar.

Select the cursor to open in the **[Mkr]** box.

In **[Marker 1]** box, set the position for cursor 1.

3.5 Analyze Data



Figure 3.32 Creating a cursor using the Cursor menu

2) Cursor toolbar

a) [Marker] box

Select the cursor you want to define.

b) [Marker n] box

Define the stimulus value for the X-axis of the selected cursor.

c) [Max] check box

When this check box is checked, the maximum value of the measurement result will be marked.

d) [Min] check box

When this check box is checked, the minimum value of the measurement result will be marked.

e) [Tracking] check box

When this check box is checked, the marker tracking function is turned on, and the maximum and minimum values can be tracked.

f) [Marker Function] toolbar

This toolbar allows for selecting the start, stop, center, span, reference, delay, etc. of the marker measurement.

g) [Marker Search] button

When this button is clicked, the **Marker Search** dialog box will pop up.

h) [Marker Settings] button

When this button is clicked, the **Marker Settings** dialog box will pop up.

3.5.1.2 Move the Cursor

Move the cursor using one of the following methods:

- a) Click [Marker n] box in the **marker toolbar**, and move a cursor as follows:
 - i. Enter the value to be marked by the cursor directly.

3.5 Analyze Data

- ii. Turn the knob to move the cursor.
- iii. Click the knob button in the **[Marker n]** box to change the cursor stimulus value.
- iv. Press **[↑]** or **[↓]** in the **[Marker n]** key area to change the cursor stimulus value.

3.5.1.3 Cursor Search

Use the Cursor search function to search for a specific measurement. If there is no matching measurement data, the cursor will remain unchanged at the current position.

1) Method to search for cursor

Menu path: **[Marker]** -> **[Marker Search]**.

Complete the settings of cursor search in the dialog box, click **[OK]** to start cursor search and close the dialog box.

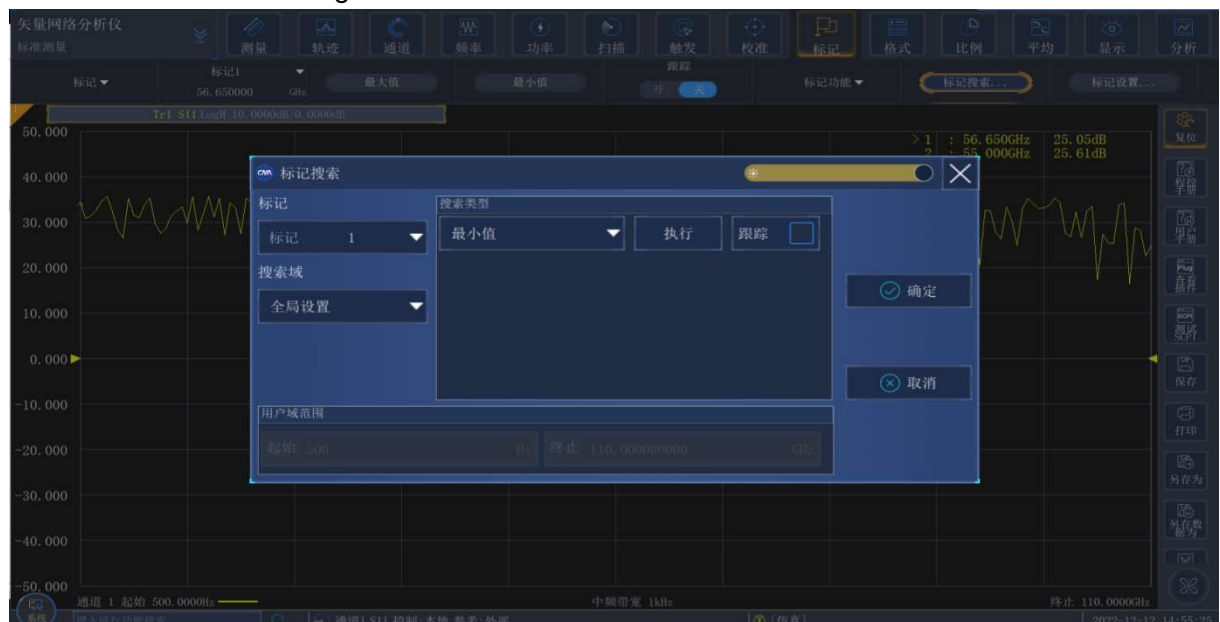


Figure 3.33 Cursor search

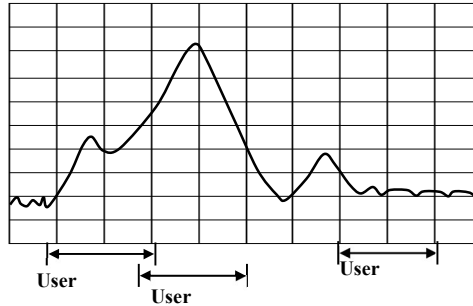
3.5 Analyze Data

2) **Cursor search dialog box**a) **Cursor zone**

Select the cursor for search definition.

b) **Search domain zone**

Define the cursor search range. The default search domain is Full Bandwidth. In addition, 9 user-set search ranges are supported. When User Settings are selected, the user settings must be defined in the User Domain range zone. The User Settings search domains can overlap each other, and different cursors can use the same



search domain. Figure 3.34 shows a set of user-set search domains:

Figure 3.34 Schematic diagram of user-set cursor search domain

c) **User domain range zone**

The [Start] box and [Stop] box in the User Domain Range zone are used to define the search domain set by the user.

d) **Search type zone**

Define the type of cursor search. The analyzer supports the following 10 search types:

- **Min.:** Search for the minimum measurement data point.
- **Max.:** Search for the maximum measurement data point.
- **Next Peak** Search for the next peak that is lower than the current cursor magnitude, provided the peak has been defined.
- **Right Peak** Search for the next valid peak to the right of the cursor position, provided the peak has been defined.

The peaks are defined by **[Threshold]** and **[Offset]**. The **[Threshold]** box defines the minimum peak point. The peak point of the valid peak must be above the threshold, and the valleys on both sides can be below the threshold. The **[Offset]** box defines the minimum vertical distance between the peak and valley points. The vertical distance between the peak of the valid peak and the valley on both sides must be greater than the offset value. As shown in Figure 3.35, the analyzer are set as follows:

Threshold: 50 dB

Offset: 10dB

Scale: 10dB/div.

- i. Peak A is a valid peak, with both threshold and peak in compliance with the requirements;

3.5 Analyze Data

- ii. Peak B is not a valid peak, with offset not in compliance with the requirements;
- iii. Peak C is not a valid peak, with threshold not in compliance with the requirements;

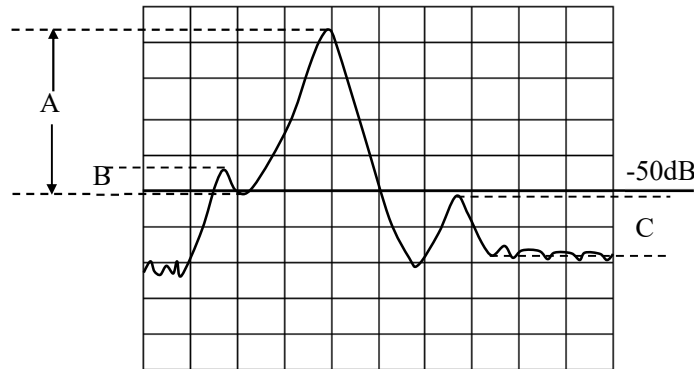


Figure 3.35 Schematic diagram of the peaks in the cursor domain

- **Left Peak** Search for the next valid peak to the left of the cursor position, provided the peak has been defined.
- **Target:** Enter the search target value in the **[Logarithmic Value]** box, click the **[Execute]** button, the cursor moves to the first target value to the right of the current cursor position, then click the **[Execute]** button consecutively, the cursor moves to the next target value to the right until the highest end of the stimulus value, and then return to the lowest end of the stimulus value to search for the target value.

① When the **[Discrete Marker]** check box is checked in the **[Marker Settings]** dialog box, the cursor moves to a discrete data point equal to the target value.

② When the **[Discrete]** check box is unchecked in the **[Marker Settings]** dialog box, the cursor moves to the interpolated data point equal to the target value.

- **Left Target** searches only the target value to the left of the current cursor.
- **Right Target** searches only the target value to the right side of the current cursor.
- **Bandwidth** When the **Bandwidth** search function is selected, set the level falling from the peak on both sides in the **[Level]** box (default is -3dB) to perform bandwidth measurement at this level, as shown in Figure 3.36. Measurements are made using Cursor 1 to Cursor 4. Cursor 1 searches for the maximum peak, Cursor 2 searches for the point where the left side of the peak drops to the specified level, Cursor 3 searches for the point where the right side of the peak drops to the specified level, and Cursor 4 searches for the center of the bandwidth. The following message is displayed when the measurement is complete:

Bandwidth: The frequency difference between the right cutoff frequency

3.5 Analyze Data

point and the left cutoff frequency point;

Center: The frequency of the midpoint between the right cutoff frequency point and the left cutoff frequency point;

Q: The value obtained by dividing the center frequency by the bandwidth;

Loss: The measured value at the center frequency point of the bandwidth;

Left cutoff: The lowest frequency of the two points below the specific bandwidth level of the signal peak;

Right cutoff: The highest frequency of the two points below the specific bandwidth level of the signal peak.

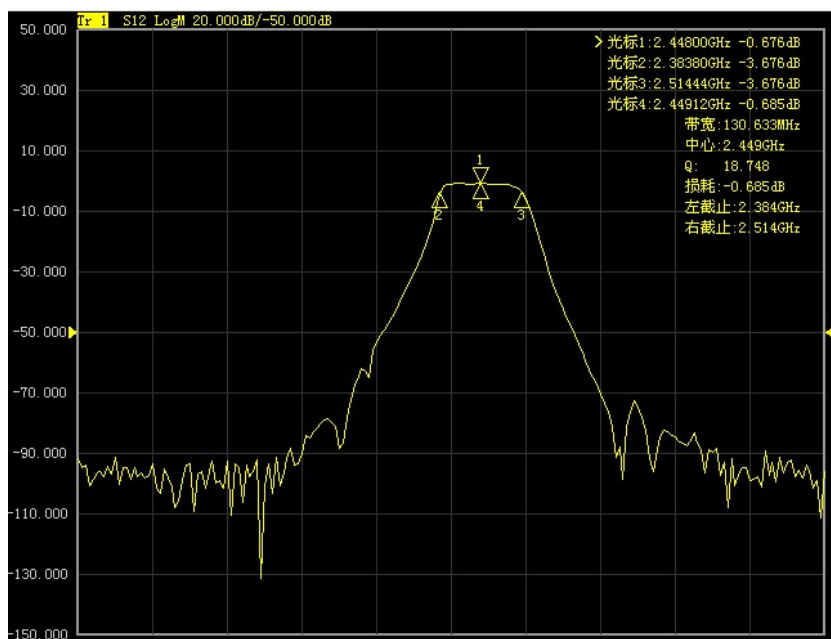


Figure 3.36 Bandwidth search

- **Notch Bandwidth** The analyzer tests the notch bandwidth of the bandpass filter, as shown in Figure 3.37.

Level: set the level value of bandwidth in the input box;

Reference position: set the reference position in the check box;



Figure 3.37 Notch bandwidth

e) **[OK] button**

Click **[OK]** to perform a cursor search of the specified type.

f) **[Tracking] check box**

When **[Tracking]** check box is checked, the analyzer executes the search function after each sweep according to the current settings of search type and search field, ensuring that the cursor is in the desired position after each sweep.

3.5.1.4 Cursor Functions

1) Steps to use cursor functions

Some instrument settings, such as start frequency, stop frequency, etc., can be changed by activating the position of the cursor, which is set as follows:

Menu path: **[Mrk]** → **[Marker Functions]**.



Figure 3.38 Setting a measurement using the Cursor Function check box

2) Cursor functions dialog box

a) **[Marker->Start] button**

Set the start value of the sweep equal to the stimulus value of the active cursor when clicked.

b) **[Marker -> Stop] button**

Set the stop value of the sweep equal to the stimulus value of the active cursor when clicked.

c) **[Marker->Center] button**

Set the center value of the sweep equal to the stimulus value of the active cursor when clicked.

d) **[Marker->Reference] button**

Set the reference value of the sweep equal to the stimulus value of the active cursor when clicked.

e) **[Marker->Delay] button**

When the **[Mkr->DLY]** button is clicked, the analyzer uses the phase slope at the active cursor to adjust the electrical delay of the receive path, which flattens the phase locus near the active cursor. This function can be used to measure the electrical length and phase offset. This function is only applicable to ratio

3.5 Analyze Data

measurement.

f) **[Marker->Span] button**

When clicked, set the span value of the sweep equal to the difference in stimulus between the active cursor and the reference cursor (Δ cursor value). This button is only valid when the Δ cursor function is enabled for the active cursor.

3.5.1.5 Marker Settings

1) **Steps to use the marker settings function**

Menu path: click **[Mrk]**→**[Marker Settings...]** to display the **Marker Settings** dialog box.



Figure 3.39 Marker Settings dialog box

2) **Marker display box**a) **[Marker Readings]**

The corresponding marker value is displayed on the screen when it is checked, and the cursor readout is turned off when it is unchecked.

b) **[Large Readout]**

The corresponding marker value is displayed in the large font when it is checked, and is displayed in normal font when it is unchecked.

c) **[Readouts per Trace]**

Set the number of readouts for each trace within 1-16.

d) **[Show All Tr Markers]**

All trace markers are displayed on the screen when it is checked, and no trace markers are displayed when it is unchecked.

e) **[Decimal Places] box**

Select the number of decimal places for stimulus and response.

f) **[Readout Position] box**

Select the percentage to move right and down.

g) **[Group Display] box**

Select to display traces and markers or not.

3) Marker properties box

a) [Marker n] check box

Select the number of marker to be set.

b) [ON] check box

The corresponding cursor is displayed on the screen when it is checked, and the cursor display is turned off when it is unchecked.

c) [Δ Marker] check box

The reference cursor under the set stimulus is turned on when it is checked, and is turned off when it is unchecked.

d) [Discrete Marker] check box

When it is checked, the cursor displays only the data of the actual measurement point, and when it is unchecked, it also displays the data points interpolated between measurement points.

e) [Format] box

Select the format used for the cursor display data. The format of cursor data can be different from that of screen trace data. By default, both data are the same.

f) [Fixed Marker]

When checking fixed cursor, according to the position on the trace when it is set as a fixed type cursor, the cursor keeps the fixed X-axis and Y-axis coordinates unchanged and does not move with the change of trace data amplitude. Its position on the X-axis can be moved by changing the cursor stimulus value, but the Y-axis coordinates remain unchanged. This type of cursor is mainly used to observe changes in the trace data. For example, a fixed cursor can be used to compare the change in insertion loss before and after filter tuning. When clearing, cancel the fixed marker, and restore the standard cursor. If it has a fixed X-axis position, and the Y-axis position changes with the amplitude of the trace data, the cursor position on the X-axis can be moved left and right by changing the cursor stimulus value.

g) Marker coupling

The marker coupling is turned on when it is checked, with coupling types including channel coupling and all coupling available for selection; the marker coupling is turned off when it is unchecked.

3.5.1.6 Cursor Table

1) Cursor table

Open the Cursor Table to display all cursor data for the active trace. The cursor data is displayed in the format specified for each cursor.

Turn on and off the display of the Cursor Table as follows:

2) Opening/closing method

Menu path: click **[Mrk]** → **[Marker]**, check **[Marker Table]** to open **[Marker Table]**/uncheck **[Marker Table]** to close **[Marker Table]**.

3.5 Analyze Data

标记▼

☒ 标记1

☒ 标记2

☐ 标记3

☐ 标记4

☐ 标记5

☐ 标记6

☐ 标记7

☐ 标记8

☐ 标记9

☐ 标记10

☐ 标记11

☐ 标记12

☐ 标记13

☐ 标记14

☐ 标记15

☐ 参考标记R

全部关闭

☒ 标记表

光标	参考	激励	响应
1		55.000GHz	23.59dB
2		55.000GHz	23.59dB

Figure 3.40 Setting Cursor Table display via Cursor menu

3.5.2 Trace Operation and Statistics

The analyzer can perform four types of mathematical operations on the current active trace and memory trace, and also provide three trace statistics functions: average, deviation and peak-to-peak.

- [Trace Calculation.....66](#)
- [Trace Statistics.....67](#)

3.5.2.1 Trace Operation

1) Steps for setting the trace calculation

The memory must be saved with a trace first before conducting any trace calculation, and trace calculation is conducted via vector calculation on previous complex number data displayed in formatting. The steps for setting the trace calculation is shown below:

Menu path: **[Analy]**→**[Save]** to display the Operate/Save dialog box.

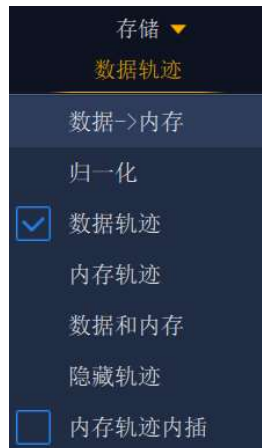


Figure 3.41 Setting trace operation

2) Operate/Save dialog box

a) [Data->Memory] button

Save the current measurement data to the memory.

b) [Normalize] button

Normalize the current measurement data to the reference, and divide the data measured by it.

c) [Data Trace] button

Display only data traces.

d) [Memory Trace] button

Display only memory traces.

e) [Hide Trace] button

No traces will be displayed.

f) [Data and Memory] button

Display both data traces and memory traces.

3.5.2.2 Trace Statistics

The analyzer provides 5 kinds of trace statistics functions: average, deviation, peak-to-peak, maximum and minimum, and can calculate statistics within the full stimulus bandwidth or user-defined bandwidth. Each channel supports 9 user-set ranges that are the same as the user-set search domain in cursor search, and they use the same memory address and therefore share the same stimulus settings. If the search domain set by the channel user is defined through the cursor search function, the same stimulus settings can be invoked in the trace statistics by selecting the corresponding user settings, and the range of such user settings can also overlap each other.

Using the trace statistics function, it is not necessary to search the maximum and minimum values, and it is convenient to measure the peak-to-peak values of passband ripple. Trace statistics are calculated according to the format of data display:

- **Cartesian format:** Calculate statistical values based on the displayed scalar

3.5 Analyze Data

data.

- **Polar and Smith chart formats:** Calculate statistical values based on the displayed values of the data in logarithmic magnitude format.

1) Active trace statistics

Menu path: **[Analy]**→**[Trace Stats]** to display the **Trace Statistics** dialog box.

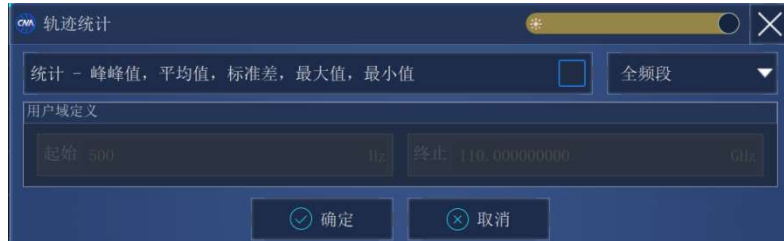


Figure 3.42 Trace Statistics dialog box

2) Trace Statistics dialog box

- a) **[Statistics - Peak to Peak, Mean, Standard Deviation, Max, Min] check box**
Turn on Trace Statistics when it is checked, and turn off Trace Statistics when it is unchecked.
- b) **[Full Span] box**
Select the span settings of trace statistics. You can select full bandwidth or customize 9 user settings.
- c) **User field definition zone**
 - i. **[Start] box**
Define the start value of the user-set span.
 - ii. **[Stop] box**
Define the stop value of the user-set span.

3) Trace Operation dialog box

The **[Trace Operation]** box is used to select the type of trace operations. The analyzer supports the following trace operations:

- a) **Data**
No mathematical operations of any kind are performed.
- b) **Data+storage**
The trace data displayed is the result of “measured data + memorized trace data”.
- c) **Data-Memory**
The trace data displayed is the result of measured data - memorized trace data”. This function can be used for simple error correction: first, the measured vector error is stored in the memory, and then the measurement error is subtracted from the measured data of the DUT.

d) Data*Memory

The trace data displayed is the result of “measured data * memorized trace data”

e) Data/Memory

The trace data displayed is the result of “measured data / memorized trace data”, which is mainly used for ratio measurements, such as gain and attenuation

3.5.3 Limit Test

The limit test function compares the measured data with defined constraints (limits) and the user-defined limits are displayed visually on the screen in the form of limit lines, which have the following advantages:

- Providing a visual indication for device debuggers.
- Providing a criterion for the device characteristics index to meet the requirements, and
- Providing a visual comparison between the measured data of the device and the specification requirements.

The limit test function compares measured data with defined limits and provides PASS and FAIL information, and supports up to 100 discrete limit line segments per trace for precise definition of limits.

- [Create and Edit Limit Lines.....69](#)
- [Limit Table.....70](#)
- [Set Limit Test.....70](#)

3.5.3.1 Create and Edit Limit Lines

The analyzer supports the creation of limit lines for all measurement traces. The limit lines consist of several discrete limit line segments, each determined by 4 coordinate values: start and stop stimulus values for the X-axis, start and stop response values for the Y-axis. The limit lines are created and edited via a limit table.

- 1) Menu path: click **[Analysis]** → **[Test]** → **[Limit Test]** to display the **Test** dialog box.
- 2) Click the **[DISP TBL]** button in the dialog box to open the Limit Table display;
- 3) Click the **[OK]** button to close the dialog box;
- 4) Click **[Limit Type]** box in the **Limit Table**, and select the limit test type in the drop-down box.

3.5 Analyze Data



Figure 3.43 Limit test

3.5.3.2 Limit Table

	类型	起始激励	终止激励	起始响应	终止响应	极限类型
1	OFF	500.000 Hz	110.000 GHz	-100.000 dB	100.000 dB	SLOP

Figure 3.44 Table of limits

Type box

The Type box is used to set the type of limit test.

- MAX: The limit test fails when the trace is above the limit line segment
- MIN: The limit test fails when the trace is below the limit line segment
- OFF: turn off this limit segment test

Start Stimulus box and Stop Stimulus box

Set the start and stop stimulus values of the limit line segment (X-axis start and stop coordinates of the limit line segment)

Start Response box and Stop Response box

Set the start and stop response values of the limit line segment (Y-axis start and stop coordinates of the limit line segment)

Limit type box

Set the limit line display mode during limit test:

- SLOP: display sloped limit lines
- FLAT: display flat limit lines
- POINT: display point limit lines

3.5.3.3 Set Limit Test

After the limit lines are created, you can choose to display or hide the limit line of a trace. When the limit lines are hidden, the corresponding limit tests are still valid. If the limit test cannot be performed for the stored trace, you can choose to send a warning

3.5 Analyze Data

sound and display a failure flag (FAIL) to indicate the limit test when the limit test fails. When the Limit Test function is turned on, the trace display of the failed part of the limit test is red by default, which can be changed by Set Color of the Limit Failure Color in the Right Mouse menu, while the color of the passed part of the limit test remains unchanged.

1) Setting method

- a) Menu path: click **[Analysis]** → **[Test]** → **[Limit Test]** to display the **Test** dialog box.
- b) Set the limit test in the dialog box.
- c) Click the [OK] button to close the dialog box.

The limit test is only conducted at the actual sweep measurement points. When the sweep points is small, the performance index of the DUT may not meet the requirements but still pass the limit test. Therefore, during the actual measurement, sufficient sweep points must be used for the limit test.

2) Limit Test dialog box**[LIM Test (ON/off)] check box**

Turn on the limit test function of the active trace when it is checked, and turn off the limit test function of the active trace when it is unchecked.

[LIM Line DISP (ON/off)] check box

Turn on the limit line display of active traces on the screen when it is checked, and turn off the limit line display of active traces when it is unchecked, but it does not affect the limit test function.

[Fail AW] check box

When checked, the buzzer sounds a warning tone if a trace data point test fails.

[Fail flag (ON/off)] check box

When checked, a fail flag is displayed on the screen if a trace data point test fails.

[PT] radio box

The symbols "v" (MAX for measurement type) and "^" (MIN for measurement type) are used to indicate the discrete limit value corresponding to the measured data point when selected.

[Line] radio box

Connect all discrete limit set points with a line when selected..

[DISP TBL] button

When clicked, the limit table is displayed for editing, and this button becomes disabled when the Limit Table is open.

[Hide TBL] button

Turn off the Limit Table display when clicked, and this button becomes disabled when the Limit Table is closed.

[EXP TBL] button

When clicked, the Save dialog box pops up. You can select a storage path to save the

3.5 Analyze Data

file in *.csv format.

[[IMP TBL] button

When clicked, the Save dialog box pops up. You can select the save path of the file to be opened and open the file in *.csv format.

[LIM Test TYP] drop-down selection box

Click the drop-down selection box to select the limit test type. There are four types: multi-type limit line, slope limit line, single-point limit line and flat limit line.

3.5.4 Ripple Test

Ripple test can be done by setting the ripple limit to assess whether the test results pass. The test compares the measured data with defined ripple limits and provides PASS and FAIL information. The test results are visually displayed on the screen. Each trace supports up to 12 discrete limit segments. Each limit segment can set the start stimulus, stop stimulus and maximum fluctuation value, and the stimulus settings of different segments can overlap.

3.5.4.1 Create and Edit Ripple Limit Lines

Ripple limit test can be carried out for multiple traces at the same time, simply by activating the traces and then setting the ripple limits.

1) Setting method

- Menu path: click **[Analysis]** → **[Test]** → **[Ripple Test]** to display the **Test** dialog box;
- Click **[Show Table]** button in the dialog box to open the ripple limit table.



Figure 3.45 Display Ripple Test dialog box

2) Ripple limit table

	类型	起始激励	终止激励	最大波动
1	关	500.000 Hz	110.000 GHz	100.000 dB

Figure 3.46 Ripple Limit table

Type box

The Type box is used to set the type of limit test.

ON: Activate this limit segment test.

OFF: turn off this limit segment test.

Start Stimulus box and Stop Stimulus box

Set the start and stop stimulus values of the ripple limit line segment.

Max. Ripple box

Set the maximum ripple value that can be displayed by the limit line in this ripple limit segment test.

3.5.4.2 Set Ripple Test

After the ripple limit lines are created, you can choose to display or hide the ripple limit line of a trace. When the limit lines are hidden, the corresponding ripple limit tests are still valid. It allows you to send a warning sound and display a failure flag (FAIL) to indicate the ripple test when the ripple test fails. When the Ripple Test function is turned on, the trace display of the failed part of the ripple test is red by default, which can be changed by Set Color of the Limit Failure Color in the Right Mouse menu, while the color of the passed part of the ripple test remains unchanged.

Menu path: click **[Analysis]** → **[Test]** → **[Ripple Test]** to display the **Test** dialog box.

The ripple test is only conducted at the actual sweep measurement points. When the sweep points is small, the performance index of the DUT may not meet the requirements but still pass the ripple test. Therefore, during the actual measurement, sufficient sweep points must be used for the ripple test.

Ripple Test dialog box:**[Ripple Test (On/ OFF)] check box**

Turn on the ripple test function of the active trace when it is checked, and turn off the ripple test function of the active trace when it is unchecked.

[Ripple LnDisp (On/ OFF)] check box

Turn on the tipple line display of active traces on the screen when it is checked, and turn off the tipple line display of active traces when it is unchecked, but it does not affect the tipple test function.

[Fail AW (On/OFF)] check box

When checked, the buzzer sounds a warning tone if a trace data point test fails.

[Fail flag (On/OFF)] check box

When checked, a fail flag (FAIL) is displayed on the screen if a trace data point test

3.5 Analyze Data

fails.

[DISP TBL] button

When clicked, the ripple limit table is displayed for editing, and this button becomes disabled when the Table is open.

[Hide TBL] button

Turn off the Ripple Limit Table display when clicked, and this button becomes disabled when the Table is closed.

[EXP TBL] button

When clicked, the Save dialog box pops up. You can select a storage path to save the file in *.csv format.

[IMP TBL] button

When clicked, the Save dialog box pops up. You can select the save path of the file to be opened and open the file in *.csv format.

[RipVAL TYP] drop-down selection box

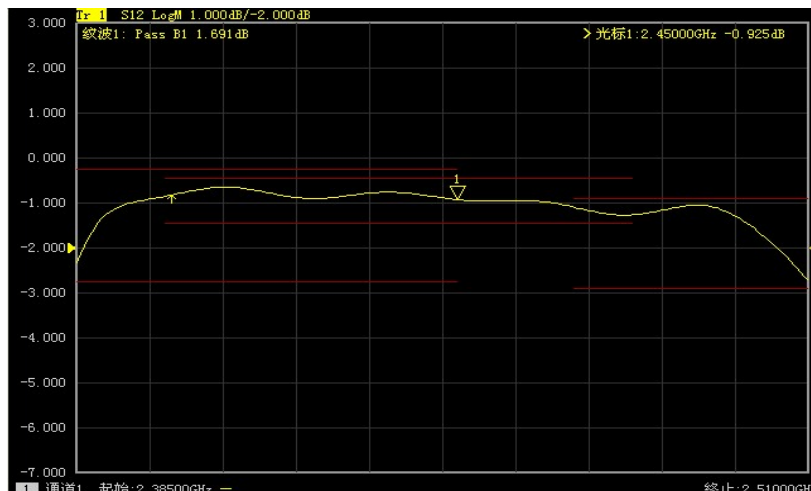
Click the drop-down selection box to select the type of ripple value: None, Absolute and Residual. The analyzer is powered on by default to Absolute.

[RipLIM Seg] drop-down selection box

Click the drop-down selection box to select the number of ripple limit segments to open, up to 12 segments.

3.5.4.3 Display Ripple Test Results

If the ripple test result passes, "PASS" will be displayed on the top left of the screen, otherwise, "FAIL" will be displayed. If multiple traces are tested at the same time, each trace will display "PASS" or "FAIL" respectively. The stimulus ranges of different ripple limit



segments can be overlapped and different fluctuation limits can be set.

Figure 3.47 Ripple test result

3.5.5 Bandwidth Test

The bandwidth test function is mainly used to test the bandwidth of the bandpass

filter.

The bandwidth test finds the peak of the signal in the passband and positions two points on each side of the passband at a specific amplitude, which is lower than the peak of the signal and can be adjusted by setting the N dB point (default setting is 3 dB). The frequency range between these two points is the bandwidth of the measured filter.

The Minimum Bandwidth and Maximum Bandwidth allowed by the user can be set before the bandwidth test, and the measured bandwidth will be automatically compared with these two values. If it does not meet the requirements, the failure message (FALL) or sound prompt will be present on the screen, so that the user can intuitively see whether the characteristics of the DUT meet the requirements.

3.5.5.1 Turn On and Set Bandwidth Test

Before performing the bandwidth test, you need to first set the bandwidth threshold (N dB point), the maximum bandwidth and the minimum bandwidth. Multiple traces can be set.

- 1) Menu path: click **[Analysis]** → **[Test]** → **[BW Test]** to display the **Test** dialog box;
- 2) Check the **[BW Test (ON/OFF)]** check box in the dialog box to turn on the bandwidth test;
- 3) Check the **[BWV Disp (ON/OFF)]** check box in the dialog box to display the measured bandwidth value on the screen;
- 4) Check the **[BWM Disp (ON/OFF)]** check box in the dialog box to display the positioning mark for the bandwidth on the screen;
- 5) Click the **[N dB point]** box to set the bandwidth threshold;
- 6) Click the **[Min BW]** box to set the minimum bandwidth;
- 7) Click the **[Max BW]** box to set the maximum bandwidth;
- 8) Check the **[Fail AW (ON/OFF)]** check box in the dialog box to provide an audible alert in the event of a bandwidth test failure;
- 9) Check the **[Fail flag (ON/OFF)]** check box in the dialog box to display the FALL prompt in case of a bandwidth test failure;
- 10) Click **[OK]** to complete the bandwidth test settings.

3.5 Analyze Data



Figure 3.48 Bandwidth Test dialog box

3.5.5.2 Display Bandwidth Test Results

Bandwidth test results are displayed in the upper left corner of the trace, and a PASS prompt will be displayed when the test is passed. The red T-shaped bandwidth flag are



distributed on both sides of the bandwidth.

Figure 3.49 Bandwidth Test display results

3.5.5.3 Turn Off Bandwidth Test

- 1) Click **[Analysis]** → **[Test]** → **[BW Test]** to display the **Test** dialog box;
- 2) Check the **[BW Test (ON/OFF)]** check box and uncheck the box to turn off the Bandwidth Test.
- 3) Click the **[BWV Disp (ON/OFF)]** check box and uncheck the box to turn off the measured bandwidth value display;

3.5 Analyze Data

- 4) Click the [BWM Disp (ON/OFF)] check box to uncheck the box to turn off the bandwidth locator mark display;
- 5) Click [OK] to turn off the bandwidth test.

3.5.6 Formula Editor

● Overview.....	77
● How to Use Formula Editor.....	78
● Data Used by Formula Editor and Precautions.....	82
● Save Formula Editor Data.....	84

3.5.6.1 General

For the purpose of illustration, the following 2 terms are first defined:

Reference Trace: the trace used in the formula is used as data.

Formula Trace: It is the trace after the formula operation, which will be displayed in the currently active trace.

The formula editor allows the user to enter an algebraic expression that performs mathematical operations on the measurement data, and the results of the operations can be displayed as a data trace. The measurement data used for the formula traces can be from the same channel or from different channels. The mathematical expressions entered by the user may consist of basic operators, built-in functions, and parameters. The measurement parameters or trace data used in the formulas are taken from the vector network analyzer. When a calculable formula is entered into the formula input box and the activation box is checked, the currently active trace will become the formula trace. The data of the formula trace is the calculated data, and the formula trace curve can be updated in real time with the change of data. For example, if you enter the formula $S_{21}/(1-S_{11})$, the data for each point on the formula trace calculated by this expression is obtained by dividing the S_{21} data of the corresponding point by 1 minus the data of S_{11} . If the trace has 201 points, this expression will operate 201 times, i.e., once for one point.

For example, if measuring a three-port DUT, which is not a routine measurement task for the analyzer. It can be accomplished with the formula editor. If one of the results desired by the user is a formula trace in logarithmic format, it can be expressed as: $S_{21}+S_{23}-S_{13}$. However, the data used by the formula editor is unformatted complex data, so the user will need to enter the formula to be changed to $S_{21}*S_{23}/S_{13}$ in order to achieve the purpose, as shown in Figure 3.50; when the formula is entered, it is displayed as $Eq=S_{12}*S_{23}/S_{13}$ on the currently active trace of the vector network analyzer, as shown in Figure 3.51.

3.5 Analyze Data



Figure 3.50 Formula Editor dialog box

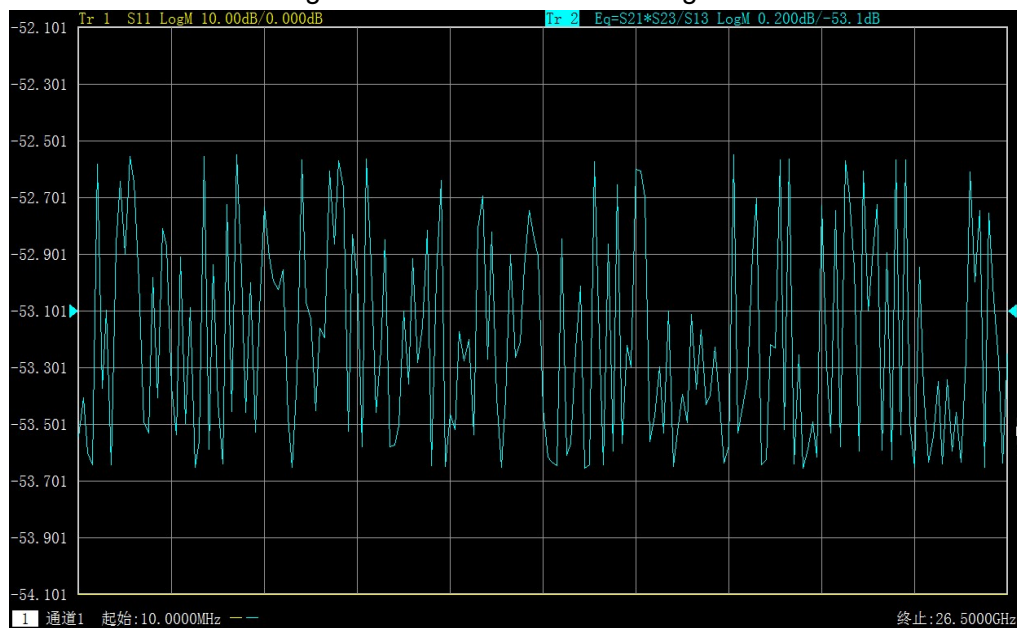


Figure 3.51 Display of formula trace

3.5.6.2 Method to Use the Formula Editor

1) Open the Formula Editor dialog box

The user clicks **[Analy]**→**[FRM Editor]** in the VNA menu to open the Formula Editor dialog box.

2) Method to enter a formula in the Formula Editor dialog box

a) Input of functions

You can enter a function in the formula input box by clicking the function list in the [Function/Constant], or you can directly enter a function with the keyboard.

b) Input of operators

Enter operators in the Formula Input box by clicking the Operators List under [OPER], or enter operators with the keyboard.

c) Input of traces and channels

Click the drop-down list option under [Trace] or [Ch PRM] with the mouse to enter trace and channel parameters as parameters or variables of the formula, or enter operators with the keyboard.

d) Input of data

Enter a number by clicking the number button on the right, or enter an operator with the keyboard.

3) Activation of check boxes

The default state of the active check box is checked. Only when it is checked will it be judged whether the formula entered by the user is correct. If the formula is computable, it will be calculated. The expression Eq=XXX of the current formula can be displayed through the active trace in the currently active window of the vector network analyzer. If the Activate check box is unchecked, no calculation will be performed regardless of whether the entered expression is computable or not. In addition, if the Activate check box is checked, but the formula entered is not computable (e.g., enter "2+1+"), the check box will change to a gray button.

4) Saving and deletion of formula

The user can save any of the expressions entered by the user in the formula input box by clicking the [StoFormu] button, and these formulas are saved to the drop-down menu of the Formula Input box. The user can select a formula by clicking the drop-down arrow to the right of the Formula Input box and then clicking the formula to be used in its drop-down list.

In addition, the user can delete the formula in the Formula Input box by clicking [DelFormu].

5) Backspace button

[<-BS]: This button is used to delete the previous character of the cursor. If there is no character on the left, the cursor position remains unchanged.

6) Left-shift cursor and Right-shift cursor

[<-]: Left-shift cursor button, used to move the current cursor to the left by one character. The character on the left of the cursor will not be deleted. If there is no character on the left, the cursor position will not change.

3.5 Analyze Data

[->]: Right-shift cursor button, used to move the current cursor to the right by one character. The character on the right of the cursor will not be deleted. If there is no character on the right, the cursor position will not change.

7) Selection of function/constant

It is the built-in function library option by default, under which there are commonly used formulas and constant variables that can be selected by clicking. The roles of functions/constants are shown in Table 3.5.. It should be noted that if the parameter of a function is a complex number, the user can also use scalar as its parameter. Scalar data is a complex number whose imaginary part is zero.

Table 3.5 Function/constant list

acos(scalar a)	Return the arcsine of parameter a, in radians. Parameter a is a scalar number
asin(scalar a)	Return the arcsine of parameter a, in radians. Parameter a is a scalar number
atan(scalar a)	Return the arctangent of parameter a, in radians. Parameter a is a scalar number
atan2	Return the phase of complex a, in radians The following two parameter forms are available: atan2 (complex a) - return the phase, in radians and the parameter a is a complex number; atan2 (scalar a, scalar b) - return the phase, in radians, and the parameters a and b are a scalar numbers
conj(complex a)	Return the conjugate complex of complex a
cos(complex a)	Find the cosine of complex a; both the real and imaginary parts of a are in radians
cpx(scalar a, scalar b)	Return a complex value (a+jb), both a and b are scalar numbers
getNumPoints()	Return the number of points currently swept
im(complex a)	Return the imaginary part of complex a
kfac(rComplex a, rComplex b, rComplex c, rComplex d)	Return value $k = (1 - a ^2 - d ^2 + a*d-b*c ^2) / (2 * b*c)$, the parameters a, b, c and d are complex number
ln(complex a)	Return the natural logarithm of complex a
log10(complex a)	Return the logarithm of 10 of complex a
mag(complex a)	Return the modular value of complex a. the parameter a is

	a complex number
mu1(complex a, complex b, complex c, complex d)	Return value $\mu_1 = (1 - a ^2) / (d - \text{conj}(a) * (a*d - b*c) + b*c)$, the parameters a, b, c and d are complex numbers
mu2(complex a, complex b, complex c, complex d)	Return value $\mu_2 = (1 - d ^2) / (a - \text{conj}(d) * (a*d - b*c) + b*c)$, the parameters a, b, c and d are complex numbers
phase(complex a)	Return atan2(a), which is the calculation of the phase, in radians, and the parameter is a complex number
PI	Constant circumference with the value 3.141592
re(complex a)	Return the real part of complex a; Parameter a is a complex number
sin(complex a)	Return the sine of complex a; Parameter a is of complex number, in radians
sqrt(complex a)	Return the square root of the complex a module; Parameter a is a complex number
tan(complex a)	Return the tangent of complex a; Parameter a is a complex number, in radians

8) Selection of operators

Operators are shown in Table 3.11, the user can select the operators to be used such as the four basic operators "+", "-", "*", "/" and the left or right brackets by mouse click. In addition, the Formula Editor also supports direct input of complex data for operation. For example, if the user wants to enter a complex constant of $2+j3$, he/she has to enter <2:3> in the formula editor to represent the complex constant $2+j3$. If the user wants to enter the formula $(2+j3)*S11$, he/she has to enter the expression <2:3>*S11 in order to calculate it.

Table 3.6 Operators List

+	Addition operator
-	Subtraction operator
*	Multiplication operator
/	Division operator
(	Left bracket
)	Right bracket
,	Comma, used to separate multiple parameters
<	Enter the start character of the complex number

3.5 Analyze Data

:	Used to separate the real and imaginary parts of the complex number
>	Enter the terminator of a complex number

9) Trace data

The user can select a trace or trace storage data in the list below it, but cannot select the currently active trace.

10) Channel parameter data

The user can only select the S-parameter of the currently active channel as the parameter data for the input expression.

11) Number keypad

The user can enter numbers by clicking the number keys in the number pad, positive and negative symbols by clicking [+ / -], and decimal points by clicking [•].

3.5.6.3 Data Used by Formula Editor and Precautions

The data of each point in the formula trace is obtained by using the data of the corresponding point of the reference trace. If the trace has 201 points, the formula will calculate 201 times, i.e. once for each data point. For example, if the current trace is Tr4 and the formula is Tr2+S11, then Tr4 will become a formula trace, and both Tr1 and S11 are the reference traces of the equation trace. And the currently active trace Tr4 will be shown as Tr4 Eq= Tr2+S11 on the display window of the vector network analyzer. as shown in Figure 3.52.

Notice

Precautions for using formula editor

- 1) If the formula is active and computable, the currently active trace does not show the original measurement parameter name, but becomes a formula trace. For example, if the currently active trace is Tr4 S22, it will become a formula trace Tr4 Eq= Tr2+S11 immediately after entering the formula Tr2+S11. However, if the input formula is not computable, it will be displayed as the original trace name. For example, if the user input Tr2+S11+, the currently active trace will be displayed as the original Tr4 S22.
- 2) The formula trace cannot use the currently active trace as the reference trace. For example, if the currently active trace is Tr4, Tr4 cannot be entered as a parameter in the formula.
- 3) The reference trace can be selected from S-parameter and memory trace. If the memory trace is used as a parameter, it must be ensured that the memory trace already exists, otherwise the formula is not computable.
- 4) When using traces of other channels as reference traces, it should be noted that the reference trace must be the displayed trace and must be referenced in the way of Trx. Only when the data points of the reference trace and the formula trace are the same can they be calculated. For example, the number of data points in the reference trace is 201, and the number of data points in the formula trace is also 201.

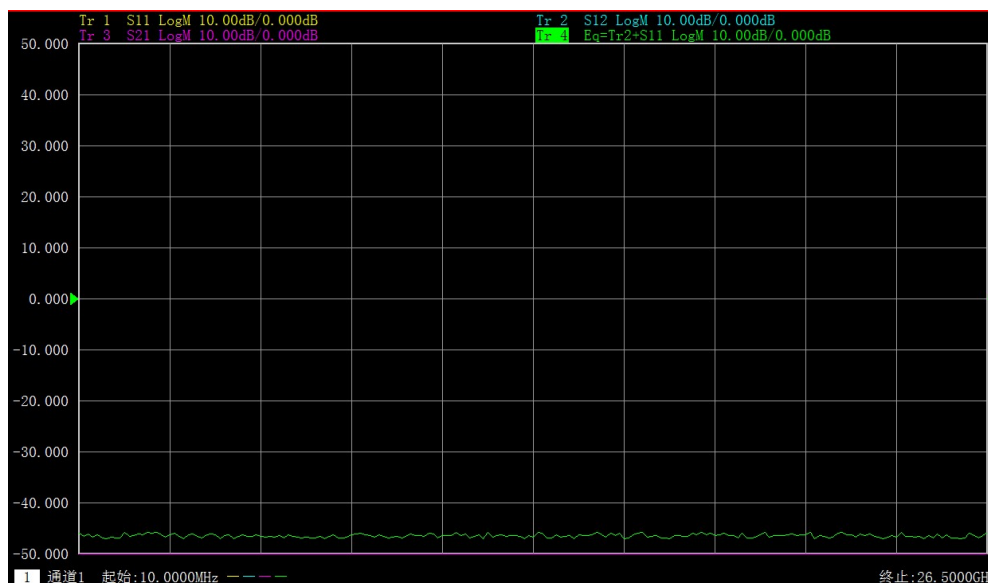


Figure 3.52 Formula Trace display window

3.6 Data Output

3.5.6.4 Save Formula Editor Data

Formula data can be saved as *.cti, *.prn, *.dat format data, but the name of the test parameter in the saved file is still the name of the original measurement parameter. For example, the original name of Tr2 is S22. If the input formula is 10+S22, the name of the saved measurement parameter remains the same, or S22, but the data will change to the value of 10+S22, and the equation trace cannot saved as a data file in *.snp format.

3.6 Data Output

- [Save and callback files.....84](#)
- [Printer Displaying Measurement Results.....90](#)

3.6.1 Save and Callback Files

The 3657 series vector network analyzer supports the functions of file save and callback in various formats.

3.6.1.1 Save Files

1) Method to save files

System path: click **[System]** → **[Save]/[Save As...]**
Menu path: click **right toolbar** → **[Save]/[Save As...]**.

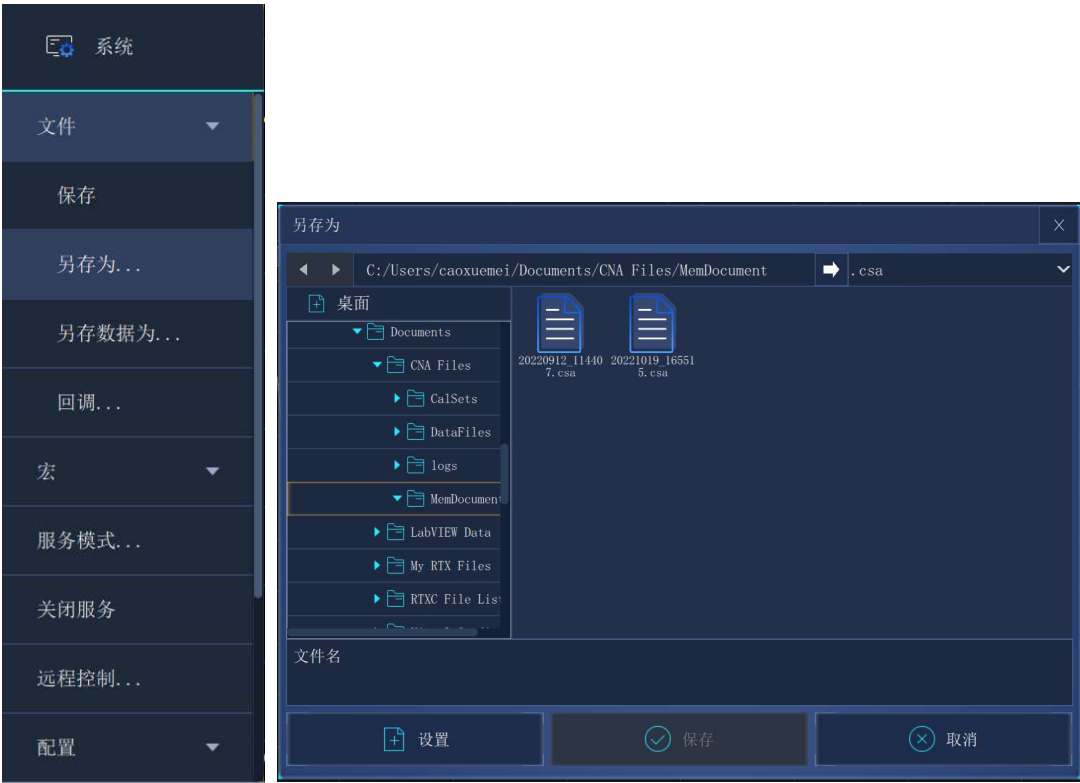


Figure 3.53 Save files

2) [Save] file menu item

When **[Save]** is clicked, the analyzer saves the instrument status and calibration data to the default file (default.csa) in the specified directory. If the default file already exists, the analyzer displays a dialog box to confirm whether to overwrite it.

3) [Save as...] menu item

Open the Save As dialog box to save the file.

a) [Save in] drop-down box

Display and set the path where the file is saved.

b) File list box

Display the folder under the current path and all files matching the save type. Click the file to set the saved file name, and click the folder to change the current path.

c) [File Name] box

Display the file name entered or clicked in the file list box.

d) [Save TYP] box

Select the type of file to save. The analyzer supports the following file types:

i. csa file

The csa file stores the state and actual calibration settings of the analyzer.

ii. cst file

The cst file stores the state and link to calibration set of the analyzer.

iii. sta file

The sta file stores only the state data for the analyzer.

iv. cal file

The cal file stores only the calibration set information for the analyzer.

v. snp file

The snp file stores valid data for all S-parameters on the active measurement channel, as follows:

The s1p file stores the data of 1 S-parameter for single-port measurement;

The S2p file stores the data of 4 S-parameters for two-port measurement;

The s3p file stores the data of 9 S-parameters for three-port measurement;

The s4p file stores the data of 16 S-parameters for four-port measurement;

vi. csv file

The csv file stores the stimulus and response data for the currently active trace or all traces, with data separated by commas.

vii. cti file

The cti file stores the stimulus and response data for the active trace or all traces.

viii. mdf file

3.6 Data Output

The mdf file stores the stimulus and response data for the currently active trace, with data separated by space.

ix. prn file

The prn file stores only the stimulus and response data for the currently active trace.

x. jpg file

The jpg file stores the information displayed on the screen in the format of compressed bitmap.

xi. bmp file

The bmp file stores the information displayed on the screen in the format of lossless bitmap.

3.6.1.2 Callback File

1) Callable status and calibration data:

a) csa file

The csa file contains all state data and actual calibration settings of the analyzer.

b) cst file

The cst file stores all the measurement state and calibration data of the analyzer, so calling the cst file can save test time.

c) sta file

The sta file stores the analyzer state data, including analyzer settings, trace data, limit lines and cursors.

d) snp file

The snp file contains valid data of all S-parameters on the measurement channel. Specifically, the s1p file contains data of 1 S-parameter for single-port measurement, the s2p file contains data of 4 S-parameters for two-port measurement; the s3p file contains data of 9 S-parameters for three-port measurement; the s4p file contains data of 16 S-parameters for four-port measurement.

e) cal file

The cal file stores only calibration data and does not contain state data of the analyzer. The correction accuracy of calibration data is related to the status settings of the instrument. Therefore, it is necessary to ensure that the status settings of the instrument when calling back the file is consistent with that during calibration in order to obtain the highest measurement accuracy, otherwise the calibration accuracy cannot be guaranteed.

2) Method to call back files:

System path: **[System]** → **[Recall...]**

3.6 Data Output

Menu path: click **Right Toolbar**→**[Recall...]** to display the **Open** dialog box.

Select the type of file to be loaded in the [File TYP] box.

Set the directory of the callback file by the [Find Range] box and the [File List] box below.



Figure 3.54 Callback file

Load the recall file as follows:

- i. Double-click the recall file in the **[File List]** box.
- ii. Click the recall file in the **[File List]** box, and then click **[Open]**.
- iii. Input the name of recall file in the **[File Name]** box, and then click **[Open]**.

3.6.1.3 Data File

Data files save measurements in ASCII format. These files can be edited using text editing software, spreadsheet software, but cannot be called up by the analyzer itself. The analyzer can save three types of data files.

1) cti file

The cti file saves the measurement data of the active trace or all traces. The data is stored in formatted or unformatted form. The data storage method is defined via the **Data Save Settings** dialog.

The dat file is saved as follows:

- a) Menu path: **[File]** → **[Save Data As...]** to display the **Save As** dialog box.
- b) Set the type of the saved file as Data File (*.dat) in the [Save TYP] box.
- c) Set the directory where the file is saved with the [Save in:] box and the [File List] box.
- d) Set the name of the saved file in the [File name] box.
- e) Click **[Settings]** to display the **Define Data Saves** dialog box.

3.6 Data Output

- f) Click the [Save] button to finish saving the data.
- g) Set the content and format of the save file in the dialog box, click the [FRMT Data] or [UnForMa] button to save the data file, and close the dialog box.



Figure 3.55 Data Save Settings dialog box

CTI Contents Area

The Save Content zone defines which trace data will be saved to a file.

- a) **[Auto] radio box**
Save the data of all window activation traces.
- b) **[Active Trace] radio box**
Save the data of the currently active trace.
- c) **[All Trace] radio box**
Save the data of all traces in the currently active window.
- d) **[Active Channel Trace] radio box**
Save the data of all traces in the currently active channel.

CTI Save Format zone

- a) **[Auto] radio box**
Save the data in the actual display format of the trace.
- b) **[Log/Angle] radio box**
Save trace data in logarithmic amplitude format.
- c) **[Line/Angle] radio box**
Save trace data in linear amplitude format.
- d) **[Real/Imag] radio box**
Save trace data in real part/imaginary part format.
- e) **[Display Format] radio box**
Save trace data in the format set in the Save Format zone.

2) snp (s1p and s2p) files

A file in snp format can be called by computer-aided engineering (CAE) software

3.6 Data Output

(such as ADS from Agilent) and is a data output file, but cannot be called by the analyzer itself. The s1p file saves the characteristics of a single-port device and contains only 1 S-parameter (S11 or S22), while the s2p file saves the characteristics of a dual-port device and contains 4 S-parameters. If the full dual-port correction is turned on, all 4 S-parameters will be saved in the s2p file. If the full dual-port correction is turned off, the analyzer will save as much of the measurement data as possible in the s2p file. For example, if the full dual-port correction is turned off, the currently active trace is S11, and there are also S21 measurements in the channel, the measurements of S11 and S21 will be saved in the s2p file, because there are no valid measurements of S22 and S12, and the corresponding data in the s2p file is 0.

The snp file is saved as follows:

- a) Menu path: click right menubar → [Save As...] to display the Save As dialog box.
- b) Set the type of the saved file to **Data File (*.s1p)** or **Data File (*.s2p)** in the **[Save Type]** box.
- c) Set the directory where the file is saved with the [Save in:] box and the [File List] box.
- d) Set the name of the file in the [File Name] box.
- e) Click **[Setting]** box, and open **Define Data Saves** dialog box, as shown in Figure 3.65.
- f) Click the [Save] button to finish saving the data.

SNP Save Format zone

- a) **[Auto] radio box**
Save the data in the actual display format of the trace.
- b) **[Log/Angle] radio box**
Save trace data in logarithmic amplitude format.
- c) **[Line/Angle] radio box**
Save trace data in linear amplitude format.
- d) **[Real/Imag] radio box**
Save trace data in real part/imaginary part format.

3) prn file

The prn file saves the measurement data of the active trace in rows and columns. Each row corresponds to a measurement point. The first column corresponds to the measured stimulus value, and the second column corresponds to the measured response value. The columns are separated by commas (,), in the following format:

S ₁₁	Log Mag
LIN_SWEEP(Hz)	LOG_FORMAT(dB)
3.000000e+005	-9.232986e+000
7.502250e+008	-3.219671e-001
1.500150e+009	-6.892332e+000

3.6 Data Output

```
2.250075e+009    ,   -1.146303e+000
3.000000e+009    ,   -1.245240e+001
```

Saving method of prn file

Menu path: click right menubar → [Save As...] to display the Save As dialog box.

Set the type of the saved file as List File (*. prn) in the [Save TYP] box.

Set the directory where the file is saved with the [Save in:] box and the [File List] box.

Set the name of the file in the [File Name] box.

Click the [Save] button to save the file and close the dialog box.

3.6.2 Printer displaying measurement results

The analyzer allows the measurement display to be output through a printer or printed to a specified file. The printer may be a local or network printer, or a USB/LAN interface printer. It can be used as long as the printer is added via the Windows operating system.

3.6.2.1 Printing Content Settings

Method to set printing content

Menu path: click **right toolbar** → [Print] → [Settings] to display **Page Settings** dialog box.



Figure 3.56 Setting print content

Page Settings dialog box

1) Window information zone

a) [Print] check box

Print window trace information when it is checked, and does not print when it is

3.6 Data Output

unchecked. The following two check boxes are only valid if this check box is selected.

b) [PRT ONE Pg] check box

when it is checked, only one window will be printed per page. when it is unchecked, all window contents will be printed continuously without mandatory page breaks.

c) [PRT Active Win] radio box

When selected, only the currently active window is printed.

d) [PRT All] radio box

When selected, only the currently active window is printed.

2) Other information table zone

a) [Print] check box

Print channel setting information when selected. The check box below is valid only when selected.

b) [Time], [Ch State], [Seg Table], [Lim Table], [Marker Table] check boxes.

Print the corresponding information when selected.

3) [Multicolor] and [Invertcolor] radio boxes for print color

Print the corresponding information when selected.

4) [Win & Info SEP] check box

The window and other information are printed separately when it is checked and combined when it is unchecked.

3.6.2.2 Print

After the printer is added and the print content is set in the analyzer, the measurement information can be output through the printer as follows:

Menu path: click **right toolbar** → **[Print]** to display **Print Configuration** dialog box.

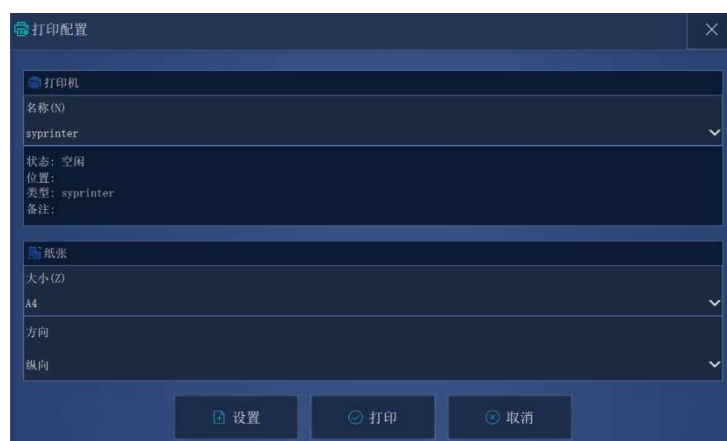


Figure 3.57 Printing measurement information

3.6.2.3 Print to File

The analyzer supports outputting the printed content to a bitmap (bmp) file. If multiple pages are needed for printing, multiple bitmap files will be created automatically, each corresponding to one page, and the other files are identified by 'file name (number).bmp', such as amp.bmp, amp(1).bmp, amp(2).bmp, which are printed as follows:

Menu path: **right toolbar** → **[Print to File...]**.

Set the directory and file name where the file is stored in the dialog box.

Click the [Save] button to store the file.

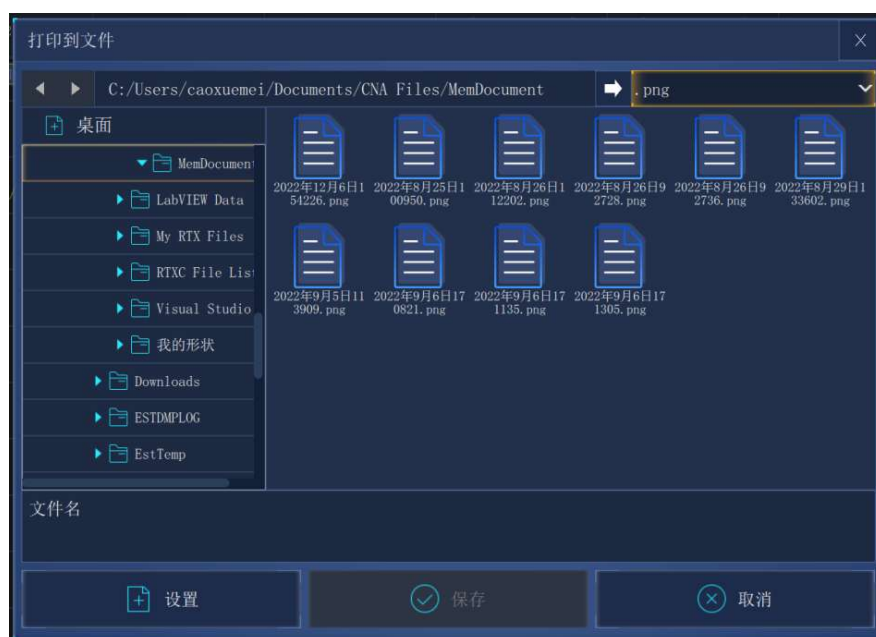


Figure 3.58 Print to file

4 Measurement Settings

This chapter introduces the operation of the 3657 series vector network analyzer during measurement: how to create a new measurement with a known state by resetting the analyzer, then select the measurement settings and adjust the display of the analyzer for better observation of the measurement results. It includes

●	Reset Analyzer.....	93
●	Select Measurement Parameters.....	97
●	Set Frequency Range.....	106
●	Set the Power Level of Signal.....	106
●	Set the Sweep	109
●	Trigger Mode.....	115
●	Set Data Format and Scale.....	119
●	Observe Multiple Traces and Open Multiple Channels.....	124
●	Set Analyzer Display	125

4.1 Reset Analyzer

●	Default Resetting Status.....	93
●	User Resetting Status.....	96
●	Reset Analyzer.....	97

4.1.1 Default Resetting Status

Press [RST] key on the front panel to return the analyzer to a known default state, called the reset state, which is set as follows:

- 1) **Measurement parameters:** S11
- 2) **Frequency settings:**
 - a) Start frequency: 9kHz
 - b) Stop frequency: 9GHz
 - c) Point frequency: 2 GHz
- 3) **Power settings:**
 - a) Test port power: -5dBm
 - b) Coupling port power: ON
 - c) Attenuation: auto attenuation
 - d) Attenuation value: 0 dB
 - e) Power slope: OFF
 - f) Slope: 0dB/GHz
- 4) **Sweep settings:**

4.1 Reset Analyzer

- a) Sweep type: linear frequency
- b) Sweep time: auto
- c) Sweep points: 201

5) Segment sweep settings:

- a) Number of open segments: 1
- b) Start frequency: 9kHz
- c) Stop frequency: 9GHz
- d) Points: 21
- e) Power: -5dBm
- f) IF bandwidth: 1kHz

6) Trigger settings:

- a) Trigger source: internal
- b) Trigger mode: continuous sweep

7) Display format:

Format: Logarithm magnitude

When different formats are selected, the corresponding detailed settings are shown in Table 4.1:

Table 4.1 Default format settings for 3657 series vector network analyzer

Format	Scale	Reference Position	Reference Value
Log Mag	10dB	5	0dB
Phase	45	5	0
Group Delay	10ns	5	10fs
Linear Mag	100mU	5	500mU
SWR	1U	5	6U
Real part	2U	5	0U
Imaginary part	2U	5	0U
Polar coordinate	1U	None	1U
Smith Chart	1U	None	1U

8) Response settings:

- a) Number of channels: 1
- b) IF bandwidth: 1kHz
- c) Ave: Off
- d) Averaging factor: 1

- e) Smooth: Off
- f) Smoothing factor: 2.49% of the value range
- g) Electrical delay: 0 s
- h) Velocity factor: 1
- i) Phase offset: 0°
- j) Display trace: Data Trace

9) Calibration settings:

- a) Corr: OFF
- b) Interpolation: ON
- c) Calibration type: none
- d) Number of calibration kit: Current calibration kit number
- e) System impedance: 50Ω
- f) Port Extension: OFF
- g) Port extension value: 0s

10) Cursor settings:

- a) Start frequency: full-scale center frequency
- b) Reference cursor R: OFF
- c) Discrete cursor: OFF
- d) Format: Track format
- e) Type: standard
- f) Cursor search type: minimum value
- g) Search domain: full bandwidth
- h) Cursor table: empty

11) Limit test settings:

- a) Limit test: OFF
- b) Limit line display: OFF
- c) Failure sound warning: OFF

12) Limit table settings:

- a) Type: OFF
- b) Start excitation: 9kHz
- c) Stop excitation: 9GHz
- d) Beginning response: -100dB
- e) End response: 100dB

13) Time domain conversion settings (option):

- a) Time domain conversion: OFF
- b) Transform mode: bandpass
- c) Transition start: -5ns
- d) Transform stop: 5ns

4.1 Reset Analyzer

- e) Kaiser window beta factor: 6.0
- f) Gate state: closed
- g) Gate start: -5ns
- h) Gate stop: 5ns
- i) Gate shape: standard
- j) Gate type: bandpass

14) Global display settings:

- a) Trace state: ON
- b) Frequency/Excitation: ON
- c) Cursor Reading: ON
- d) Status bar: OFF

4.1.2 User Define State

The analyzer can be reset to a known default state or to a user-defined state. By default, the analyzer is reset to the default state, which can be set to reset the analyzer to a user-defined state. The specific steps for setting the user reset state are as follows:

- 1) Menu path [SYS] → [Define User Status...] to display the Define User Reset Status dialog box.
- 2) Click the [Enable RSTate] check box.

NOTE: a) If [Save LastState as RSTate] is checked, the last state before exiting the program will be saved as the User Reset Status;

b) By clicking the [Save CurState as RSTate] button, the analyzer saves the current instrument settings as the User Reset Status;

c) If you want to use an existing status, click the [Load ExisFile as RSTate] button, select the Status File ON in the Open dialog box, and the analyzer will take the selected file as the User Reset Status file.

4.2 Measurement Parameter Selection



Figure 4.1 Define user resetting state

4.1.3 Reset Analyzer

Menu path: click **[Reset]** in right menubar

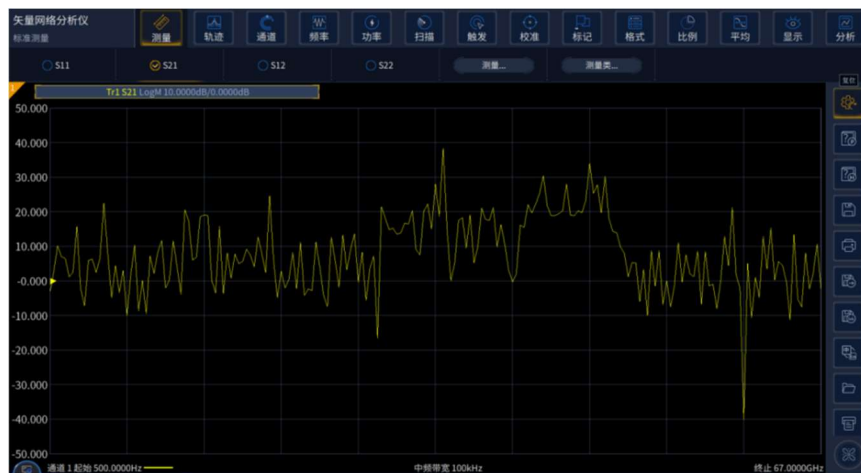


Figure 4.2 Reset analyzer

Tips

【Reset】 shortcut key

There are [RST] shortcut keys in the Function keypad and Shortcut menubar.

4.2 Measurement Parameter Selection

The 3657 series vector network analyzer can be set up to measure the electrical

4.2 Measurement Parameter Selection

characteristics of a device using the following parameters:

- S-parameter (fixed ratio)
- Arbitrary ratio (custom ratio measurement)
- Non ratio power measurement (absolute power measurement)
- **S Parameter**98
- **Differential Balance S-parameter**104
- **Arbitrary Ratio**104
- **Non-ratio Power Measurement**105
- **Chang Trace Measurement Type**105

4.2.1 S-parameter

1) Overview of S-parameters

S-parameter (scattering parameter) is used to describe how a device changes the input signal. It describes the reflection and transmission characteristics of the DUT. The S-parameter represents the ratio relationship between two complex vectors containing amplitude and phase information in a conventional numerical arrangement: $S_{\text{output input}}$. Output refers to the output signal port number of the DUT; input refers to the input signal port number of the DUT. The analyzer has four test ports to measure single-, dual-, three- and four-port devices.

For example, four S-parameter measurements can be made simultaneously when a dual-port device is connected to Port 1 and Port 2.

The four S-parameters of the dual-port device at this point are S11, S12, S21, and S22. The S-parameters are further illustrated in Figure 4.3, where

- a represents the stimulus signal input to the DUT
- b represents the reflected and transmitted signal (response signal) of the DUT

The S-parameter is a complex linear value, and its measurement accuracy depends on the specifications of the calibration kit and the measurement connection technique used, but also on the termination of the non-measurement port (the port that is not excited).

2) Application of S-parameters

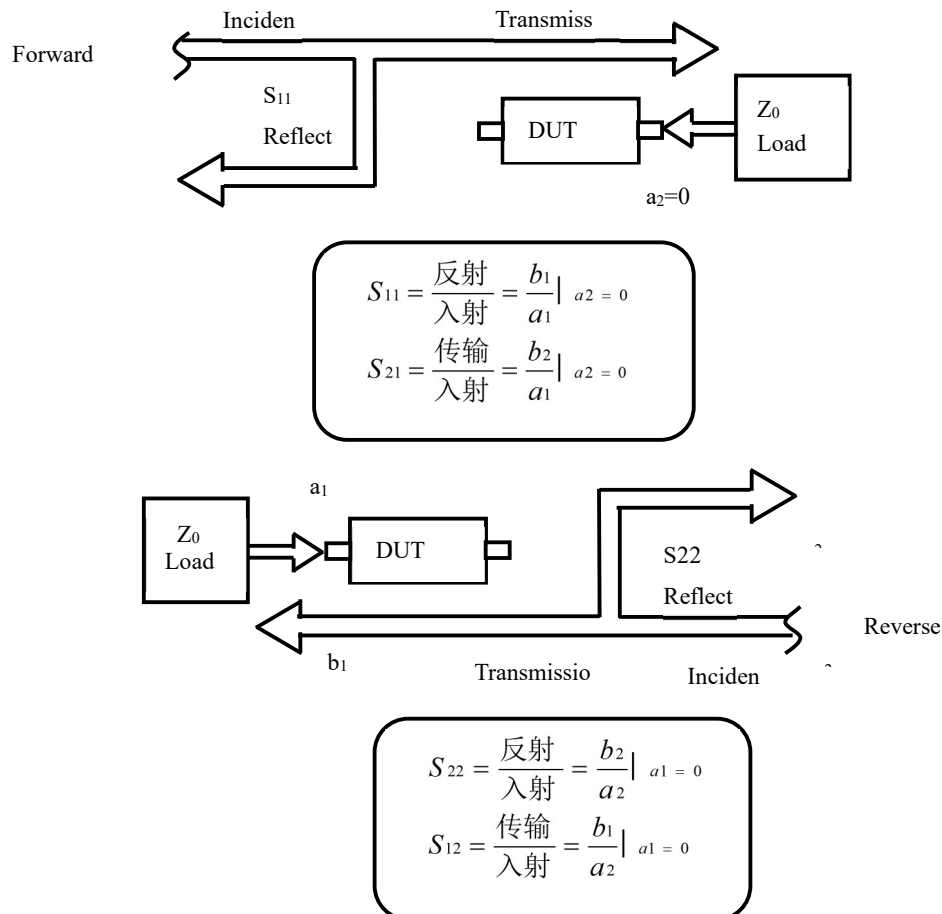
The following parameters can be measured with the S-parameter:

- a) Reflection measurement: SXX (X=1,2,3,4)
 - i. Return loss
 - ii. Standing-wave ratio (SWR)
 - iii. Reflection coefficient
 - iv. Impedance
 - v. S11, S22, S33, S44
- b) Transmission measurement: SXY (X=1,2,3,4; Y=1,2,3,4; X≠Y)
 - i. Insertion loss

4.2 Measurement Parameter Selection

- ii. Transmission coefficient
- iii. Gain
- iv. Group Delay
- v. SXY

The following is an example of S-parameters for Port 1 and Port 2 measurement



of a dual-port device:

Figure 4.3 S-parameter definition

3) New S-parameter measurement trace

Menu path: [Trace] → [New Trace] to click S-parameter in the dialog box.

4.2 Measurement Parameter Selection



Figure 4.4 S-parameter definition

4) New Trace dialog box

- a) **Parameter selection zone**
Used to select the new S-parameter measurement trace to be created.
- b) **[Receiver] button**
Open the dialog box to create an arbitrary ratio or non-ratio power measurement trace.
- c) **[Balance Parameter] button**
Open the dialog box to create a new balance parameter measurement trace.
- d) **[Ch SEL] box**
Used to select the channel where the new trace is located.
- e) **[Create in New Win] check box**
Create a trace in a new window when it is checked, and create a trace in the currently active window when it is unchecked.

4.2.2 Differential Balance S-parameter

1) Overview of differential balance S-parameter

Conventional RF microwave devices are single-port type, i.e., single input and single output, and the signals on the input and output interfaces have a common reference ground plane, as shown in Figure 4.5.

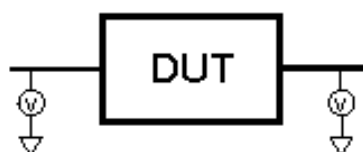


Figure 4.5 Single-port device

4.2 Measurement Parameter Selection

However, with the advent of advanced MMIC integrated circuits, more and more RF circuits are being designed in the form of differential balance. The differential balanced clock rate of the backplane in computers and servers has reached hundreds of Gbps. Such a high rate must also be considered according to RF and microwave devices.



Figure 4.6 Balanced device

Balanced devices have either a two-port input or output. The signal transmitted by the balanced device is the difference or average of the levels between the two ports. The two input ports or the two output ports refer to each other rather than to the ground, as shown in Figure 4.6.

Ideally, when a differential mode signal with equal amplitude and 180 degree phase difference is added to the input of the differential balanced device, the output is also a differential mode signal. This mode of operation is called "differential mode/differential mode". The ideal differential transmission line will not transmit signals of equal amplitude and same phase, i.e., common mode signals, and has good suppression of common mode interference. In fact, the differential transmission line input and output signals cannot be ideal, and common mode signals referenced to ground are present in both the input and output signals. The mode of operation in which a common mode signal is obtained by stimulus of a differential mode signal is called "differential mode/common mode". If the input signal contains a common mode signal, the differential mode and common mode signals will also be excited, and the corresponding modes of operation are "common mode/differential mode" and "common mode/common mode" respectively. The "common mode/differential mode" will introduce noise into the output differential mode signal, so the ability of the differential transmission line to suppress the generation of differential mode signal excited by the common mode signal will be an important indicator to judge the performance of the device. The traditional S-parameter can not distinguish the differential mode signal from the common mode signal, nor can it reflect the transmission of different modes of the differential transmission line and the conversion characteristics of different modes. Therefore, it is impossible to accurately measure the performance of a differential balanced device. In order to completely characterize a differential balanced device, it is necessary to know its response under differential and common mode stimulus, as well as the mode transition information under these two stimulus. Taking the 4-port balance parameters as an example, the mixed mode S-parameter matrix can be used to completely characterize its characteristic index.

$$S_{mm} = \begin{bmatrix} S_{dd11} & S_{dd12} & S_{dc11} & S_{dc12} \\ S_{dd21} & S_{dd22} & S_{dc21} & S_{dc22} \\ S_{cd11} & S_{cd12} & S_{cc11} & S_{cc12} \\ S_{cd21} & S_{cd22} & S_{cc21} & S_{cc22} \end{bmatrix}$$

4.2 Measurement Parameter Selection

The mixed-mode S-parameter is expressed in the form of Sabxy, with the first two subscripts indicating the modes of the response and stimulus signals, respectively, d for the differential-mode signal, c for the common-mode signal, and the last two digital subscripts indicating the ports of the response and stimulus, respectively. The upper left quadrant of the matrix represents the differential mode response of the transmission line under differential mode stimulus, and the lower right quadrant represents the common-mode response of the transmission line under common-mode stimulus; the lower left quadrant of the matrix represents the common-mode response of the transmission line under differential mode stimulus, and the upper right quadrant represents the differential mode response of the transmission line under common-mode stimulus. These two quadrants describe the mode conversion information of the differential transmission line.

2) Method to set differential balance S-parameter

Steps for setting the trace are shown below:

Menu path: [Trace]→[New Trace]→[Bal PARAM] to display the New Trace dialog box.



Figure 4.7 Trace creation for differential balance S-parameter

3) Balanced device topology setting method

Steps for setting the balanced device topology are as follows:

Path: Click **[New Trace]** → **[Balanced Parameters]** → **[Configuration]** to display the DUT Configuration dialog box.

The logical port is for describing the mapping relationship of the physical test port of the vector network analyzer in the new balance test.

- Any two physical test ports can be mapped to a balanced logical port;
- Any physical test port can be mapped to a single-ended logical port

4.2 Measurement Parameter Selection



Figure 4.8 Differential Balanced S-parameter Topology Settings dialog box

The four-port vector network analyzer supports four kinds of balanced device topologies, which can be set in the dialog box shown in the figure and are described below:

a) Balanced/balanced (two differential balanced logical ports and four 3657 series physical ports)



b) Single-ended/balanced (two logical ports and three 3657 series physical ports)

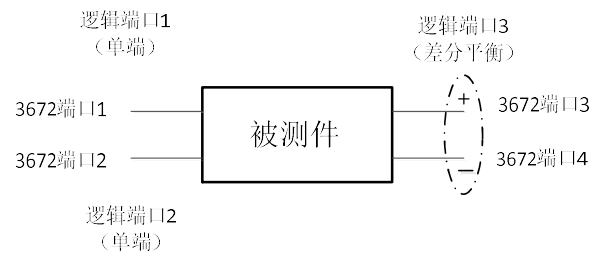


c) Balanced/single-ended (two logical ports and three 3657 series physical ports)



d) Single-ended - single-ended/balanced (three logical ports and four 3657 series physical ports)

4.2 Measurement Parameter Selection



4.2.3 Arbitrary Ratio

Arbitrary ratio allows the selection of input and reference signals from A, B, C, D, R1, R2, R3, and R4 receivers for ratio measurements.

1) New arbitrary ratio measurement trace

Menu path: **[Trace]→[New Trace]** to click **Receiver Parameter** in the dialog box



Figure 4.9 S-parameter definition

4.2.4 Non-ratio Power Measurement

Non-ratio power measurement parameters can be used to measure the absolute power of A, B, C, D, R1, R2, R3, and R4 receivers. Non-ratio power measurements cannot be made for phase, group delay, or any other open averaging function.

1) New non-ratio power measurement trace

Menu path: **[Trace]→[New Trace]** to click **Receiver Parameter** in the dialog box

2) New Trace dialog box (arbitrary ratio and non-ratio power measurement)

- a) **[Activate]** check box
Create a new trace when checked.
- b) **[Numerator]** zone
Select the numerator for arbitrary ratio measurement or the measurement

4.2 Measurement Parameter Selection

receiver for non-ratio power measurement.

c) **[Denominator] zone**

Select the denominator for arbitrary ratio measurement; 1 is selected for non-ratio power measurement.

d) **[SRC Port] box**

Select the source signal output port of the analyzer.

e) **[Ch SEL] box**

Select the channel to which a new trace is added.

f) **[Create in New Win] check box**

Create a trace in a new window when it is checked, and create a trace in the currently active window when it is unchecked.

g) **[Create Win AUTO] check box**

When checked: If the number of traces in the window is more than 8, the excess traces will be created automatically under a new window.

When it is unchecked: If the number of traces in the window is more than 8, the excess traces will not be created.

4.2.5 Change Trace Measurement Type

In the analyzer, if you need to set and modify a trace, you must make it the currently active trace.

1) **Change the active state of the trace**

Click the Trace Status button in the window to make the corresponding trace the currently active trace.

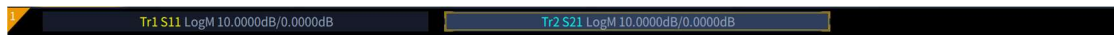
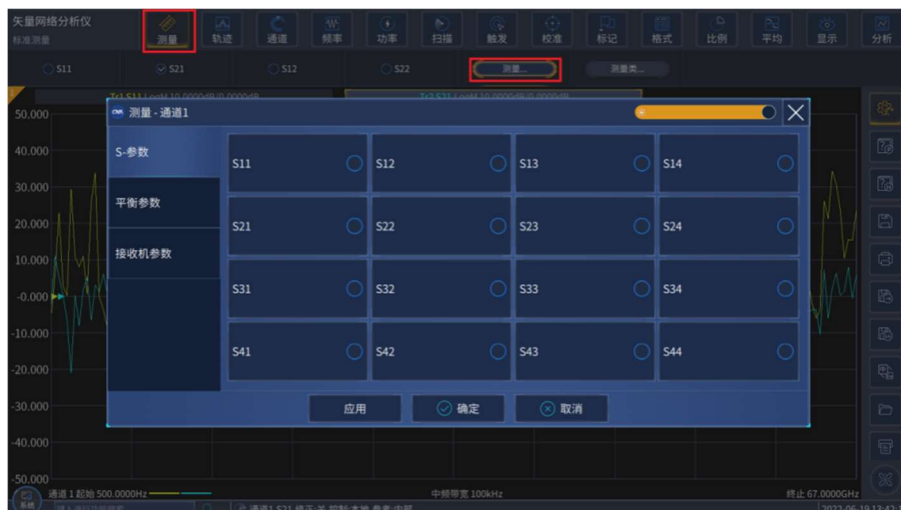


Figure 4.10 Changing the active state of the trace

2) **Change the measurement parameters of the currently active trace**

Menu path: Click **[Meas]** to display the **Measurement** submenu. as shown in Figure 4-7.



4.3. Set Frequency Range

Figure 4. 11 Set the measurement type of S parameter of current active trace

4.3. Set Frequency Range

Frequency range: 10MHz~13.5GHz/26.5GHz/43.5GHz/50GHz/67GHz

Frequency resolution: 1Hz

1) Frequency range can be set in the following two methods:

- Specify start/stop frequency.
- Specify center frequency and frequency span

2) Set the start and stop frequencies

Menu path: **[Freq]** in top menubar



Figure 4.12 Set the start and stop frequencies

3) Set center frequency and frequency span

Menu path: **[Freq]** in top menubar



Figure 4.13 Set center frequency and frequency span

4.4 Set the Power Level of Signal

Power level refers to the power level of the source output signal of the analyzer test port. The indicators of the power level that can be set at the 3657 series vector network analyzer port are as follows:

Table 4.2 Source output signal power level index

Frequency range:	Source power range (dBm)	Configuration
9kHz~9GHz	-25 ~ +20	Standard configuration

1) Set the power level

Menu path: **[Power]** in top menubar.

4.4 Set the Power Level of Signal



Figure 4. 14 Set the power level

2) Turn off port power

Menu path: **[PWR]** → **[PWR Stat ON/OFF]** or switch the power on/off by changing the **[PWR ON/OFF]** check box in the Power Setting dialog box.

3) Set attenuation manually

Menu path: Click **[Power]** → **[Power and Attenuators...]** to display **Power and Attenuators** dialog box ;

Click to uncheck the **Auto Attenuator** check box. Click the **Attenuator** input box and enter the attenuator value.

4) Set power slope

Menu path: Click **[Power]** → **[Power and Attenuators...]** to display **Power and Attenuators** dialog box ;

Click to check the **[Power Slope]** check box. Click the **Slope** input box to enter the power slope value.

5) Power dialog box

a) [Power ON/OFF] check box

When it is checked, the port normally outputs power, and when it is unchecked, the port output power is turned off.

b) [Power-Port 1]

Select to set the power level of port 1.

c) [Power Slope] check box

4.4 Set the Power Level of Signal

The power slope function can be enabled by checking the **[Power Slope]** check box and entering the slope value in the **[Slope]** input box. The power slope is intended for compensating for the increased power loss of the cable and test fixture when the frequency increases.

- When the power slope function is enabled, the output power of the test port will increase (or decrease) as the sweep frequency increases.
- The power slope is expressed in dB/GHz.
- The setting range of power slope is -2 ~ +2.

6) Power and Attenuators dialog box**a) [PWR ON/OFF] check box**

When it is checked, the port normally outputs power, and when it is unchecked, the port output power is turned off.

b) [Port Powers Coupled] check box

Check the **[Port Powers Coupled]** check box by default, and the power level of the 2 ports of the analyzer is set the same. But for some measurement applications, different power levels may be required for each port. For example, to measure the gain and reverse isolation of a high gain amplifier, the power of each port of the analyzer must be set separately because the power required by the input port of the amplifier is much lower than that of the output port. When the **[Port Powers Coupled]** check box is unchecked, the analyzer allows the power level of each port to be set separately.

c) [Status bar]

The state is set to **[Auto]** by default, and in this case, the port power ON/OFF status needs to be changed according to the measurement requirements: **[ON]** means that the port power is always on, and **[OFF]** means that the port power is always off.

d) [Port Power] column

It is used to set the power level of port.

e) [Auto Attenuation] column

When the check box is checked, any power level within the allowable range of analyzer can be set. When the check box is unchecked, the source power and attenuator value can be set manually for a certain measurement, such as measurement of reflection amplifier (oscillator or unstable amplifier of certain state) measurements, in which a very good source impedance matching (i.e. better than 20dB RL) is required within a wide range.

f) [Attenuation] column

It is used to set the attenuator value.

g) [Amplitude Stabilization Mode] column

It is used to set the leveling modes among **[Internal]**, **[Receiver]** and **[Open Loop]**. **[Open Loop]** means no leveling is performed.

h) [Power Slope] check box

4.5 Set Sweep

The power slope function can be enabled by checking the **[Slope]** check box and entering the slope value in the **[Slope]** input box. The power slope is intended for compensating for the increased power loss of the cable and test fixture when the frequency increases.

- When the power slope function is enabled, the output power of the test port will increase (or decrease) as the sweep frequency increases.
- The power slope is expressed in dB/GHz.
- The setting range of power slope is -2 ~ +2.

i) Attenuator settings and power range

The analyzer is adopted with a programmable attenuator to cover the whole power range, which, by adjusting the power level of the DUT without changing the power level on the reference path of the analyzer, improves the accuracy and characteristics indicator of the source output signal as well as the source matching accuracy,

At the power point where the measurement calibration is performed, a completely accurate error correction is possible. In case of change of power level at the same attenuator setting as the measurement calibration, the ratio measurement can achieve better error correction accuracy than the ratio measurement.

4.5 Set Sweep

Sweep is the process of measuring continuous data points with stimulus values in a specified order.

● Overview of Sweep Type.....	109
● Set Sweep Type.....	110
● Sweep Time.....	113
● Sweep Setting.....	114

4.5.1 Overview of Sweep Type

The network analyzer supports the following six sweep types.

1) Linear frequency

This is the default sweep type of the instrument, with linear and continuous coverage of the entire frequency range.

2) Logarithmic frequency

At the logarithmic frequency setting, the source frequency is incremented in logarithmic steps with the same frequency ratio at two adjacent frequency points.

4.5 Set Sweep

3) Power sweep

Power sweep is performed in the point frequency, and for power sweep, the maximum sweep range allowed to be set is 45dB, and the default range of power sweep is -25dBm~20dBm.

4) Fixed

The point frequency sweep mode sets the analyzer to a single sweep frequency, sampling the measurement data precisely and continuously at intervals determined by the sweep time and the measurement points, and displaying the variation of the measurement data over time.

5) Segment sweep

The segment sweep setting initiates a sweep consisting of multiple segments, each of which can be independently defined in terms of power level, IF bandwidth and sweep time. Once calibration has been completed on all segments, a calibrated measurement can be made on one or more segments. The segments are defined in increasing order of frequency, and the frequency ranges cannot overlap. The power level of all segments must have the same attenuator setting to prevent damage due to frequent attenuator switching. When the currently defined segment has a different attenuator setting than the defined segment, the analyzer automatically changes the power level and attenuator setting of the defined segment.

6) Phase sweep

Sweep the phase of one or more sources with respect to another source, measuring -360° to +360°.

4.5.2 Sweep Type Settings

1) SetSwpType

Menu path: [Swp]→[Swp TYP].

Select the sweep type in the auxiliary menubar or in the Sweep Type Settings dialog box.

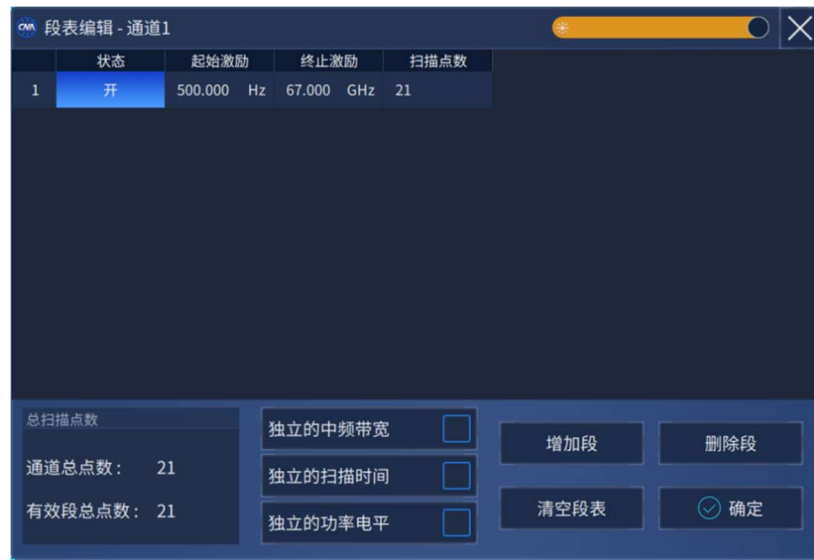


Figure 4. 15 Set the linear frequency sweep type

2) Set segment sweep type

Menu path: [Swp]→[Swp TYP].

Auxiliary menubar: click [Seg Table...].



4.5 Set Sweep

Figure 4.16 Set segment sweep type

Description of dialog box settings:

- [SEP PwrLvl] check box: When checked, each segment can set a separate power level, but it must be the same attenuator setting.
- [SEP IFBW] check box: When checked, a separate IF bandwidth can be set in each segment.
- [SEP Swp TM] check box: When checked, a separate sweep time can be set in each segment.

3) Insert and delete segments

The segment table can be edited only when it is in the displayed state.

Menu path: Click **[Sweep]** → **[Seg Table...]** to display the **Segment** submenu.

Click **[Add Seg]** in the **Segment Table** submenu to insert a new segment before the selected segment. Click **[Empty Seg]** in the **Segment Table** submenu to delete the selected segment, and click **[Clear Segment Table]** to delete the entire segment table.



Figure 4.17 Insert and delete segments

4) Edit the segment table

- Double-click the [Status] box and select ON or OFF to turn the segment on or off.
- Double-click the [Start Excite] box to enter the start frequency of the segment.
- Double-click the [Stop Excite] box to enter the stop frequency of the segment.
- Double-click the [Swp PTS] box to enter the sweep points of the segment.
- Double-click the [PwrLvl] box to enter the power level of the segment (if the Indep PwrLvl option is turned on).
- Double-click the [IFBW] box to enter the IF bandwidth (if the Indep IFBW option is turned on).
- Double-click the [Swp TM] box to set the sweep time (if the Indep Swp TM option is turned on).

5) Set sweep type to phase sweep

Menu path: Click **[Sweep]** → **[Sweep Type]** to display the **Sweep Type** submenu.
Click **[Phase Control...]** in the auxiliary menubar.



Figure 4. 18 Setting to phase sweep

4.5.3 Sweep Time

The analyzer will use the fastest sweep time after the measurement settings is completed, but the sweep time can be increased to meet some specific measurement needs. If the sweep time is set to 0 second, the analyzer will automatically select the fastest sweep time. When the sweep time is greater than or equal to 300 ms, the analyzer will display a sweep indicator to indicate a point-to-point measurement sweep. The measurement indicator is a small upward-pointing arrow that points to the point on the trace where the measurement has just been completed.

1) Set SwpTime

Menu path: **[Swp]** → **[Swp TM]** to display the **Sweep Time** dialog box.

Enter the sweep time directly in the [Swp TM] box. If the button and auxiliary menubar are used for settings mentioned herein, enter the sweep time directly in the input toolbar.

4 Measurement Settings

4.5 Set Sweep



Figure 4.19 Set sweep time

4.5.4 Sweep Settings

1) Sweep settings

Menu path: **[Swp]**→**[Swp Setting...]** to display the **Sweep Setting** dialog box.



Figure 4.20 Sweep setting

2) Sweep settings dialog box

a) [Channel] box

Select the channel to which the sweep setting applies.

b) [Step Sweep] check box

4.6 Trigger Mode

When the Step Sweep mode is selected, the source is tuned to a frequency point, the frequency point is measured after waiting for the specified dwell time, and then tuned to the next frequency point, which can accurately measure the long electrical delay device. When this check box is unchecked, the analyzer may operate in Analog Sweep mode or in Step Sweep mode, depending on the sweep time and IF bandwidth settings.

c) [Dwell Time] input box

Specify the dwell time of the analyzer at each point before acquiring measurement data, suitable only for the Step Sweep mode.

d) [Sweep Delay] input box

It is for setting the waiting time of the analyzer before acquiring measurement data. Sweep display refers to the delay before the measurement of first point.

e) [Fast Sweep] check box

If the [Fast Sweep] check box is checked, it means that the analyzer performs analog sweep and collects data during sweep, thus reducing the sweep waiting time.

4.6 Trigger Mode

The trigger signal is used to let the analyzer conduct measurement sweep, and the trigger setting determines the analyzer sweep mode and the time to stop sweeping and enter the holding state. In terms of trigger function setting, the vector network analyzer provides extremely high flexibility.

- Simple Trigger Settings.....115
- Detailed Trigger Settings.....115
- Trigger Dialog Box.....116

4.6.1 Simple Trigger Settings

The simple trigger setting can only set the trigger mode for the currently active channel.

Menu path: [Trig] to display the **Trigger** submenu.
Click [CONT], [SGL] or [Hold] in the submenu to select the trigger mode.



Figure 4-21 Simple trigger setting

4.6.2 Detailed Trigger Settings

Menu path: [Trig]→[Trig...] to display the **Trigger** dialog box.

4.6 Trigger Mode



Figure 4.22 Trigger submenu



Figure 4.23 Trigger Settings dialog box

4.6.3 Trigger Dialog Box

1) Trigger Settings options

a) Trigger source zone

The trigger source setting determines where the trigger signal of the channel comes from, and a valid trigger signal can only be generated when the vector network analyzer is not sweeping. Three trigger sources are available for the analyzer, including **internal trigger**, **manual trigger**, or **external trigger (measurement trigger)**. Once a trigger source is set, the analyzer will use it as the trigger source of all channels.

➤ **[Internal Trig] (default)**

- 1) Trigger signal is generated automatically by the analyzer.
- 2) The analyzer generates a trigger signal immediately after completing a measurement.

➤ **[Manual Trigger]**

Manual trigger signals can be generated using the following methods:

- 1) Click the [MAN Trig] button in the [Trig] dialog box.
- 2) Click **[Manual Trig]** in the **[Trig]** submenu.

➤ **[External Trig (Meas Trigger)]**

- 1) The trigger signal is input through the External Trigger Input of BNC connector on the rear panel.
- 2) The signal is at TTL level.
- 3) Level trigger: high active or low active, can be set by External Trigger dialog box.
- 4) Pulse width is at least 1us, not longer than the sweep time (pulse width greater than the sweep time may cause multiple triggers).

b) Trigger range zone

4.6 Trigger Mode

The trigger range setting determines which measurement channels of the analyzer will receive the trigger signal. There are two types of trigger range settings: Global and Channel.

- **[Global]**

This is the default setting of the analyzer, and all channels except those in Hold mode receive the trigger signal.

- **[Channel]**

Only the next channel that is not in Hold mode receives the trigger signal. After the current channel finishes measuring, the analyzer automatically selects the next channel, and all channels will be selected sequentially except for the channel in Hold mode. To make this setting valid, set the **trigger source** to **Manual**. Selecting the Channel setting will make the Point Sweep function available.

- **[Activate Channel]**

Only the trigger signals of the currently activated channels will be received.

- c) **Trigger settings zone**

The trigger setting determines how many trigger signals a channel will receive, and there are four channel trigger states.

- **[CONT]**

The channel receives an unlimited number of trigger signals and sweeps continuously.

- **[GRP]**

The channel receives only the specified number of trigger signals in the [GRP] input box, performs the specified number of sweeps, and then enters the Hold mode.

- **[Single]**

The channel receives a trigger signal, conducts sweep once, and then enters the **holding** mode.

- **[Hold]**

The channel does not receive the trigger signal and stops sweeping.

- **[Point] (Trig Mode check box)**

Only when the **Trigger Source** is **Manual** or **External** and the **Trigger Range** is **Channel**, the **Point Sweep** trigger mode can be selected. In **Point Sweep** mode, the channel receives a trigger signal to measure the next data point in the sweep, and the channel will keep receiving the trigger signal until all measurements in the channel are completed, and then trigger the next channel that is in non-Hold mode.

- **[Channel] (Trig Mode check box)**

All traces within the channel will be triggered.

- **[Sweep] (Trig Mode check box)**

Only when the **Trigger Source** is **Manual** or **External** and the **Trigger Range** is **Channel**, the **Sweep** trigger mode can be selected. Traces within a channel that share a source port will be triggered.

4.6 Trigger Mode

- **[Trace] (Trig Mode check box)**
The active traces will be triggered.

2) Measurement and trigger options

a) Main trigger input zone

The property of received external trigger signal is set in the Trigger Input area.

➤ **[Time Delay]****[Delay]**

The delay time is to be input in the delay input box. The analyzer, when receiving an external trigger signal, will perform sweep after this delay time.

➤ **[Source]**

It is for setting the input interface of external trigger signal, including BNC connector and 18# pin of auto test interface.

➤ **[Level/Edge]**

Set the trigger form of receiving external trigger signal, including High Level, Low Level, Rising Edge and Falling Edge.

➤ **[Ready for Trigger Indicator]**

The trigger ready signal can be initiated by the BNC connector and 21# pin of auto test interface, and high level and low level are available for selection of signal level.



Figure 4. 24 Meas Trigger dialog box

3) Aux1/2 Trigger options

a) Auxiliary trigger output zone

➤ **[Enable]**

When the check box is checked, the external trigger output of the corresponding channel is enabled, and when the check box is not checked, the external trigger output is disabled.

➤ **Polarity**

It is for setting the polarity of the trigger output signal by checking the **[Pos Pulse]** radio box or **[Neg Pulse]** radio box.

➤ **[Pos]**

4.7 Set Data Format and Scale

It is for setting the timing to send the trigger output signal by checking **[Before Acq]** radio box or **[After Acq]** radio box.

➤ **[Imp Width]**

This input box is for entering the pulse span for transmitting the trigger output signal.

➤ **[Enable Wait-for-Device]**

When the check box is checked, the vector network analyzer will not start data acquisition until the specific level signal is received from the **aux trigger input (1&2) connectors** on the back panel. This signal indicates that the external device is ready to allow the network analyzer to acquire data. Otherwise, the network analyzer will start the data acquisition without waiting, but a trigger signal will be sent when the network analyzer is ready, which is not used to trigger the network analyzer, but is generated by internal source, manual source or external source.

➤ **[AUX TRIG IN]**

After the network analyzer is ready, a confirmation signal, which may be set to Positive, Negative, High or Low, will be sent to the AUX1&2 connectors.

➤ **[Time Delay]**

After receiving the handshake signal, the network analyzer will not start the data acquisition until the preset delay expires.



Figure 4. 25 AUX1/2 Trigger dialog box

4.7 Set Data Format and Scale

The data format is the way the analyzer graphically displays the measurement data, and the data format that is most appropriate for understanding the information about the characteristics of the DUT should be selected for the measurement. This section describes nine different data formats and how to set the scale for better display of measurement information.

- [Data Format.....](#)120
- [Set Data Format.....](#)122
- [Scale.....](#)123

4.7 Set Data Format and Scale**4.7.1 Data Format****1) Cartesian format**

Seven of the nine data formats display measurement data information in Cartesian coordinates, a display format also known as Cartesian, X/Y, or linear format, which is ideal for displaying frequency response information of the DUT. Cartesian coordinates shown in Figure 4.26 display the following information:

- The X-axis displays the stimulus values (frequency, power or time) in a linear scale by default.
- The Y-axis displays the corresponding response values for different stimulus values.

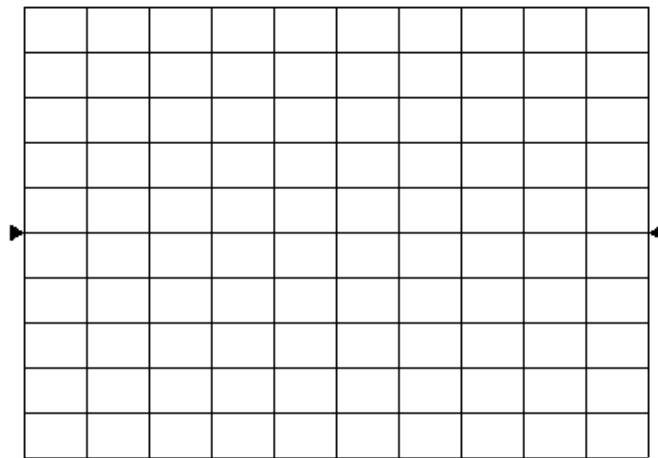


Figure 4.26 Cartesian format

- a) Logarithmic amplitude format
 - Display amplitude information (no phase information)
 - Y axis unit: dB
 - Typical measurements: return loss, insertion loss and gain
- b) Phase format
 - Display phase information (no amplitude information)
 - Y-axis unit: phase (degree)
 - Typical measurements: linear phase deviation
- c) Group delay format
 - Display the transmission time of the signal through the DUT
 - Y-axis unit: time (second)
 - Typical measurements: group delay
- d) Linear amplitude format
 - Display only positive values
 - Y-axis: no unit (U) for ratio measurement, milliwatt (mW) for non-ratio measurement
 - Typical measurements: reflection and transmission coefficients (amplitude value), time domain conversion
- e) VSWR format

4.7 Set Data Format and Scale

- Display the reflection measurement data calculated by equation $(1+\rho)/(1-\rho)$, where ρ is the reflection coefficient
 - Valid only for reflection measurement
 - Y-axis: no unit
 - Typical measurements: VSWR
- f) Real number format
- Display real part of the measurement complex data
 - Similar to linear amplitude format, but can display positive and negative values
 - Y-axis: no unit
 - Typical measurements: time domain, auxiliary input voltage measurements for maintenance purposes
- g) Imaginary number format
- Only display imaginary part of the measurement data
 - Y-axis: no unit
 - Typical measurements: Impedance measurements when designing matching networks

2) Polar coordinate format

As shown in Figure 4. 27, the polar format shown in the above figure contains amplitude and phase information, and the vector value is read in the following ways:

- a) The amplitude value of any point is determined by the displacement from that point to the center point (or zero point). By default the amplitude is linearly scaled and the outer circle is set to a ratio of 1.
- b) The phase value of any point is determined by the angle to the X-axis.
- c) Since there is no frequency information, the frequency information is read by cursor. The default cursor format is real and imaginary, or you can open the Cursor dialog box through the Cursor/Analysis menu and select another format in the Advanced Cursor.

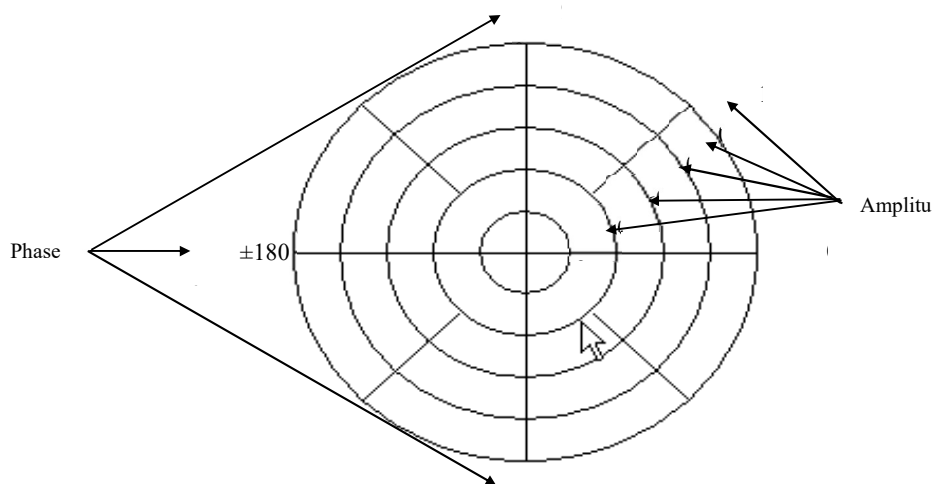


Figure 4. 27 Polar coordinate format

3) Smith chart format

The Smith chart shown in Figure 4.28 below is a tool to map the reflection measurement data of the DUT into an impedance. Each point on the diagram represents a complex impedance consisting of a real resistance (R) and an imaginary reactance ($\pm jX$). The resistance value, reactance value, equivalent capacitance value and inductance value of the DUT can be read through the cursor.

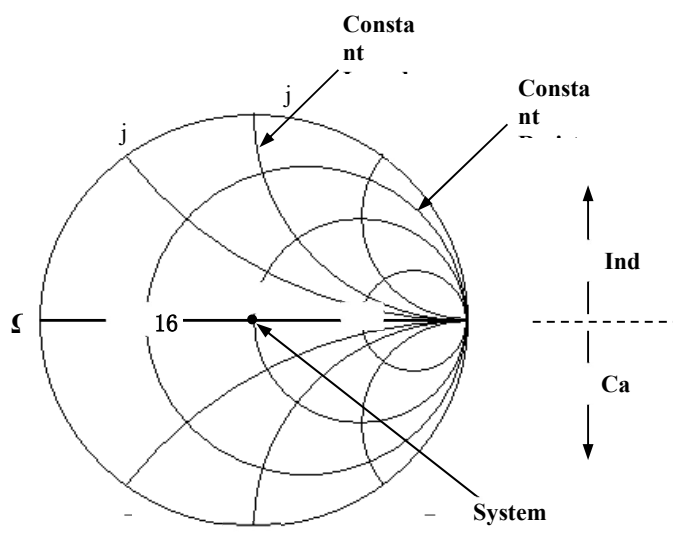


Figure 4.28 Smith chart format

- The central horizontal axis of Smith chart represents pure resistance, and the central point of horizontal axis represents system impedance; The leftmost resistance of horizontal axis is zero, which represents short circuit; The resistance on the rightmost side of the horizontal axis is infinite, which represents open circuit.
- The points on the circle intersecting the horizontal axis on the Smith chart have equal resistance values.
- The points on the arc tangent to the horizontal axis on the Smith chart have the same reactance value.
- The reactance of the upper half of Smith chart is positive, so it is an inductive zone.
- The reactance of the lower half of Smith chart is negative, so it is a capacitive zone.

4.7.2 Set Data Format

Menu path: Click **[Form]** to pop up the **Format** submenu.



4.7 Set Data Format and Scale



Figure 4.29 Set data format

4.7.3 Scale

Scale is used to set the scale of the vertical part of the display grid. In polar coordinates and Smith chart format, it is used to set the full scale value of the outer circle. The setting of scale and format determines the display mode of measurement data on the screen. In logarithmic format, the setting range of scale is 0.001dB/grid-500dB/grid.

1) Set scale

Menu path: [Scale] to pop up the Scale submenu.

Click the corresponding input zone or button to set the appropriate proportion, reference position and reference level.



Figure 4.29 Setting scale

2) Scale Settings dialog box

a) Scale zone

➤ [Scale] box

Enter a scale setting value.

➤ [Auto Scale] button

Click [Auto Scale] to let the analyzer select the vertical scale, so as to better display the activation trace in the vertical grid on the screen (the incentive value is not affected, with only the scale and reference values changed).

- i. The analyzer selects an appropriate scale factor to display the data in 80% of the screen.
- ii. The center value of the trace on the screen is selected as reference value.

➤ [All Auto Scale] button

By clicking the [AutoScale All] button, the analyzer sets the appropriate

4.8 Observe Multiple Traces and Open Multiple Channels

scale for all traces in the window so that they are better displayed in the vertical grid of the window.

b) Reference setting zone

➤ [Ref Pos] button

Reference position refers to the reference line position in the rectangular chart. The position of the lowest bottom line is 0 and that of the top-most line is 10 in the chart, and the default value of reference position is 5.

➤ [Ref Value] button

Reference value refers to the reference line value in the rectangular format or the external circle value in polar coordinate and Smith chart format. The setting range in logarithmic amplitude format is: -500dB to +500dB.

4.8 Observe Multiple Traces and Open Multiple Channels

The analyzer can observe different measurement parameters by creating new trace and window, and can open multiple measurement channels for different measurement settings.

4.8.1 Automatically Set to Multi-window Display

Menu path: Click [Display]→ [Window Layout] and select Single Window/Two Windows/Three Windows/Four Windows.

All currently created traces are displayed in order in the manner of Overlay 1x/Stack 2x/Split 3x/Quad 4x on average.



Figure 4.30 Setting pre-configured measurement

Menu path: [DISP] to pop up the Display submenu.

Select **[Window Layout]** in the Disp submenu.

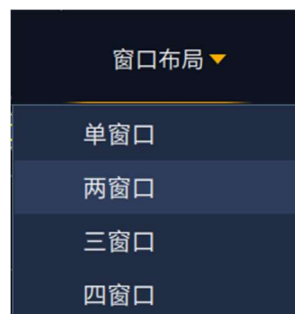


Figure 4.31 Set window arrangement

➤ Overlay 1x

4.9. Set the Analyzer Display

When this window layout is selected, all traces will be displayed in one window in the manner of overlay.

➤ **Stack 2x**

When this window layout is selected, all traces will be displayed in two vertically stacked windows with the number of traces automatically averaged between them, and if it is impossible to distribute the traces uniformly to two windows, the traces in the lower window shall be less.

➤ **Split 3x**

When this window layout is selected, all traces will be displayed in three windows with two on the upper part and one on the lower part, and with the number of traces automatically averaged among those three windows, and if it is impossible to distribute the traces uniformly to the three windows, the traces in the lower window shall be less.

➤ **Quad 4x**

When this window layout is selected, all traces will be displayed in four windows with each window accounting for quarter of the screen, and with the number of traces automatically averaged among those four windows, and if it is impossible to distribute the traces uniformly to the four windows, the traces in the lower windows shall be less.

4.8.2 Manually Add Measurement Traces and Windows

Manually add measurement

4.9. Set the Analyzer Display

We can customize the screen display by showing or hiding the display elements of the screen through the Display menu of the analyzer. These display elements are very useful for observing, setting up and modifying measurements, including Status bar, Toolbar, List, Measurement display, Data Trace, Memory Trace and Titlebar.

● Status Bar	125
● Tool Bar	126
● List	126
● Display Contents	127

4.9.1 Status Bar

通道1 S11 修正:关 控制:本地 参考:内部

1) When the Status bar is turned on, it is displayed at the bottom of the screen and shows the following:

- Currently active channels
- Measurement parameters for active traces
- Error correction status of active trace

4 Measurement Settings

4.9. Set the Analyzer Display

- Control: local or remote
- Reference (REF) signal state: internal or external

4.9.2 Toolbar

- 1) The toolbar allows you to select different quick access options, and to select the quick operations required, click [Disp] → [Toolbar display] in the menubar for setting. The toolbar is on the right side of the interface.



Figure 4.32 Display

4.9.3 List

a) Segment list

	状态	起始激励	终止激励	扫描点数	中频带宽	扫描时间	功率电平
1	关	500.000 Hz	67.000 GHz	21	100.000 kHz	303.450 μs	-5.000 dBm
2	关	67.000 GHz	67.000 GHz	21	100.000 kHz	303.450 μs	-5.000 dBm

Figure 4.33 Segment table

The Segment Table is used to display, set and modify the segment sweep parameters, which include

- State (ON/OFF)
- Start and stop stimulus values
- Sweep Points
- Power Level (each segment can be set individually)
- IFBW (each segment can be set individually)
- Sweep Time (each segment can be set individually)

4.9.4 Display Contents

1) We can display and hide six types of measurement information to observe the current measurement status, including:

- Title
- Trace state
- Frequency/stimulus state
- Cursor display
- Display time state

a) Title

Each window can have a title, which is created through the **Window Title** dialog box, and the entered title is displayed in the upper left corner of the window. If you want to clear the title, delete the title in the **Window Title** dialog box, and then click the **[OK]** button to close the dialog box.

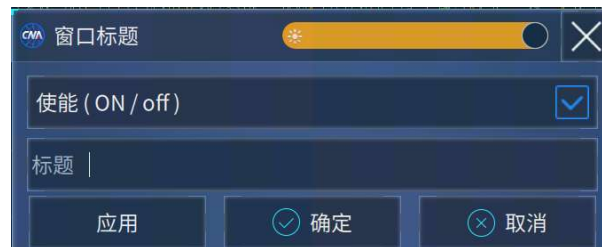


Figure 4.34 Title Input dialog box

b) Trace state

The Trace Status is displayed in the upper left corner of each window on the screen, and it shows the following:

- Measurement parameters
- Format
- Scale
- Reference Value

Click the Trace Status button to make the corresponding trace become the currently active trace in order to set the trace. Right click the button to delete the trace or set the scale, color and linetype of the trace via the Context Menu.

c) Frequency/stimulus state

The frequency/stimulus information is displayed below each window on the screen. It is possible to hide this display for confidentiality purposes, and it displays the following information:

- Channel Number
- Beginning stimulus value
- End stimulus value

4.9. Set the Analyzer Display

d) Cursor display

The cursor display information includes Cursor Reading ON/OFF, One Cursor per Trace ON/OFF, Large Font ON/OFF, Display Position Left, Display Position Down, where the Cursor Reading information is displayed in the upper right corner of each window on the screen, which displays the following information:

- Cursor number
- Stimulus value
- Response value

e) Display time state

When Display Time is turned on, the system time is displayed in the upper right corner of the screen.

2) Trigger display contents

Menu path: Click **[Disp]** and then the **[Display Items]** in the **Disp** submenu.

Select the corresponding content to display or hide by checking the drop-down menu in the menubar or by clicking the auxiliary menubar button.



Figure 4.35 Triggering display content

5 Menus

Menu is the path for you to operate and set the vector network analyzer. The 3657 series vector network analyzer is designed with a complex menu system and various setting dialog boxes under each level of menu. Being familiar with the menu system of the network analyzer is conducive to skilled operation of the analyzer. The menu system of 3657 series vector network analyzer is divided into main menu, level 1 menu, level 2 menu, etc. Those menus of different levels allow you to perform some settings and operations to the analyzer, and for some settings and operations, it is required to activate various setting and input dialog boxes through menus of various levels.

- [Menu Structure.....129](#)
- [Menu Description.....152](#)

5.1 Menu Structure

The menus of different levels are as follows:

- [Meas.....130](#)
- [Trace.....131](#)
- [Chan.....132](#)
- [Freq.....133](#)
- [Power.....134](#)
- [Sweep.....135](#)
- [Trigger.....136](#)
- [Cal.....137](#)
- [Mrk.....138](#)
- [Form.....139](#)
- [Scale.....140](#)
- [Avg.....141](#)
- [Disp.....142](#)
- [Analysis.....146](#)
- [System.....149](#)

5.1.1 Meas

测量
S11
S21
S12
S22
测量...
测量类...

注：后面带省略号（...）的菜单按下之后打开相应对话框，下同。

Figure 5. 1 Meas menu

5.1.2 Trace

轨迹
新建轨迹 ...
删除轨迹 ...
删除激活轨迹
移动轨迹 ...
轨迹标题 ...
轨迹最大化 开 关

Figure 5. 2 Trace menu

5.1 Menu Structure

5.1.3 Chan



Figure 5. 3Chan menu

5.1.4 Freq

频率
起始频率
终止频率
中心频率
频率跨度
频率步进
点频扫描 开 关
点频频率
频率偏移 ...

Figure 5. 4Freq menu

功率
功率状态 ☒ 开 ☐ 关
功率电平
功率扫描 ☐ 关
起始功率
终止功率
衰减器控制
衰减方式 ☐ 手动 ☒ 自动
功率和衰减...

Figure 5.6 Power menu

5.1.6 Sweep



Figure 5.7 Sweep menu

触发
保持
单次
连续扫
组扫描
重扫
手动触发
触发...

Figure 5.8 Trigger menu

5.1.8 Cal



Figure 5.9 Calibration menu

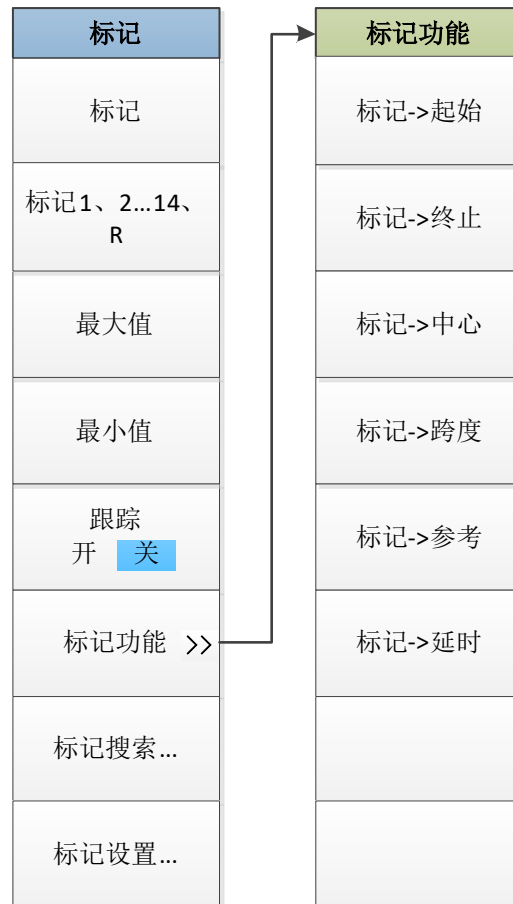


Figure 5.10 Mrk menu

5.1.10 Format

格式
对数幅度
相位
群时延
史密斯圆图
极坐标
驻波比
线性幅度
格式...

Figure 5.11 Format menu

比例	
自动比例	幅度偏移
全自动比例	幅度偏移-斜坡
比例	速度因子
参考值	介质 同轴 波导
参考位置	波导-截止频率
比例耦合 ...	
电延时	
相位偏移	

Figure 5.12 Scale menu

5.1.12 Average

平均
平均因子
平均 开 关
低频段减小中 频带宽 开 关
中频带宽
平滑 开 关
平滑百分比
平滑点数
群时延孔径 ...

Figure 5.13. Avgs menu

5.1 Menu Structure

5.1.13 Display

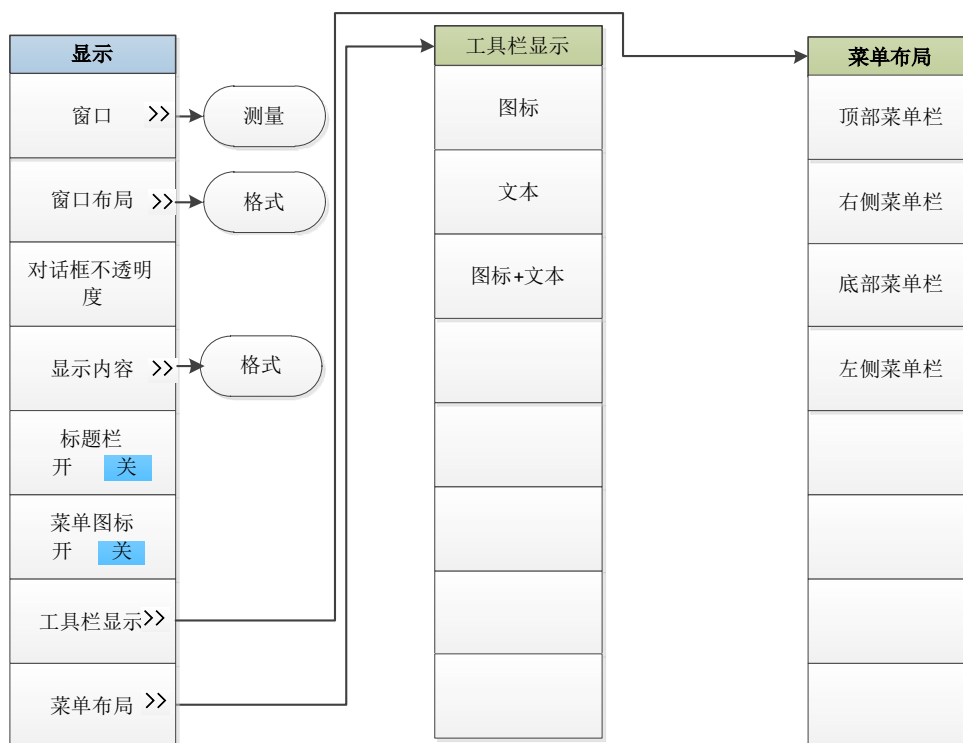


Figure 5.14 Display menu 1



Figure 5.15 Display menu 2

窗口布局
单窗口
两窗口
三窗口
四窗口

Figure 5.16 Display menu 3

显示内容
窗口标题
轨迹状态
频率/激励状态
标记读数
显示时间

Figure 5.17 Display menu 4

5.1 Menu Structure

5.1.14 Analysis

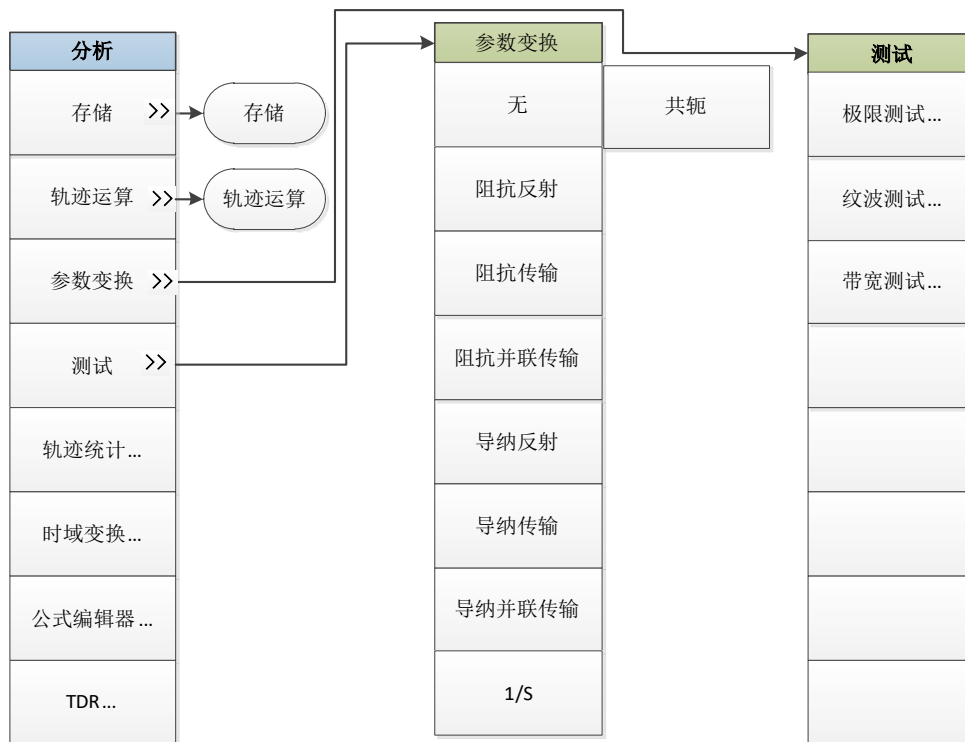


Figure 5.18 Analyzer menu 1

存储
数据->内存
归一化
数据轨迹
内存轨迹
数据和内存
隐藏轨迹

Figure 5.19 Analyzer menu 2

5.1 Menu Structure

轨迹运算
数据+内存
数据-内存
数据*内存
数据/内存
关闭

Figure 5.20 Analyzer menu 3

5.1.15 System

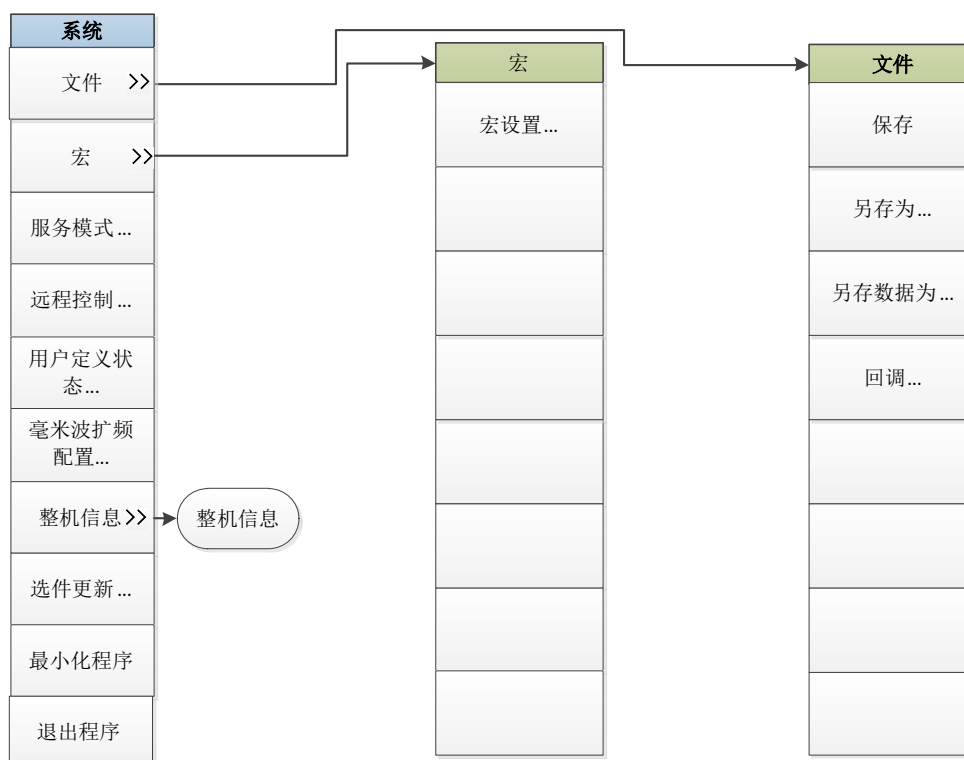


Figure 5. 21 System menu 1



Figure 5.22 System menu 2

1) Main menu

The 3657 series vector network analyzer has 10 main menus.

- **File:** used to save, recall, print test data, minimize the window and exit the program;
- **Trace:** used to perform relevant operations on the trace of the analyzer;
- **Channel:** used to perform relevant operations on the channel of the analyzer;
- **Stimulus:** used to set the stimulus signal of the analyzer;
- **Response:** used to set the signal reception and measurement display of the analyzer;
- **Calibration:** used to calibrate and set the analyzer;
- **Mrk:** this menu is used to operate the cursor system of the analyzer;
- **Analysis:** this menu is used to analyze and make statistics to the test results of the analyzer, and set the time domain function of the analyzer;
- **System:** used to set relevant configuration setting, macro setting, pulse function setting, language setting, and reset and set the analyzer;
- **Help:** used to call up the User' Manual, programming manual and technical support of the analyzer, and view the error log and software version information.

2) Submenu

The submenu of the main menu refers to level 1 menu, the submenu of the level 1 menu refers to level 2 menus, and so on. In principle, when you want to open the menu of a certain level, it is required to open the menu of upper level at first, but for your operation convenience, some commonly-used level 2 menus and their corresponding menus of upper level are displayed side by side.

3) Method to open menus at all levels

There are three ways to open menus at all levels:

- By the menubar: click the main menu to display its primary menu, and click the corresponding menu to display its subordinate menu or pop up the corresponding dialog box.
- By the front panel keys: press the main menu key in the Function keypad on the front panel to pop up the auxiliary menubar to display its primary menu, and click the corresponding menu key in the auxiliary menubar to display its subordinate menu or pop up the corresponding dialog box.
- For convenience, a Shortcut menubar is set on the right side of the auxiliary menubar to display some common menus here. The shortcut menubar and auxiliary menubar can be displayed and hidden by dragging the shortcut bar push-pull button on the left side of the auxiliary menubar.

Tips

For ease of presentation, the screenshots used in this manual are the way to open all levels of menus by the menubar.

5.2 Menu Description

5.2 Menu Description

This section details menu items, functions, parameters and other information.

● Meas.....	152
● Trace.....	154
● Chan.....	155
● Freq.....	157
● Power.....	159
● Sweep.....	161
● Trigger.....	163
● Cal.....	164
● Mrk.....	169
● Form.....	172
● Scale.....	174
● Avg.....	141
● Disp.....	179
● Analysis.....	179
● System.....	183

5.2.1 Meas

Press the [Meas] button on the front panel or click the [Meas] at the top of user interface to pop up a menus related to measurement, including: [S11], [S21], [S12], [S22], [Measurement], [Measurement Class]. The menu items are described below:

5.2.1.1 S11

Function description:

Click this menu to select S11 parameters.

Parameter description:

None

5.2.1.2 S22

Function description:

Click this menu to select S12 parameters.

Parameter description:

None

5.2.1.3 S12

Function description:

Click this menu to select S12 parameters.

Parameter description:

None

5.2.1.4 S22

Function description:

Click this menu to select S21 parameters.

Parameter description:

None

5.2.1.5 Measurement

The menu is used for measurement settings. Click the menu to enter the submenu. The detailed menus include:

Table 5.1 Measurement

Menu
✧ S-parameter
✧ Balanced parameter
✧ Receiver parameter

1) S parameter

Function description:

This menu is for selection of S-parameters.

Parameter description:

None

2) Balanced parameter

Function description:

Click this menu to select balanced parameters.

Parameter description:

None

3) Receiver parameter

Function description:

Click this menu to select receiver parameters such as R1, R2, A and B.

Parameter description:

None

5.2.1.5 Measurement Class

Function description:

5.2 Menu Description

Click this menu to call the multi-function measurement option setting interface.

Parameter description:

None

5.2.2 Trace

Press [Trace] on the front panel or click the menu item [Trace] at the top of the user interface to pop up the menu related to the trace, including [New Trace], [DEL Trace], [SEL Active Trace], [Move Trace], [Trace Title], and [Trace Max ON/OFF]. The menu items are described below:

Set trace

Trace is a series of measurement data points. The trace setting will affect the mathematical operation and display of measurement data. Its setting can be changed only when the trace is active. Click the corresponding trace status button to activate the trace.

5.2.2.1 New Trace**Function description:**

By clicking this menu, the system pops up the New Trace dialog box, and the user is enabled to create different types of traces, including S-parameters (S11, S12, S22, S21) and receivers (R1, R2, A, B).

Parameter description:

None

5.2.2.2 Delete Trace**Function description:**

This menu is for deletion of traces. Specifically speaking, when the menu is clicked, the menu of lower level will be called, and the user can quickly delete the existing data traces.

Parameter description:

None

5.2.2.3 Delete Active Trace**Function description:**

Click this menu to enable the system to delete the currently active traces (one active trace will be deleted each time the menu is clicked, until all traces are deleted).

Parameter description:

None

5.2.2.4 Move Trace**Function description:**

By clicking this menu, the system pops up the Move Trace List, in which all existing windows are displayed. The user can move the currently active trace to any window (select [New WIN] to create a new window). Click [Apply] to move the trace and continue, and click [OK] to move and exit.

Parameter description:

None

5.2.2.5 Trace Title**Function description:**

Click this menu to change the system default name of the currently active curve. After entering the curve title, select the [Enable] check box and click [OK] to complete the trace title change.

Parameter description:

None

5.2.2.6 Track Maximizer ON OFF**Function description:**

By clicking this menu, the system maximizes the currently active trace display and hides other trace curves. Click this menu to maximize traces, when the button displays [Trace Maximizer ON OFF]; click this menu again to restore all traces to normal state, when the button displays [Trace Maximizer ON OFF].

Parameter description:

OFF [OFF | ON].

5.2.3 Channel

Press the [Channel] button on the front panel or click [Chan] menu item on the user interface to pop up a channel-related menu for setting channel-related parameters, including: [Open Channel], [Close Channel], [Close Current Channel], [Select Channel], [Copy Channel], [Hardware Configuration]. The menu items are described below:

Tips**Channels and Traces**

Traces are included in the channels, and the analyzer supports up to 64 channels. Traces in the same channel have the same channel settings. A channel can only change its settings when it is active. Whenever a trace is activated in a channel, the channel is also activated at the same time.

5.2.3.1 Open Channel**Function description:**

By clicking this menu, the system pops up the Open Channel List, in which closed channels are displayed. The user can select the channel to be opened (select [New WIN] to create a new window). Click [Apply] to open the channel and continue, and click [OK] to open the channel and exit.

Parameter description:

None

5.2.3.2 Close Channel**Function description:**

By clicking this menu, the system pops up the Close Channel List, in which all the opened channels

5.2 Menu Description

are displayed. The user can select a channel, click [Apply] to close the channel and continue, click [OK] to close the channel and exit.

Parameter description:

None

5.2.3.3 Close Current Channel**Function description:**

Click this menu to close the channel of the currently selected trace; after the menu is clicked, you can click [Apply] to close the channel and continue, or click [OK] to close the channel and exit.

Parameter description:

None

5.2.3.4 Select Channel**Function description:**

By clicking this menu, the system pops up the Select Channel List, in which all the existing channels are displayed. The user can select the active channels. Click [Apply] to select the channel and continue, and click [OK] to select the channel and exit.

Parameter description:

None

5.2.3.5 Copy Channel**Function description:**

By clicking this menu, the system pops up the Copy Channel List. The user can select the existing channel from the Copy Channel drop-down menu and select the target channel. Click [Apply] to copy the channel and continue, and click [OK] to copy the channel and exit.

Parameter description:

None

5.2.3.6 Hardware Configuration

Select this menu to configure hardware. Click the menu to enter the submenu, as shown in the table below:

Table 5.2 Hardware configuration

Menu

- ✧ RF path configuration
- ✧ Extended interface control

1) RF path configuration**Function description:**

Click this manual to configure the corresponding RF path.

Parameter description:

None

2) Extended interface control

Function description:

Click this menu to configure the input/output pins of extended interface on the rear panel.

Parameter description:

None

5.2.4 Frequency

Press the [Frequency] button on the front panel or click the [Freq] at the top of user interface to pop up a frequency-related menu for setting stimulus-related parameters, including [Start Frequency], [Stop Frequency], [Center Frequency], [Frequency Span], [Frequency Step], [Point Frequency Sweep ON OFF], [Point Frequency], [Frequency Offset]. The menu items are described below:

This menu is used for frequency setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.3 Frequency

Menu

- ✧ Start Freq
- ✧ Stop Freq
- ✧ Center frequency
- ✧ Frequency span
- ✧ Frequency Step
- ✧ Point frequency sweep ON OFF
- ✧ Point frequency
- ✧ Frequency offset

Tips

Frequency unit

All frequency parameters accept parameters in hertz (Hz). Therefore, the digital input must take four frequency units (GHz, MHz, kHz or Hz) as the termination key. When the input is finished, the new frequency value is automatically displayed in the appropriate units.

Notice

5.2 Menu Description

The stop frequency shall not be less than the start frequency

This machine can only sweep upward in the step sweep mode, so the stop frequency shall not be less than the start frequency. If the input start frequency is greater than the stop frequency, then the stop frequency will be equal to the start frequency; if the input stop frequency is less than the start frequency, then the start frequency will be automatically adjusted to the same frequency value as the stop frequency.

5.2.4.1 Start Freq

Function description:

Set the start point of the sweep frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

9kHz/100kHz [9kHz/100kHz ~ 4.5GHz/9GHz].

5.2.4.2 Stop Frequency

Function description:

Set the end point of the sweep frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

4.5GHz/9GHz [9kHz/100kHz ~ 4.5GHz/9GHz]

5.2.4.3 Center Frequency

Function description:

Set the center point of the sweep frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

4.5GHz/9GHz

5.2.4.4 Frequency Span

Function description:

Set the frequency span.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

4.5GHz/9GHz

5.2.4.5 Frequency Step

Function description:

Set the frequency span between two successive sweep points.

5.2 Menu Description

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

4.5GHz/9GHz

5.2.4.6 Point Frequency Sweep ON OFF**Function description:**

Activate the point-frequency state and allow to set the point frequency.

Parameter description:

OFF [OFF | ON].

5.2.4.7 Point Frequency**Function description:**

Set the point frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

4.5GHz/9GHz

5.2.4.8 Frequency Offset**Function description:**

Set carrier frequency offset.

Click this menu to perform settings related to frequency offset. Refer to Annex 3-Frequency Offset Measurement".

Parameter description:

None

5.2.5 Power**Tips****Power unit**

All power parameters is dBm. After input, click [OK] on the panel to complete power setting.

5.2.5.1 Power State ON OFF**Function description:**

Set power to ON/OFF.

Parameter description:

None

5.2 Menu Description

5.2.5.2 Power Level

Function description:

Set the power level of the port output stimulus signal.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

-5dBm

[-55 ~ +15dBm].

5.2.5.3 Power Sweep

Function description:

Change the sweep mode from frequency sweep to power sweep.

In the frequency sweep mode, the frequency is automatically set to the point frequency, and the start power, stop power, sweep step and other settings can be performed.

Parameter description:

None.

5.2.5.4 Start Power

Function description:

Set the start power during power sweep.

Parameter description:

-15dBm

5.2.5.5 Stop Power

Function description:

Set the stop power during power sweep.

Parameter description:

0dBm

[-55 ~ +15dBm].

5.2.5.6 Attenuator Control

Function description:

This function is available only after the programmable step attenuator option is configured. It allows you to manually control the attenuation of the attenuator in manual attenuator mode.

Parameter description:

0dB

5.2.5.7 Power and Attenuators

Function description:

Set power attenuator mode, leveling mode, etc.

Parameter description:

None

5.2.6 Sweep

This menu is used for sweep setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.4 Sweep-segment table

Menu

- ✧ **Swp Time**
- ✧ **Sweep Points**
- ✧ **Swp Type**
- ✧ **Sweep settings**
- ✧ **Segment Table ON OFF**
- ✧ **Segment list**
- ✧ **Pulse Modu Config**
- ✧ **Phase control**

5.2.6.1 Sweep Time

Function description:

Set total time from the start frequency to the stop frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description: (201 points)

198ms[198ms ~ 86400s].

5.2.6.2 Sweep Points

Function description:

Set total number of points from the start frequency to the stop frequency.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

201[1 ~ 32001].

5.2.6.3 Sweep Type

Function description:

Set sweep type.

5.2 Menu Description

The available sweep types include linear frequency, logarithmic frequency, power sweep, point frequency, segment sweep, and phase sweep.

Parameter description:

None

5.2.6.4 Sweep Settings**Function description:**

Set sweep function.

Sweep functions include step sweep, fast sweep, dwell time and sweep delay.

Parameter description:

None

5.2.6.5 Segment Table ON OFF**Function description:**

Click this menu to enable all menu items, allowing the user to set the segment sweep function.

Parameter description:

OFF [OFF | ON].

5.2.6.6 Segment Table

This menu is used for settings related to the segment table. Click the menu to enter the submenu. The detailed menus include:

Table 5.5 Segment table

Menu

- ✧ Add segment
- ✧ Del Seg
- ✧ Empty segment table

Note: The segment table editing function is only available under the standard measurement category. Other measurement categories have independent segment table editing interfaces.

5.2.6.7 Pulse Settings**Function description:**

Set up the pulse measurement function. See Annex 4-Pulse Measurement.

Parameter description:

Pulse width: 33ns ~ 60s

Pulse rise/fall time: 30ns.

5.2.6.8 Phase Control

Function description:

Set the test port phase, including start phase, stop phase and phase ON/OFF, etc.

Parameter description: null.

0°[0°~ 360°]

5.2.7 Trigger

This menu is used for Trigger setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.6 Trigger

Menu

- ✧ Hold
- ✧ Single
- ✧ Continuous sweep
- ✧ Group sweep
- ✧ Number of groups
- ✧ Sweep again
- ✧ Manual trigger
- ✧ Trigger

5.2.7.1 Hold

Function description:

By selecting this menu, the system stops sweeping and remains at the current point frequency.

Parameter description:

None

5.2.7.2 Single

Function description:

By selecting this menu, the system enters the Hold state automatically after one sweep.

Parameter description:

None

5.2.7.3 Continuous

Function description:

By selecting this menu, the system restarts sweeping after one sweep.

Parameter description:

None

5.2 Menu Description

5.2.7.4 Group Sweep

Function description:

By selecting this menu, the system sets the sweep process and then enters the Hold state automatically.

Parameter description:

None

5.2.7.5 Number of Groups

Function description:

The number of sweeps in group sweep mode before the hold state is activated.

Parameter description:

2

[2 ~200000]

5.2.7.6 Re-sweep

Function description:

By selecting this menu, the system restarts sweeping from the start frequency.

Parameter description:

None

5.2.7.7 Manual Trigger

Function description:

Light up this menu through the trigger menu setting, and select this menu to complete a manual trigger.

Parameter description:

None

5.2.7.8 Trigger

Function description:

Select this menu to enter the trigger function setting interface. See 4.6 Trigger Mode for details.

Parameter description:

None

5.2.8 Cal

Press the [Calibration] button on the front panel or click the [Cal] menu at the top of the user interface to pop up a menu related to calibration, including: [Calibration], [Correction ON/OFF], [Interpolation ON/OFF], [Fixture], [Edit Calibration Kit], [Cal Set], [Attribute], and [Power Calibration]. The menu items are described below:

5.2.8.1 Calibration

Function description:

Selecting this menu, the system enters the calibration interface, which provides three types of calibration: wizard calibration, non-wizard calibration and electronic calibration. When measuring, the appropriate calibration method should be selected according to the requirements of measurement type and measurement accuracy. For more information, see "6.3 Calibration Wizard".

Parameter description:

None

5.2.8.2 Correction ON OFF

Function description:

Select this menu to enable the correction function and add calibration data.

Parameter description:

OFF [OFF | ON].

5.2.8.3 Interpolation ON OFF

Function description:

Select this menu to enable the interpolation function.

Parameter description:

OFF [OFF | ON].

5.2.8.4 Fixture

Click this menu to set the fixture function. Click the menu to enter the submenu. The detailed menus include:

Table 5.7 Fixture

Menu

- ✧ Fixture enable
- ✧ Fixture simulation processing sequence
- ✧ Port extensions
- ✧ 2-port fixture de-embedding
- ✧ Matching circuit embedding
- ✧ Port impedance conversion
- ✧ Four-port fixture de-embedding
- ✧ Balanced port impedance conversion
- ✧ AFR
- ✧ Calibration plane manager

5.2 Menu Description

1) Fixture enable

Function description:

Click this menu to enable the fixture simulator function, and only in this state will other functions of the fixture simulator be effective..

Parameter description:

OFF [OFF | ON].

2) Fixture simulation processing sequence

3) Port Extensions

Function description:

Select this menu to move the measurement reference plane electrically. This avoids performing other calibrations.

After that, an additional section of cable is required. The port extension feature can be used to “tell” the analyzer the increased cable length at specific ports; when direct calibration is not possible, port extension can be used to compensate for the time delay (phase offset) caused by the fixture. See the “Setting port extension” section in “5.5 Improving phase measurement accuracy” for the method of setting the port extension.

Parameter description:

None

4) 2-port fixture de-embedding

Function description:

Click this menu to enable the 2-port fixture de-embedding function. For setting method of 2-port fixture de-embedding function, see ["7.9.4 2-port Fixture De-embedding"](#) in ["7.9 Fixture Compensation Calibration"](#).

Parameter description:

None

5) Matching circuit embedding

Function description:

Click this menu to enable the matching circuit embedding function. For setting method of matching circuit embedding function, see ["7.9.3 Matching Circuit Embedding"](#) in ["7.9 Fixture Compensation Calibration"](#).

Parameter description:

None

6) Port impedance conversion

Function description:

Click this menu to enable the port impedance conversion function.. For setting method of port

5.2 Menu Description

impedance conversion, see "7.9.5 Port Impedance Conversion" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

7) Four-port fixture de-embedding**Function description:**

Click this menu to enable the 2-port fixture embedding/de-embedding function.

Parameter description:

None

8) Four-port fixture de-embedding**Function description:**

Click this menu to enable the 2-port fixture embedding/ de-embed function. For setting method of 4-port fixture embedding/de-embedding function, see ["7.9.6 4-port Fixture De-embedding"](#) in ["7.9 Fixture Compensation Calibration"](#).

Parameter description:

None

9) AFR**Function description:**

Click this menu to call the AFR setting window.

Parameter description:

None

10) Calibration plane manager**Function description:**

Click this menu to enter the Cal Plane Manager Settings window.

Parameter description:

None

5.2.8.5 Edit Calibration Kit**Function description:**

Selecting this menu allows users to define and edit their own calibration kit information. To set up the edit calibration kit, refer to ["7.6.3 Edit calibration kit"](#) in ["7.6 Edit calibration kit definition"](#).

Parameter description:

None

5.2.8.6 Calibration Set

Click this menu to set the fixture function. Click the menu to enter the submenu. The detailed menus

5.2 Menu Description

include:

Table 5.8 Fixture

Menu

- ✧ Fixture enable ON OFF
- ✧ Matching circuit embedding
- ✧ 2-port fixture de-embedding
- ✧ Port impedance conversion
- ✧ Four-port fixture de-embedding/embedding

1) Fixture enable ON OFF**Function description:**

Click this menu to enable the fixture simulator function, and only in this state will other functions of the fixture simulator be effective..

Parameter description:

OFF [OFF | ON].

2) Matching circuit embedding**Function description:**

Click this menu to enable the matching circuit embedding function. For setting method of matching circuit embedding function, see "[7.9.3 Matching Circuit Embedding](#)" in "[7.9 Fixture Compensation Calibration](#)".

Parameter description:

None

3) 2-port fixture de-embedding**Function description:**

Click this menu to enable the 2-port fixture de-embedding function. For setting method of 2-port fixture de-embedding function, see "[7.9.4 2-port Fixture De-embedding](#)" in "[7.9 Fixture Compensation Calibration](#)".

Parameter description:

None

4) Port impedance conversion**Function description:**

Click this menu to enable the port impedance conversion function.. For setting method of port impedance conversion, see "[7.9.5 Port Impedance Conversion](#)" in "[7.9 Fixture Compensation Calibration](#)".

Parameter description:

None

5) 4-port fixture de-embedding/embedding**Function description:**

Click this menu to enable the 2-port fixture embedding/de-embedding function. For setting method of 4-port fixture embedding/de-embedding function, see ["7.9.6 4-port Fixture De-embed"](#) in ["7.9 Fixture Compensation Calibration"](#).

Parameter description:

None

5.2.8.7 Properties**Function description:**

This menu is valid after the instrument calibration is completed. Select this menu to display the current calibration properties such as channel, calibration time, calibration type, frequency, sweep points, sweep time, port power, sweep type, etc.

Parameter description:

None

5.2.8.8 Power Calibration**Function description:**

Click this menu to perform calibration and related settings of source power and receiver calibration. For setting method of power calibration, see ["Port Power Calibration"](#) in ["6.3 Scalar Mixer Measurement Calibration"](#) of ["Annex 6 Mixer Measurements"](#).

Parameter description:

None

5.2.9 Mrk

Press the [Marker] button on the front panel or click the [Mrk] menu at the top of the user interface to pop up the menu related to the cursor, including [Marker], [Marker Frequency], [Max], [Min], [Tracking ON OFF], [Marker Functions], [Marker Search], [Marker Setting]. The menu items are described below:

5.2.9.1 Marker

This menu is used for Cursor setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.9 Cursor

Menu

✧ Marker 1/2/3/4/5/6/7/8/9/10/11/12/13/14

5.2 Menu Description

✧ Reference cursor R

1) Marker 1/2/3/4/5/6/7/8/9/10/11/12/13/14

Function description:

This menu is for enabling marker 1/2/3/4/5/6/7/8/9/10/11/12/13/14 function. Read measurement data, search for specific types of values, or change stimulus settings.

Parameter description:

None

2) Reference cursor

Function description:

Select this menu to enable the reference cursor function for relative measurements.

Parameter description:

None

5.2.9.2 Marker Frequency Point

Function description:

Manually enter the frequency point to be marked.

Parameter description:

It is the center frequency in the frequency sweep range by default, which can be set in the range between the start frequency and stop frequency.

5.2.9.3 Max value

Function description:

By selecting this menu, the cursor searches for the maximum measurement data point.

Parameter description:

None

5.2.9.4 Min Value

Function description:

By selecting this menu, the marker searches for the minimum measurement data point.

Parameter description:

None

5.2.9.5 Track ON OFF

Function description:

Click this menu to enable the cursor search tracking function.

Parameter description:

OFF [OFF | ON].

5.2.9.6 Marker Functions

Click this menu to set marker functions. Click the menu to enter the submenu. The detailed menus include:

Table 5.10 Marker functions

Menu

- ✧ Marker → Start
- ✧ Marker → Stop
- ✧ Marker → Center
- ✧ Marker → Span
- ✧ Marker → Ref
- ✧ Marker → Delay

1) Marker → Start

Function description:

Select the menu to set the active marker frequency to start frequency.

Parameter description:

None

2) Marker → Stop

Function description:

Select the menu to set the active marker frequency to stop frequency.

Parameter description:

None

3) Marker → Center

Function description:

Select the menu to set the active marker frequency to center frequency.

Parameter description:

None

4) Marker → Span

Function description:

Select this menu to set the active Δ marker frequency to the frequency span.

Parameter description:

None

5.2 Menu Description

5) **Marker → Ref****Function description:**

Select this menu to set the power value of the active marker to the power reference value.

Parameter description:

None

6) **Marker → Delay****Function description:**

Select this menu to normalize the group delay of the active marker point to 0.

Parameter description:

None

5.2.9.7 Marker Search

Function description:

By selecting this menu, the system enters the Marker Search Shortcut menu.

Parameter description:

None

5.2.9.8 Marker Settings

Function description:

Click this menu to call the Marker Setting dialog box.

Parameter description:

None

5.2.10 Format

This menu is used for Format setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.11 Format

Menu

- ✧ Log Mag
- ✧ Phase
- ✧ Group Delay
- ✧ Smith Chart
- ✧ Unwrap Pha
- ✧ Linear Mag
- ✧ SWR

- ✧ Real part
- ✧ Imaginary part
- ✧ Polar coordinate
- ✧ Pos Phase
- ✧ Impedance
- ✧ Format

5.2.10.1 Log Mag

Function description:

Select this menu to sweep in logarithmic format.

Parameter description:

None

5.2.10.2 Phase

Function description:

Select this menu to display the phase curve.

Parameter description:

None

5.2.10.3 Group Delay

Function description:

Select this menu to display the group delay curve.

Parameter description: null

5.2.10.4 Smith Chart

Function description:

Select this menu to display the Smith chart curve.

Parameter description:

None

5.2.10.5 Polar

Function description:

Select this menu to display the polar curve.

Parameter description:

None

5.2.10.6 VSWR

Function description:

5.2 Menu Description

Select this menu to display the VSWR curve.

Parameter description:

None

5.2.10.7 Linear Mag**Function description:**

Select this menu to display the linear amplitude curve.

Parameter description:

None

5.2.10.8 Format**Function description:**

By selecting this menu, the system pops up the display format class table, where multiple format curves can be selected.

Parameter description:

None

5.2.11 Scale

This menu is used for scale setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.12 Scale

Menu

- ✧ Auto Scale
- ✧ All Auto Scale
- ✧ Scale
- ✧ Reference Value
- ✧ Reference Position
- ✧ Scale Coupling
- ✧ Electrical delay
- ✧ Phase Offset
- ✧ Mag Offset
- ✧ Offset-Slope
- ✧ Velocity Factor
- ✧ Dielectric Coaxial Waveguide
- ✧ Waveguide Cutoff Frequency

5.2.11.1 AutoScale

Function description:

Select the menu to let the analyzer automatically select the vertical scale, so as to better display the activation trace in the vertical grid on the screen (the incentive value is not affected, with only the scale and reference values changed).

The analyzer selects an appropriate scale factor to display the data in 80% of the screen.

The center value of the trace on the screen is selected as reference value.

Parameter description:

None

5.2.11.2 All Auto Scale

Function description:

By selecting this menu, all traces in the window are set to the appropriate scale so that they are better displayed in the vertical grid of the window.

Parameter description:

None

5.2.11.3 Scale

Function description:

Select this menu to enter the scale setting value.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

10dB[0.001dB ~ 500dB].

5.2.11.4 Reference Value

Function description:

Click this menu to enter the setting of the reference value.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

10dB [0.001dB ~ 500dB] (magnitude format).

5.2.11.5 Reference Position

Function description:

Click this menu to enter the setting of reference position, which represents the Y-axis position in the window where the reference value is located. The values from the bottom to the top of the window are 0-10 respectively.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

5.2 Menu Description**Parameter description:**

5[0~10] (magnitude format).

5.2.11.6 Scale Coupled**Function description:**

Click this menu to open the Scale Coupled Setting window.

Parameter description:

OFF: All traces are set with scale values independently

Window: parameters such as the scale, reference value and reference position of the currently active trace in the current window are applied to all traces in that window.

All Windows: parameters such as the scale, reference value, and reference position of the currently active trace of the current window are applied to all traces.

5.2.11.7 Electrical Delay**Function description:**

Select this menu to set the electrical delay value.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

0s[-1000s ~ 1000s].

5.2.11.8 Offset-Slope**Function description:**

Click this menu to set the offset for displayed magnitude and phase.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

Mag Offset: Add/subtract a certain offset to/from the overall magnitude display value.

0dB[-1000dB ~ 1000dB].

Slope: Add/subtract the magnitude display value to/from a certain offset according to the frequency.

0.

Phase Offset: Add/subtract a certain offset to/from the overall phase display value.

0°[-1000° ~ 1000°].

5.2.12 Average

This menu is used for Average setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.13 Average

Menu

- ✧ Avg Factor
- ✧ Avg ON OFF
- ✧ LF IF BW Reduction ON OFF
- ✧ IFBW
- ✧ Smoothing ON OFF
- ✧ Smoothing Percent
- ✧ Smoothing Points
- ✧ Group Delay Aperture

5.2.12.1 Avg Factor

Function description:

Click this menu to set the number of average curve sweeps.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

1[1 ~ 1024].

5.2.12.2 Avg ON OFF

Function description:

Select the menu to enable the **trace averaging** function.

The more average times, the more noise reduction, and the larger dynamic range.

The effect of applying sweep averaging to reduce noise is the same as that of reducing the IF bandwidth.

Parameter description:

OFF [OFF | ON].

5.2.12.3 LF IF BW Reduction ON OFF

Function description:

Click this menu to enable the automatic reduction of the IF bandwidth in low frequency band.

Parameter description:

None.

5.2.12.4 IF BW

Function description:

Select the menu to set IFBW.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

5.2 Menu Description

Parameter description:

1kHz [1Hz ~ 5MHz].

5.2.12.5 Smoothing ON OFF

Function description:

Click this menu to enable the smoothing function.

The smoothing function may reduce the peak-to-peak noise of the broadband measurement data.

The analyzer performs data averaging on a certain part of the display trace, and the number of adjacent data points averaged together is also called the smoothing aperture. This aperture can be specified as the number of data points or as a percentage of the X-axis span.

Parameter description:

OFF [OFF | ON].

5.2.12.6 Smoothing Percent

Function description:

Click this menu to set the smoothing percentage value.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

0 [0 ~ 25%].

5.2.12.7 Smoothing Points

Function description:

Click this menu to set the number of smoothing points.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

0 [0 ~ $N_{\text{Sweep Points}} \times 25\%$].

5.2.12.8 Group Delay Aperture

Function description:

Click this menu to set the number of points/percentage/frequency span of the group delay aperture. This setting is effective for group delay measurement only.

The parameter values can be set and adjusted by mouse/keyboard or rotary knob, numeric keys and step keys on the front panel.

Parameter description:

Number of points: 11 [2 ~ $N_{\text{Sweep Points}}$]

Percent: 5% [1% ~ 100%].

Frequency range: $F_{\text{Frequency Span}}$ [$F_{\text{Two-point Frequency Span}}$ ~ $F_{\text{Frequency Span}}$]

5.2.13 Display

Click [Disp] menu on the user interface to pop up and display setting menu, including: [Window], [Window Layout], [Dialog Transparent], [Display Items], [Titlebar], [Menu Icon], [Toolbar Items] and [Menu Layout].

5.2.13.1 Window

Function description:

Click this menu for window setting, which has four options, namely New Window, Close Window, Close All and Maximize.

Parameter description:

New Window: directly create a new window.

Close Window: Close currently active window.

Close All: Close all opened windows.

Maximize: Maximize the current window.

5.2.13.2 Window Layout

Function description:

Click this menu for quick setting of window layout among four options including Overlay 1x, Stack 2x, Split 3x and Quad 4x.

Parameter description:

Overlay 1x: all current windows are combined into one window for display.

Stack 2x: two windows are arranged vertically after adjustment, and all traces are uniformly distributed into these two windows.

Split 3x: three windows are arranged after adjustment with two in the upper part and one in the lower part, and all traces are uniformly distributed into these three windows.

Quad 4x: three windows are arranged after adjustment with each window accounting for quarter of the screen, and all traces are uniformly distributed into these four windows.

5.2.13.3 Dialog Transparent

Function description:

By setting this parameter, the dialog transparency pops out. The higher the value, the lower the transparency.

Parameter description:

100[10 ~ 100].

5.2.13.4 Display Items

Function description:

When this menu is selected, the display items can be checked in the drop-down menu, including Window Title, Trace State, Frequency/Stimulus State, Marker Readout, Display Time, and Toolbar. For the meaning of each item, refer to software interface introduction in 3.3.

5.2 Menu Description

Parameter description:

None.

5.2.13.4 Titlebar ON OFF

Function description:

Click this option to change the display state of the titlebar.

Parameter description:

ON: display the titlebar;

OFF: hide the titlebar.

5.2.13.4 Menu Icon ON OFF

Function description:

Click this option to change the display state of the menu icon.

Parameter description:

ON: display the menu icon and name;

OFF: display only the menu name, and hide the icon.

5.2.13.4 Toolbar Items

Function description:

Select this menu, and then select toolbar display mode in the drop-down menu.

Parameter description:

Icon Only: the toolbar keys display icon only.

Text Only: the toolbar keys display text only

Icon & Text: the toolbar keys display both icon and text

5.2.13.4 Menu Layout

Function description:

Select the menu and check the menu layout options in the drop-down menu, including Top Menubar, Right Menubar, Bottom Menubar, and Left Menubar.

Parameter description:

Top Menubar: this is the default setting, and the menubar is displayed at the top of the interface.

Right Menubar: the menubar is displayed on the right side of the interface, and the toolbar is automatically displayed at the top of the interface.

Bottom Menubar: the menubar is displayed at the bottom of the interface.

Left Menubar: the menubar is displayed on the left side of the interface, and the toolbar is automatically displayed at the top of the interface.

5.2.14 Analysis

Click the [Analysis] menu on the top of user interface to pop up the menus related to analysis, including [Memory], [Trace Operation], [Parameter Conversion], [Test], [Trace Statistics], [Time Domain

Conversion], [Formula Editor], [TDR]. The menu items are described below:

5.2.14.1 Store

This menu is used for Save setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.14 Save

Menu

- ✧ Data→Memory
- ✧ Normalization
- ✧ Data trace
- ✧ Memory trace
- ✧ Data and Memory
- ✧ Hide trace

1) Data→Memory

Function description:

Select this menu to save the current test curve data to memory.

Parameter description:

None

2) Normalization

Function description:

Select this menu to save the current test curve data to memory and perform data/memory operations.

Parameter description:

None

3) Memory trace

Function description:

Select this menu to display only the memory trace curve.

Parameter description:

None

4) Data and Memory

Function description:

Select this menu to display the data trace and memory trace curves.

5.2 Menu Description**Parameter description:**

None

5) Hide trace**Function description:**

Select this menu to hide the data trace and memory trace.

Parameter description:

None

5.2.14.2 Trace Operation**Function description:**

By selecting this menu, the system can perform four types of mathematical operations on the current active trace and memory trace, including data/memory, data*memory, data+memory, and data-memory.

Parameter description:

None

5.2.14.3 Parameter Conversion**Function description:**

Select this menu and select the parameter conversion mode in the drop-down menu.

Parameter description:

None

5.2.14.4 Test

This menu is used for Test setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.15 Test

Menu

- ✧ Limit Test
- ✧ Ripple test
- ✧ BW Test

1) Limit Test

This menu is used for settings of bandwidth test. Click this menu to open the Limit Test dialog box. For specific settings of this dialog box, refer to 3.5 [Limit Test].

2) Ripple test

This menu is used for settings of bandwidth test. Click this menu to open the Ripple Test dialog box.

For specific settings of this dialog box, refer to 3.5 [Ripple Test].

3) BW Test

This menu is used for settings of bandwidth test. Click this menu to open the BW Test dialog box. For specific settings of this dialog box, refer to 3.5 [BW Test].

5.2.14.5 Trace Statistics

Function description:

Click this menu to open the Trace Statistics dialog box to set the trace statistics state (ON/OFF) and frequency band.

Parameter description:

Checking of statistics menus: Turn on/off trace statistics. Trace statistics include Peak to Peak, Mean, Standard Deviation, Max and Min.

Frequency band setting: you can set the frequency band of trace statistics to Full Span/User Range.

5.2.14.6 Time Domain Conversion

Function description:

Click this menu to enter the time domain conversion window to set time domain conversion, gates, windows and options..

Parameter description:

None

5.2.14.7 Formula Editor

Function description:

Select this menu to enter the Formula Editor interface.

Parameter description:

None

5.2.14.8 TDR

Function description:

Click this menu to enable and configure the TDR measurement mode. For specific operation method, refer to Annex 9 "Introduction to Advanced Time Domain Analysis Functions".

Parameter description:

None

5.2.15 System

Press the [System] button on the front panel or click [System] at the bottom left of the user interface to pop up system-related menus, including [File], [Macro], [Remote Control], [User Define State], [MMW Spread Spectrum Configuration], [Instrument Information], [Option Update], [Minimize] and [Exit]. The

5.2 Menu Description

menu items are described below:

5.2.15.1 File

This menu is used for Configuration setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.16 Configuration

Menu

- ✧ Save
- ✧ Save as...
- ✧ Save Data As...
- ✧ Callback...

1) Save

Function description:

Save the user settings and calibration information data (*.CST) file.

The system saves the current user's setting status and calibration information to the latest user data file. When clicked, the dialog box of whether to overwrite the previous data pops up. Click "Yes" to overwrite and save the data; click "No" to abandon.

Parameter description:

None

2) Save as

Function description:

Save the data file required by the user.

The system saves the data required by the user in different file forms, and the user can set the file name by himself. File types include status, calibration file (*.CST), status file (*.STA), calibration file (*.CAL), data file (*.DAT, *.CTI, *.S1P), graphic file (*.BMP), etc.

Parameter description:

None

3) Save Data As...

Function description:

Save the file required by the user.

The system saves the parameters required by the user in different file forms, and the user can set the file name by himself. File types include data files (*.DAT, *.CTI, *.S1P, *.S2P) and list files (*.PRN), etc.

Parameter description:

None

4) Callback**Function description:**

Call back a saved status file or calibration file.

When clicked, the system will call back a status file or calibration file, which can be of the following types: status, calibration file (*.CST), status file (*.STA), calibration file (*.CAL), data file (*.DAT, *.CTI, *.S1P).

Parameter description:

None

5.2.15.2 Macro**Function description:**

Select this menu to run the macro or pop up the Macro Settings dialog box. For specific functions, refer to 6.12 “Use Micro”.

Parameter description:

None

5.2.9.3 Remote Control**Function description:**

When this menu is selected, the system will pop up Remote Control Settings dialog box, and the user can set user-defined IDN, network socket port number, and command monitoring enable state.

Parameter description:

Network port number: default 5025, value range: 1024~65535.

5.2.15.3 User Define State**Function description:**

Click this menu to call the User Define State dialog box. For specific functions, refer to 4.1.2 “User Define State”.

Parameter description:

None.

5.2.15.4 Instrument Information**Function description:**

Click this menu to enter the Instrument Information interface, including software information, hardware information, statistical information and option updates.

Parameter description:

None.

5.2.15.5 Minimize**Function description:**

5.2 Menu Description

Click this menu to minimize the application of this instrument, and to restore the application display, click the taskbar software icon or press the reset button.

Parameter description:

None.

5.2.15.6 Exit

Function description:

Click this menu to exit the instrument application and display system desktop and other applications.

Parameter description:

None.

6 Optimization & Measurement

Optimize measurement accuracy by adjusting the settings as follows:

● Reduce the Impact of Accessories.....	187
● Improving the Reflection Accuracy of Low-loss Two-port Devices.....	188
● Add Dynamic Range.....	190
● Improve Measurement Results of Electrical Length Devices.....	193
● Improve the Phase Measurement Accuracy.....	194
● Reduce Trace Noise.....	201
● Increase Data Points.....	203
● Improve Measurement Stability.....	205
● Improve Sweep Speed.....	206
● Improve Multi-state Measurement Efficiency	210
● Accelerate Data Transmission.....	212
● Use Macro.....	213

6.1 Reduce the Impact of Accessories

Some of the accessories configured may affect the measurement results of the DUT. Two features are available for the instrument to reduce such effect. To reduce the impact of accessories, you can choose the test configuration method that best suits your needs.

1) Selective response removal gating (for time domain option only)

- a) Gating is a feature of the time domain that allows the analyzer to mathematically remove the response. Reflection response gating and transmission response gating can be set, but their results are different.

- **Reflection response gating** will separate the desired response (e.g. filter return loss) from the undesired response (e.g. adapter reflection or connector mismatch).
- **Transmission response gating** will separate specific paths in a multiplex device with a high electrical length.

To apply the gating in time domain mode, the transform can be switched off and the frequency response of the device can be studied with the gating pulse that is still available. The time filter gating pulse may have the following shapes and characteristics:

- The **gating shape** can remove responses outside the passband.
- The **notch shape** can remove the response from undesired frequency band.

- b) Operation procedure to remove undesired response

Menu path: **[Analysis]→[Time Domain Conversion...]**.

In the **Time Domain Conversion** dialog box, click the **[Time Domain Conversion]**

6.2 Improving Reflection Accuracy of Low-loss Two-port Devices

check box to activate the transform mode. Click the desired mode in the **Transform Mode** zone: Lowpass Impulse, Lowpass Step, Bandpass. To select lowpass, click **[Set Frequency. Lowpass]**. Click **[Gate]**, and then in the **Gate Type** box, click on the drop-down list to select the gate type: bandpass, bandstop. Set the gate time boundary by setting the **Start** and **Stop**, or the **Center** and **Span**.

6.2 Improving Reflection Accuracy of Low-loss Two-port Devices

To ensure the accuracy of reflection measurements under single-port calibration, ports that are not to be measured should be terminated, which is particularly suitable for low-loss bidirectional devices, such as filter transmission bands and cables. Single-port calibration can correct directivity, source matching, and frequency response errors, but can not calibrate load matching errors. Therefore, it is necessary to minimize the load matching error.

Ports that are not to be measured can be terminated as follows:

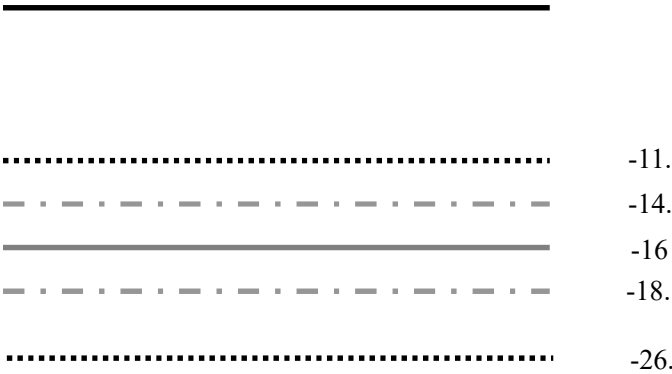
- Connect a high-performance terminal load (which can be a calibration kit) to the DUT port not subject to measurement. The measurement accuracy of this method is close to that of the full two-port calibration.
- Connect the DUT port not subject to measurement directly to the analyzer, and insert a 10dB precision attenuator between the DUT output and the analyzer, which may improve the effective load matching of analyzer to approximately one times the attenuator magnitude, i.e., 20dB.

Tips

Matching load

When measuring devices with high reverse isolation, such as amplifiers, it is not necessary to consider the load matching.

Figure 6.1 shows the measurement uncertainty with/without a 10dB precision attenuator at the output terminal of DUT.



6.2 Improving Reflection Accuracy of Low-loss Two-port Devices

Figure 6.1 Comparison of measurement instability

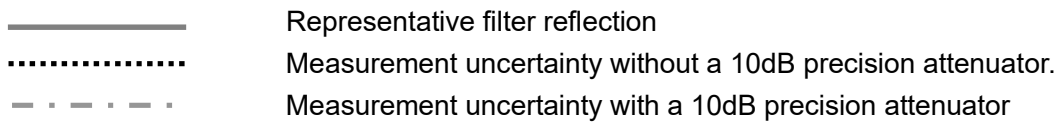


Figure 6.2 shows the accurate reflection measurement of a two-port device. The following calculation demonstrates the improvement of the analyzer's load matching by installing a high-quality 10dB attenuator.

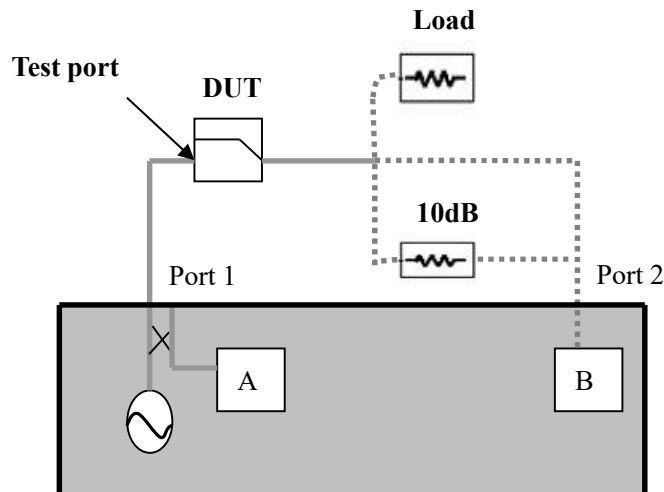


Figure 6.2 Precise reflection measurement of dual-port device

Notice

The corresponding linear values are given in parentheses.

Analyzer: Load matching (NA_{LM}) = 18dB (.126)

Directivity (NA_D) = 40dB (.010)

Filter: Insertion Loss (F_{IL}) = 1dB (.891)

Return Loss (F_{RL}) = 16dB (.158)

Attenuator: Insertion Loss (A_{IL}) = 10dB (.316)

SWR(A_{SWR}) = 1.05 (.024) Return Loss: 32.26dB

Calculation: **Without attenuator With attenuator**

$$\begin{aligned} \square \rho_{NA} &= (F_{IL}) * (NA_{LM}) * (F_{IL}) &= (F_{IL}) * (A_{IL}) * (NA_{LM}) * (A_{IL}) * (F_{IL}) \\ &= (.891) * (.126) * (.891) &= (.891) * (.316) * (.126) * (.316) * (.891) \\ &= .100 &= .010 \end{aligned}$$

$$\begin{aligned} \square \rho_{Attenuator} \quad NA &= (F_{IL}) * (A_{SWR}) * (F_{IL}) \\ &= (.891) * (.126) * (.891) \\ &= .019 \end{aligned}$$

6 Optimization & Measurement

6.3 Add Dynamic Range

$$\text{Error in worst case (E}_{wc}) = \square_{NA} = \square_{NA} \square_{Attn.}$$

$$= .1 = .01.019 = .029$$

$$\text{Uncertainty} = -20\log(F_{RL}) (E_{WC}) (NA_D) = -20\log(F_{RL}) (E_{WC}) (NA_D)$$

$$\text{upper limit} = -20\log(.158) (.100) (.010) = -20\log(.158) (.029) (.010)$$

$$= 11.4\text{dB} = 14.1\text{dB}$$

$$\text{Uncertainty} = -20\log(F_{RL}) - (E_{WC}) - (NA_D) = -20\log(F_{RL}) - (E_{WC}) - (NA_D)$$

$$\text{lower limit} = -20\log(.158) - (.100) - (.010) = -20\log(.158) - (.029) - (.010) = 26.4\text{dB}$$

$$= 18.5\text{dB}$$

Therefore for a low-loss two-port device, an accurate reflection measurement result can be ensured as long as one of the followings is met:

- Connect a high-quality load to the output of the DUT after single-port calibration.
- Install a 10dB attenuator between the DUT output and the analyzer after single-port measurement.
- Directly connect the DUT output to the analyzer after two-port measurement calibration.

6.3 Add Dynamic Range

The dynamic range refers to the delta between the max power and min. measurable power (noise base) that can be entered to the analyzer. To ensure correct and valid measurement, the value of the input signal must be in such range. If the signal amplitude to be measured varies largely (for example, the passband and stopband of the filter), it is necessary to increase the dynamic range. The Figure 6.3 shows the dynamic range for a typical measurement.

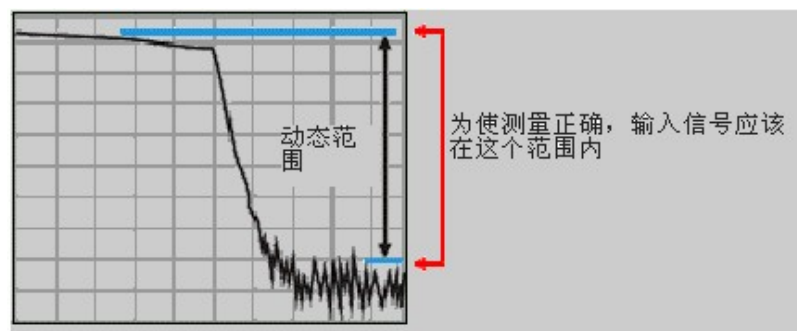


Figure 6.3 Dynamic range

To reduce measurement uncertainty, the dynamic range of the analyzer should be larger than the response of the DUT. For example, accuracy can be improved when the response of the DUT is at least 10 dB above the noise base. The following methods can help increase the dynamic range:

Notice

Dynamic range and measurement time

Since some analyzers are set up with SLI, the increase of dynamic range will lead to an increase in measurement time.

6.3.1 Increase DUT Input Power

- 1) Increasing the input power of the DUT allows the network analyzer to detect and measure the output power of the DUT more accurately.
- 2) The analyzer receiver can cause compression distortion if the input is too high, and the receiver can be damaged if it is high enough.



Warning

Input damage level of the receiver: +15dBm.

Notice

The maximum source output power cannot be used

If the DUT involves gain, the maximum source output power cannot be used, which will cause damage to the receiver.

Menu path: **[Power]→[Power Level]**.

Enter the power value directly in the input toolbar, or click [Power and Attenuation ...] in the dialog box to set it.

6.3.2 Decrease the Receiver Noise Base

Follow the method below to decrease the noise base and increase the dynamic range of the analyzer:

- a) Decrease the IFBW
- b) Use the Sweep Average

Tips

Reduce crosstalk between receivers

When the measured signal is close to the noise floor of the analyzer, crosstalk between the receivers should be reduced.

1) Decrease the IFBW

Effect of IF bandwidth on measurement results is shown in Figure 6.4.

- Reducing the bandwidth of IF receiver can reduce the influence of random noise in measurement. Every 10 times reduction in IF bandwidth can reduce the noise base by 10 dB. However, the narrower IF bandwidth, the longer the sweep time.
- The network analyzer converts the received signal from the RF/microwave band to a lower intermediate or IF frequency of 7.606 MHz. The bandwidth of the IF bandpass filter can be adjusted from 5MHz down to a minimum of 1Hz.
- The IF bandwidth can be set independently for each channel or each segment of the segment sweep.

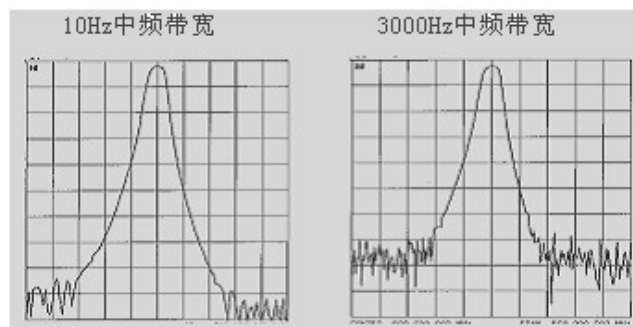


Figure 6.4 Effect of IF bandwidth on measurement results

a) Set IFBW

Menu path: **[Avg]** → **[IFBW]**.

Enter the IFBW in the **Input Toolbox**, or press the arrow buttons to select a value.

2) Increase or change sweep average

a) Increase or change sweep average

- Sweep averaging is a method to reduce the effect of random noise in measurements.
- The analyzer performs calculation to each data point by averaging the results of the same data point over several consecutive sweeps.
- The more average times, the more noise reduction, and the larger dynamic range.

6.4 Improve Measurement Results of Electrical Length Devices

Figure 6.5 shows the effect of sweep average on measurement results.

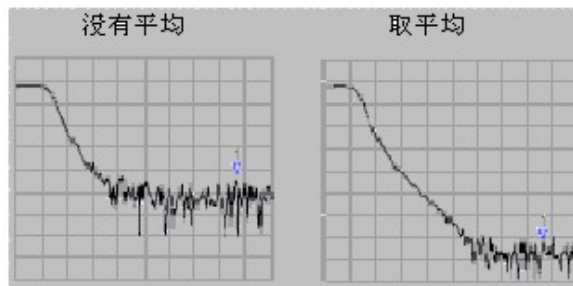


Figure 6.5 Effect of sweep average on measurement results

Tips

Method to reduce random noise

- The measurement averaging can be set for each channel independently.
- The total measurement time increases as the average times increases.
- The general noise can be minimized by both measurement averaging and IFBW.
- To minimize very low frequency noise, the measurement average is more effective than the system bandwidth reduction.

b) Set sweep average

Menu path: **[Avg]**.

Turn averaging on or off with [Avg ON/OFF], directly input the value through the **Average Factor** or press the arrow button to increase or decrease the average times of the analyzer.

6.4 Improve Measurement Results of Electrical Length Devices

The frequency of the signal output from the DUT may not be strictly equal to the frequency of the signal input into the DUT at a given moment, which may sometimes lead to inaccurate measurement results. The following sections describe the causes of inaccurate measurements and the method to compensate inaccurate measurements.

Why does device delay produce inaccurate results?

- In a vector network analyzer, the source and receiver are phase locked together and simultaneously swept over a frequency spacing.
- The signals flowing through the DUT show different colors for different frequencies.
- When the stimulus frequency passes through the DUT, and the analyzer tunes to the new frequency just before the signal reaches the receiver, this will cause inaccurate measurement results.

As shown in Figure 6.6, if the analyzer is measuring a long cable, the frequency of the signal

6.5. Improve the Accuracy of Phase Measurement

at the end of the cable will lag behind the source frequency of the analyzer. If the frequency offset is significant relative to the IF detector bandwidth of the network analyzer (typically several kHz), then the measurement results will be erroneous due to the roll-off characteristics of the IF filter.

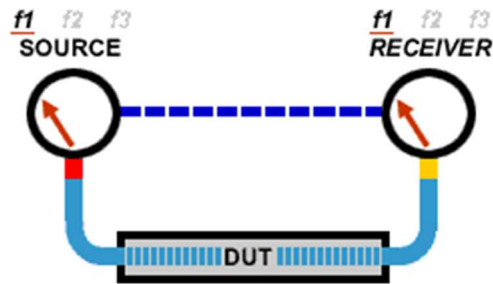


Figure 6.6 Long test cable

The delay in the electrical length equipment can be compensated for by using the following:

1) Reduce sweep speed

The methods to reduce the sweep speed are to increase the sweep time, reduce the IF bandwidth or increase the sweep points.

- a) Increase sweep time

Menu path: **[Sweep]→[Sweep Time]**.

Enter a time value or press the arrow to select a time value.

- b) Decrease the IFBW

Menu path: **[Avg]→ [IFBW]**.

Enter a value directly or press the arrow button to select a value.

- c) Increase the number of sweep points

Menu path **[Swp]→[Swp PTS]**.

Enter a value or press the arrows to select a value.

2) Sweep by step

Change the analyzer mode from analog sweep to step sweep to make the source step through each test point, thus reducing the analyzer sweep speed. It is also possible to set the time that the analyzer stays at each step or the dwell time at each test point.

Menu path **[Swp]→[Swp Setting]**.

Click to check Step Sweep, and in the Dwell Time box, enter directly or use the arrow to select a value for how long you want the analyzer to stay at each test point.

6.5. Improve the Accuracy of Phase Measurement

The accuracy of phase measurement can be increased by using the following characteristics of the analyzer.

6.5. Improve the Accuracy of Phase Measurement

● Electrical Delay.....	195
● Port Extensions.....	195
● Phase Offset.....	198
● Spacing of Frequency Points.....	198
● Specific Operations.....	199

6.5.1 Electrical Delay

- The linear phase drift through the DUT can be compensated by the electric delay feature, so that the linear phase deviation of the DUT is highlighted.
- Electrical delay is a mathematical function that simulates a loss-free transmission cable of variable length.
- An electrical delay independent of each measurement trace can be set.

6.5.2 Port Extension

- 1) After calibration, the measurement reference plane can be moved electrically by using port extension. This avoids performing other calibrations. The functions of port extension are described via the following two scenarios.
 - a) If a calibration has been performed, determine whether an additional section of cable is required in the measurement configuration. The port extension feature is used to "tell" the analyzer the length of cable that has been added to the specific port.
 - b) When the DUT cannot be calibrated directly in the test fixture, the time delay (phase offset) caused by the fixture can be compensated with port extension.
- 2) Use of port extension function.
 - a) If the electrical length of the fixture under test or additional cable is known, simply enter it in the [TM] box;
 - b) If the physical length of the fixture under test or additional cable is known, simply enter in the [DIS] box;
 - c) If both of the above are unknown, the extended reference surface must be available with an open circuit or short circuit breaker instead of the DUT. However, it is generally considered that the new reference surface after removing the DUT is in an open circuit state.
- 3) A suitable port extension value can be obtained as follows:
 - a) Select a calibrated S11 measurement and set the display format to phase format.
 - b) Connect an open circuit or short circuit breaker at the calibration plane and confirm that the displayed phase curve lies around 0° in the measurement frequency range.
 - c) Connect the fixture or transmission cable, replace the DUT with an open circuit or short circuit breaker (removing the DUT is equivalent to an open circuit), and in the Port

6 Optimization & Measurement

6.5. Improve the Accuracy of Phase Measurement

Extension dialog box, adjust the value of the [TM] box or [DIS] box until a flat phase trace is obtained.

- d) If the loss characteristics of the extension are known, it can be compensated in the [LossComp] section by one-dimensional or two-dimensional means.

Notice

Delay

Most short-circuit standards have a non-zero delay. Adjusting the port extension in this way will cause an error equal to twice the delay of short-circuit calibration. You can further determine the appropriate extension value by looking at the definition of the calibration kit.

4) Set Port Extension

Menu path: **[Calibration]** → **[Fixture]** → **[Port Extensions...]**.

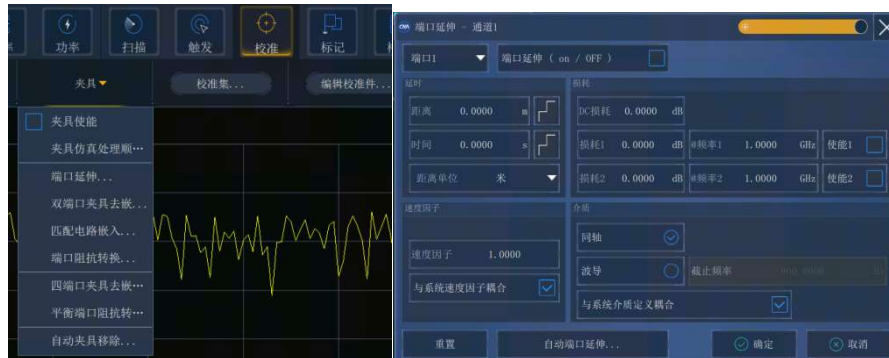


Figure 6.7 Set Port Extension

The Port Extension dialog box is displayed. Click the [Port EXT] check box. The port extension function works for all ports.

- [Port]: Select the current extension port in the drop-down box. The Port Extension setting works for all measurements under the active channel on the current port.
- [TM]: Set the time of port extension delay.
- [DIS]: Set the physical length of port extension delay.
- [DIS Unit]: Set the unit of port extension distance, selectable from meter, foot and inch.
- **[DC Loss]**: set the full band offset value.
- [Loss] [@Freq]: Set the loss value and frequency. When only [Loss 1] is selected, the analyzer compensates using the following equation:

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

When both Loss 1 and Loss 2 are selected, compensation is performed using the following equation:

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

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

- [Velocity Factor]: Set the velocity factor. Note that the velocity factor set here is the same

6.5. Improve the Accuracy of Phase Measurement

as that of the system.

- [COAX]: Set the characteristic of the current port extension as coaxial characteristic.
- [WG], [Cutoff Freq]: Set the characteristics of the current port extension to be waveguide characteristics. The cutoff frequency of the waveguide can be set at this point. Note that the media type set here is the same as that of the system.
- [Reset]: Restore the default state of the system. Note that the port extension state remains unchanged.
- [Auto Extend]: Invokes the Auto Port Extend dialog box.

The Auto Port Extend automatically completes the manual port extension. By connecting the open circuit or short circuit breaker on the extension surface, the system automatically calculates the delay and loss of port extension, and compensates the measuring end face to the extension.

Auto Port Extend steps:

- 1) Connect the added transmission line or fixture, and connect the open circuit or short circuit breaker at the new measuring end face. Normally, not connecting a calibration kit corresponds to an open-circuit state.
- 2) In the Port Extensions toolbar, click **[Automatic Port Extension...]**.
- 3) Click **[Open]** or **[Short]** to start port extension calculation. The calculated delay and loss values are automatically displayed in the toolbar.

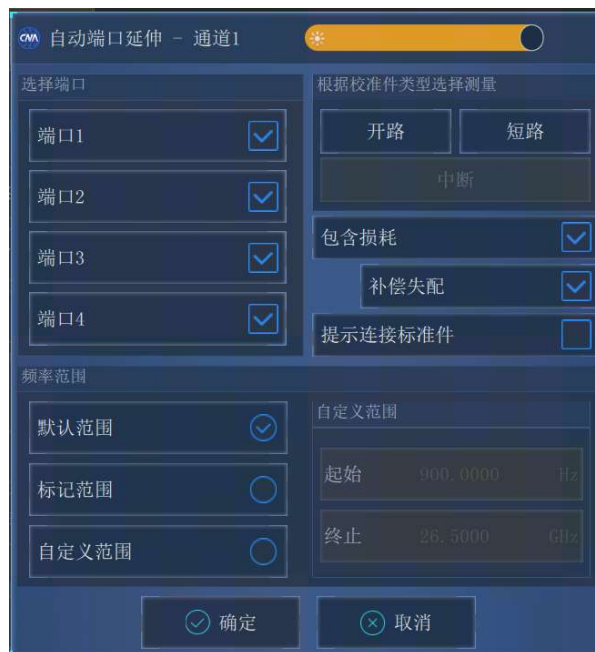


Figure 6.8 Auto port extension

- [Open], [Short]: Set whether the auto extension port is connected to an open circuit or short circuit breaker.
- **Select port**: Select which ports for auto extension calculation.
- [Freq Range]: Set the frequency range of the data collected during the auto port extension calculation.

6 Optimization & Measurement

6.5. Improve the Accuracy of Phase Measurement

[DEF Range]: The same frequency range as the current measurement setting.

[Mkr Scope]: Automatic port extension calculation using the data between the active cursor and the highest frequency.

[Custom Span]: the calculation frequency span input by the user on the right side.

- [Include Loss]: Selected to automatically calculate the loss introduced by the current port extension.
- **[Adjust for Mismatch]**: the error introduced by mismatch will be compensated when this option is selected.
- [Connect CalKit Prompt]: When selected, a prompt box for connecting the calibration kit pops up.
- **[OK]**: Apply the current calculation result to the active channel and exit the dialog box.

6.5.3 Phase Offset

Phase offset is a mathematically adjusted phase measurement with a range of 0-360°. Use this property as follows:

- **Improving the display of phase measurement results.**

This is similar to the method of changing the reference level in amplitude measurement. Change the phase response so that the response is in the center of the screen or in line with the screen.

- **Simulating the phase drift determined in measurement**

For example, if it is known that a cable needs to be added and the length of the cable will bring a certain phase offset to the measurement results, the phase offset value can be increased with the phase offset to simulate the whole device measurement.

1) Set phase offset

Menu path: **[Scale]→[Phase Offset]**.

Set the phase offset value in the input box.



Figure 6.9 Phase offset

6.5.4 Spacing of Frequency Points

- 1) The analyzer samples data at discrete frequency points and then connects each

6.5. Improve the Accuracy of Phase Measurement

sampling point to form a trace on the screen.

- 2) If the phase drift between two adjacent frequency points through the device is greater than 180° , the displayed phase slope appears to be in anti-phase. This is confusing because the data is under-sampled.

If you are measuring the group delay and the phase slope is negative, the group delay will change the symbol. For example, Figure 6.10 shows the measurement results of a surface acoustic wave (SAW) bandpass filter.

- a) The first graph is a measurement of 51 points showing that the group delay is negative, but this is not possible in practice because the response cannot be under a 0-second reference line.
- b) The second graph is a measurement of 201 points showing that the group delay is positive, that is, the response is on the 0-second reference line.

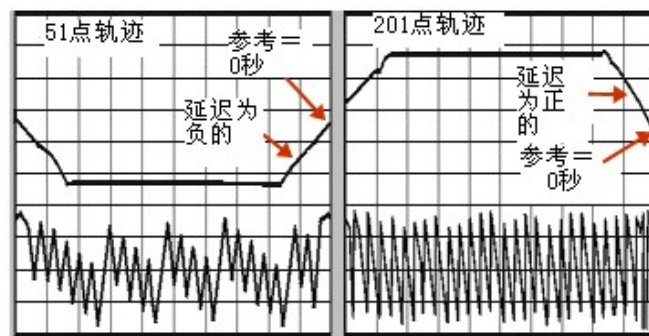


Figure 6.10 Effect of frequency spacing on phase measurement

Tips

Methods to check for confusion

To check for confusion in the measurement, you can reduce the frequency spacing to see whether there is any change in the data displayed on the analyzer. If confusion occurs, it can be avoided by increasing the number of frequency points or decreasing the frequency spacing.

6.5.5 Specific Operations

1) Compensation for linear phase drift

Menu path: **[Scale]**→**[Electrical Delay]**.

In the input box, enter the desired value or press the arrow to select a value.

In the **Velocity Factor** input box, enter a value or press the arrow to select a value between 0 and 1.0.

- 1.0 corresponds to the speed of light in vacuum
- 0.66 corresponds to the typical speed of light in polyethylene

6 Optimization & Measurement

6.5. Improve the Accuracy of Phase Measurement

- 0.70 corresponds to the typical speed of light in PTEF



Figure 6.11 Setting electrical delay

2) Electric Movement Reference Plane

Menu path: **[Cal]→[Fixture]→[Port Extensions...]**.

Check the **[Port Extensions ON/OFF]** check box.

Set the port extension value in the Port Extension dialog box for the port to extend the reference plane and increase the port extension value.

In the **Velocity Factor** box, select a value between 0 and 1.0 as the relative speed of the media of the cable or device. and then click **[OK]** .

- 1.0 corresponds to the speed of light in vacuum
- 0.66 corresponds to the typical speed of light in polyethylene
- 0.70 corresponds to the typical speed of light in PTEF

Tips

To know when you have increased the delay enough, you can do the following:

- Connect a short circuit breaker to replace the device;
- Adjust the port extension until the phase flattens accordingly.

Notice

Determine the offset delay of standard short circuit breaker

Most of the calibration kit for short circuit breaker have a non-zero delay. Therefore, adjusting the delay in this way results in a delay error that is twice that of the short circuit breaker. Determine the offset delay time of a standard short circuit breaker by checking the standard definition.

3) Phase offset measurement

Menu path: **[Scale] → [Phase Offset]**.

In the input box, enter a value or press the arrows to select the value as desired.

4) Confusion check

Menu path: **[Swp]→[Swp PTS]**.

Enter any value smaller than the current value using the number keys on the keyboard or in

6.6. Reduce Trace Noise

the input area. If the displayed data changes, it is recommended to increase the number of points or reduce the frequency span.

6.6. Reduce Trace Noise

The noise on the measurement trace can be reduced by applying the features of the network analyzer. The features available to reduce the impact of trace noise are as follows.

- [Sweep Averaging.....201](#)
- [Trace Smooth.....202](#)
- [IFBW.....203](#)

6.6.1 Sweep Averaging**1) Sweep averaging**

- a) Sweep averaging is a feature that can reduce the impact of random noise on measurements.
- b) The analyzer performs calculation to each data point by averaging the results of the same data point over several consecutive sweeps. The number of consecutive sweeps is determined by the setting of the averaging factor.
- c) The trace average is applied to all measurements in a channel, and the average counter is displayed for each channel.
- d) The more average times, the more noise reduction, and the larger dynamic range.
- e) The effect of applying sweep averaging to reduce noise is the same as that of reducing the IF bandwidth.

2) Sweep Average

Figure 6.12 shows the effect of sweep average on measurement results.

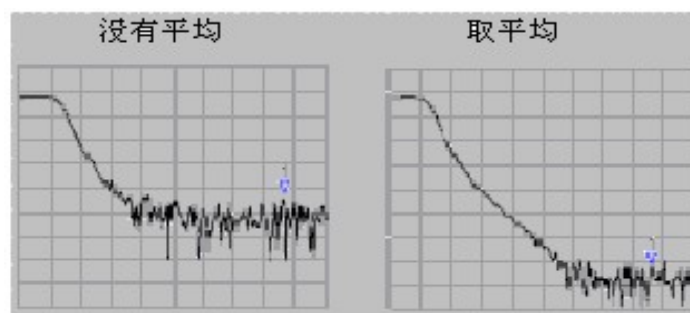


Figure 6.12 Effect of sweep average on measurement results

3) Sweep Average settings

Menu path: **[Averaging]**→ **[Averaging On Off]** .

6 Optimization & Measurement

6.6. Reduce Trace Noise

Switch the [Averaging ON OFF] to On position to enable the averaging function.

In the Average Factor box, enter a value directly or press the arrow button to increase or decrease the number of times the analyzer will be averaged.



Figure 6.13 Set sweep average

6.6.2 Trace Smoothing

The smoothing function may reduce the peak-to-peak noise of the broadband measurement data. The analyzer performs data averaging on a certain part of the display trace, and the number of adjacent data points averaged together is also called the smoothing aperture. This aperture can be specified as the number of data points or as a percentage of the X-axis span.

Tips

Use of smoothing

- Show with enough dots to avoid misleading results.
- Do not use smoothing for devices with high resonators or devices with wide trace changes. It may cause measurement errors.
- The smoothing function can be set independently for each trace.

1) Effect of trace smoothing

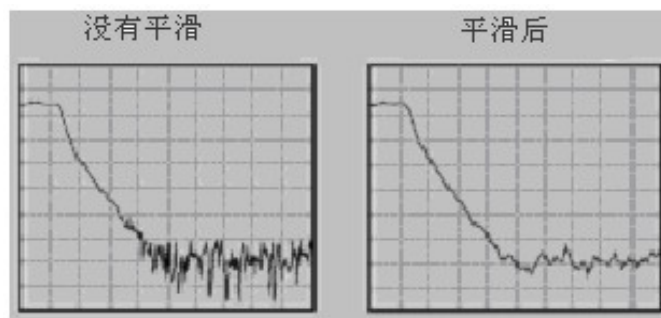


Figure 6.14 Effect of trace smoothing on measurement results

2) Trace smoothing settings

Menu path: [Averaging]→[Smoothing On Off].

Switch [Smoothing On Off] to ON position to enable the smoothing function.

Select a method to specify the value of the smoothing aperture:

- In **Smoothing Percent** box, enter a percentage value for smoothing. (max. 25%)
- In the **Smoothing Points** box, enter a smoothing point value. (Maximum value is 25% of the total number of points set for the test sweep)



Figure 6.15 Setting smoothing

6.6.3 IFBW

Reducing the bandwidth of IF receiver can reduce the influence of random noise in measurement. Every 10 times reduction in IF bandwidth can reduce the noise base by 10 dB. However, the narrow IF bandwidth can make the sweep time longer. The analyzer converts the received signal from the RF/microwave band to the lower 7.606 MHz IF, and the bandwidth of the IF bandpass filter or IF bandwidth can be adjusted from 5 MHz down to a minimum of 1 Hz.

It can individually set the IFBW for sweeping each channel or segment.

1) Reduce the effect of IF bandwidth

The effect of IF bandwidth on measurement results is shown in Figure 6.16:

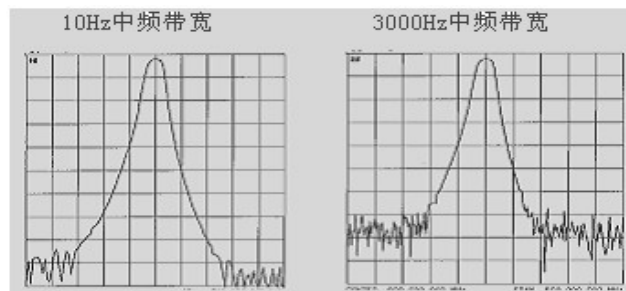


Figure 6.16 Effect of IF bandwidth on measurement results

2) Settings of IFBW

Menu path: **[Average] -> [IFBW]**.

In the input box, enter a value directly or press the arrow button to select a value.

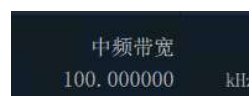


Figure 6.17 Set IFBW

6.7 Increase Data Points

A data point or a "point" is the sampling result for one piece of data at one stimulus. You can define the number of points measured in one sweep of the analyzer.

- The large number of data points facilitates to sweep the trace and thus better definition of

6 Optimization & Measurement

6.7 Increase Data Points

the device response.

- Most of the data processing operations are performed between adjacent points. A SWEEP is a sequence of consecutive data point measurements corresponding to a sequence of stimulus values.
- The default number of points per sweep is 201.

The sweep time of the analyzer can be scaled according to the sweep points. However, the total measurement cycle time is not scaled according to the sweep points, but is also influenced by other factors, such as flyback time, data calculation and molding time. Refer to the Understanding the Technical Specifications section to learn more about how points and other settings affect the sweep time.

Tips

Use of data points

- To get the best trace resolution, the maximum number of data points can be used.
- To achieve faster throughput, the minimum number of data points that gives acceptable accuracy should be utilized.
- In order to identify the optimal number of points, it is required to find a value that allows you to continue to increase the number of measurement points without significantly affecting the measurement result of that value.
- In order to ensure an accurate measurement calibration, a calibration of the same number of points used for the measurement should be performed.

1) Specific operations

Menu path: [Swp]—>[Sweep Points].

In **Sweep Points** input box, enter a value in the input box or press the arrow to select a value.



Figure 6.18 Set sweep points

2) Optimize measurement results

A single measurement may involve multiple interdependent settings, and those settings can be modified for your measurement application: faster throughput or more accurate data. You can balance those settings through selection to optimize the measurement.

- Increase measurement throughput
- Improve measurement accuracy

6.8. Improve Measurement Stability

Several factors may cause measurement instability. To ensure the possibility of repeated measurements, you may apply several methods to produce a stable measurement environment. The detrimental effects on the measurement accuracy are described as follows.

1) Frequency drift

The frequency accuracy of the analyzer depends on the accuracy of the internal 10 MHz frequency oscillator. If the measurement application requires higher frequency accuracy and stability, it is unnecessary to consider internal frequency standard. Instead, an external frequency source with high stability can be provided through the 10 MHz reference input connector on the rear panel.

2) Temperature drift

- a) Thermal expansion and contraction may change the electrical characteristics of following components.
 - Analyzer internal device
 - Standard of calibration kits
 - Test device
 - Cable
 - Adapter
- b) Ways available to reduce the impact of temperature drift on your measurements include the following:
 - Apply a temperature controlled environment.
 - Ensure the temperature stability of the calibration kit.
 - Avoid unnecessary operation to the calibration kit during calibration.
 - Ensure that the deviation between the ambient temperature and the measured calibration temperature is $\pm 1\text{ }^{\circ}\text{C}$.

3) Incorrect measurement calibration

If the measurement calibration is incorrect, the correct response of the DUT cannot be detected. To ensure a correct calibration, the following practices should be considered:

- a) Perform measurement calibration to the connection of the DUT, i.e. the reference plane.
- b) If an additional accessory (such as cable, adapter and attenuator) is inserted into the DUT after a measurement calibration, compensate the increased electrical length and delay by applying the port extension function.
- c) Apply the calibration standard defined in the calibration process.
See the Precise Measurement Calibration section for more details.

4) Device connector

Good connectors are a must for repeatable measurements. For a good connection, the

6 Optimization & Measurement

6.9 Increase the Sweep Speed

method is as follows:

- a) Check and clean the connectors of all components in test equipment.
- b) Use the correct connection method.
- c) Avoid moving cables during measurements.

5) Use an external frequency reference

Input the external frequency reference signal to the rear panel connector.

- a) Input frequency: 10MHz $\pm$ 10ppm
- b) Input level: -15 to +20dBm
- c) Input impedance: 200ohms

6) Control room temperature

- a) Run the analyzer for more than 30 minutes before a measurement calibration or a device measurement.
- b) Measure the equipment in a temperature controllable environment. All descriptions and characteristic applications are within a range of 23 °C $\pm$ 3 °C (unless otherwise specified).
- c) Ensure that the deviation between the ambient temperature and the measured calibration temperature is $\pm$ 1 °C.

7) Stabilize calibration standard temperature

- a) One hour before the measurement calibration, open the calibration kit box and take out the standard part from the protective foam.
- b) When performing a measurement calibration, avoid unnecessary operation on the calibration kit.

8) Get good connectors

- a) Check all connectors with a magnifying glass. Look for signs of damage such as wear, bending, breaking, deep scratches, dents, rounded shoulders, dirt, or metal chips.
- b) Clean all connectors with isopropyl alcohol and cotton swabs.
 - Evaporate the alcohol
 - Dry the connector gradually with compressed air
- c) Connect devices correctly
 - Connect yourself and all devices to the ground with an antistatic pad and wrist strap.
 - Arrange the connectors and line them up.
 - Only rotate the coupling nuts.
 - Use a torque wrench for final connection.

6.9 Increase the Sweep Speed

The fastest sweep speeds will increase measurement efficiency, and the following settings

are optimized for the microwave analyzer to achieve the fastest sweep speed:

1) Sweep settings

Get the fastest sweep speed by carefully making each of the following settings:

- **Frequency Span:** Only measure the frequency range that the DUT is concerned with. For the setting method, see "4.3 Setting frequency range".
- **Segment Sweep:** Measure the frequency band of most interest. For the setting method, see the "**Setting segment sweep type**" in the "[4.5.2 Sweep Type setting](#)" section of "[4.5 Setting sweep](#)".
- **Turn OFF Step Sweep:** If the measurement allows, do not use the step sweep mode so that the sweep time is minimized. For the setting method, see "4.5.3 Sweep settings" in "4.5 Setting sweep".
- **Auto Sweep Time:** The fastest sweep can be obtained with the default sweep time for the current settings. See "4.5.3 Sweep time" of "4.5 Set sweep" for the setting method.
- **Sweep Points:** The minimum sweep points required for measurement is used. To set the sweep points, see "6.7 Increasing data points".

2) Reduce noise settings

Sweep time can be reduced by properly performing the following two settings, thus helping to obtain acceptable measurement result.

- **IF Bandwidth:** The widest IF bandwidth that gives acceptable trace noise and dynamic range is used. To set the IF bandwidth, see "6.6.3 IF bandwidth" in "6.6 Reducing trace noise".
- **Averaging:** Reduce the averaging factor as much as possible, or turn off the averaging function. See "6.6.1 Sweep averaging" in "6.6 Reducing trace noise" for setting methods.

3) Select calibration type

Selecting the fastest calibration measurement that meets the measurement accuracy requirements takes approximately the same amount of time for a sweep measurement without error correction as for a sweep measurement with response calibration correction, while a full dual-port calibration measurement requires error correction by forward and reverse sweep measurements to update the measurement parameters, even when only one S-parameter is displayed, and therefore it takes the longest measurement time.

For details on calibration type, see "6.2 Selecting calibration type".

4) Disable unnecessary functions

The analyzer must be updated with information about all active functions. To speed up sweeping, turn off all of the following functions that are not necessary for the measurement application.

- **Undesired traces**
- **Unused marker**

6 Optimization & Measurement

6.9 Increase the Sweep Speed

- **Smoothing**
- **Limit Test**
- **Trace Calculation**
- **Display**

The sweep speed of the analyzer is dependent on various measurement settings and requires several attempts to obtain the fastest sweep speed and meet the required measurement results.

- a) Delete undesired traces

Press the Right Mouse button on the Status bar button where the trace is to be deleted to display the Context Menu.

Click [DEL Trace] in the Context Menu.

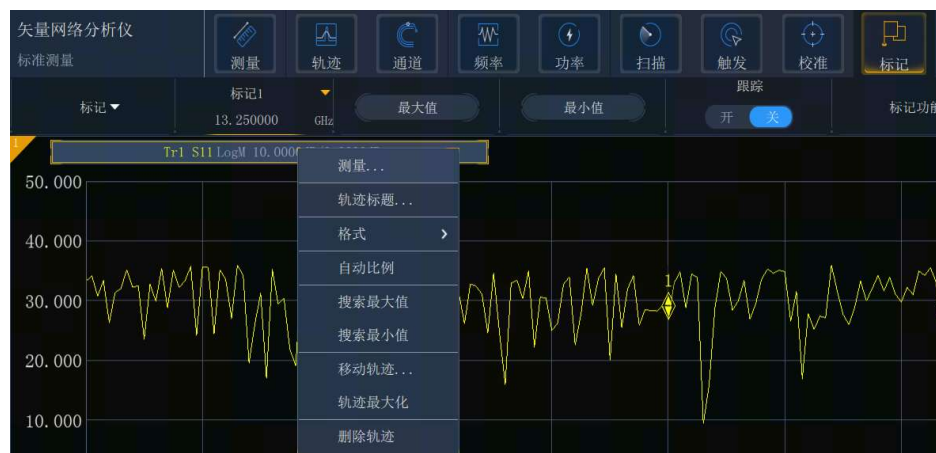


Figure 6.19 Delete undesired traces

- b) Disable the unused marker

Menu path: **[Mrk]→[Marker];**

Select the cursor to be closed and uncheck.



Figure 6.20 Closing an unused cursor

- c) Disable the smoothing function

Menu path: **[Avg]**→**[Smooth]**;

Close **[Smooth]** switch to deactivate function.

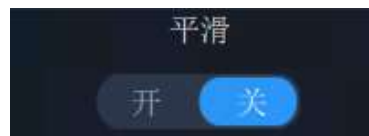


Figure 6.21 Turning-off smoothing function

- d) Turn off limit testing

Menu path: **[Analy]**→**[Test]**→**[LIM Test...]** to display the **Limit Test** dialog box.

Click [LIM Test ON/off] to clear the check box to turn off the limit test function.

6.10 Improve Multi-state Measurement Efficiency

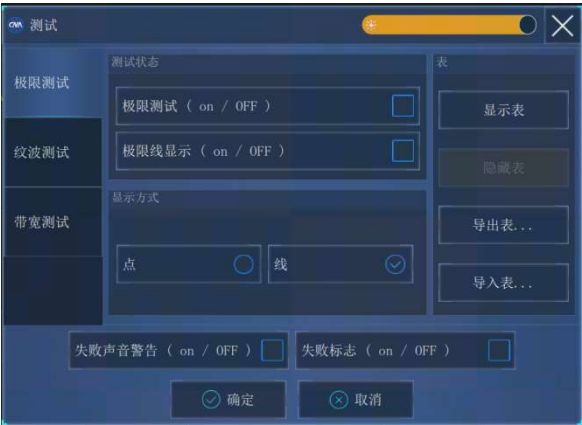


Figure 6.22 Turn off limit test

- e) Turn off the trace operation function

Menu path:[Analysis]→[Trace Transportation];
Select **Off**.



Figure 6.23 Turning-off the trace operation function

6.10 Improve Multi-state Measurement Efficiency

If multiple parameter measurements are required to characterize a device, a variety of methods can be applied to improve measurement efficiency. Try the various methods below to find the most suitable solution for your measurement application.

- [Improve Measurement Efficiency via Measurement Settings.....210](#)
- [Automatic Change of Measurement Settings.....212](#)
- [Fast Call of Measurements.....212](#)

6.10.1 Improve Measurement Efficiency via Measurement Settings

In order to improve the measurement efficiency of a DUT that requires multiple parameter measurements, it is helpful to be familiar with the operation of the analyzer, and this knowledge

6.10 Improve Multi-state Measurement Efficiency

helps to establish the optimal solution for the measurement application. For more information on traces, channels and windows related to multi-parameter measurements see ["3.4 Traces, channels and windows of the analyzer"](#).

1) Reasonable arrangement of a set of measurements

Arrange a set of measurements of a DUT in one instrument state, and store the instrument state to facilitate the above set of measurements via the callback function in the future. A set of measurements can also be carried out through the pre-configured measurement settings function of the analyzer. For details, see the ["Pre-configured measurement settings"](#) section in ["4.8 Observing multiple traces and opening multiple channels"](#).

2) Use of segment sweep

It is very convenient to use segment sweep when the following settings need to be changed for multi parameter measurement characterizing a device.

- Frequency range:
- Power level
- IFBW
- Sweep Points

Segment sweep allows the definition of a set of frequency ranges with their own properties, so that measurements of a DUT that require multiple measurement settings can be done with a single sweep. For more information on segment sweep, see the ["Setting segment sweep type"](#) in the ["4.5.2 Sweep Type setting"](#) section of ["4.5 Setting sweep"](#).

3) Selective trigger measurement

Measurements are made using the trigger function with the following settings:

- Constantly update those measurements with rapidly changing data.
- Occasionally update those measurements with rarely changing data.

For example, a filter tuning measurement can be set up like this:

- One channel is used to measure the passband response of the filter for tuning.
- One channel is used to measure the out-of-band response of the filter.

In this way, the tuning measurement of the filter can be continuously observed. If all channels are continuously updated, the response speed of the analyzer will be reduced and the filter cannot be tuned quickly. The following describes how to establish the following measurement settings:

- The measurement data in Channel 1 will be continuously updated.
- The measurement data in Channel 2 will be updated as needed.

a) Use of the mouse

- 1) Establish a two channel measurement.

Menu path: Click **[Chan]→[Open Channel...]**, check**[Use New Window]** and click**[OK]**.

- 2) Set the trigger of channel 1: continuously update the measurement data.

Menu path: →**[Trig]→[Trig...]** to display the Trigger dialog box.

6 Optimization & Measurement

6.11 Accelerating Data Transmission

Select the **[INTL Trig]** radio box in the **Trigger Source** zone.

In **Channel Trigger State** zone, click **[Channel]** box and select **Channel1**.

Click the **[CONT]** radio box in the **Trigger Settings** zone.

- 3) Set the trigger of channel 2: update the measurement data when necessary.

Click the **[Ch]** box in the **Trigger Settings** zone to select Channel 2.

Click the **[SGL]** or **[GRP]** radio box in the **Trigger Setting** zone. If the **Group Trigger** mode is selected, enter the number of groups in the **[GRP Sweep Times]** input box.

- 4) Click on the window below to make channel 2 the currently active channel.

- 5) Update the measurement data in channel 2.

- 6) Click the **[SGL]**, **[GRP Swp]** or **[Re-Swp]** button corresponding to **[Trigger]** to update the measurement data in channel 2.

6.10.2 Automatically Change Measurement Setting

Automatic change of measurement settings by programming can effectively improve the efficiency of multi-state and multi-parameter measurements.

6.10.3 Quickly Call Measurement

The most efficient way to recall a measurement is to store a set of measurements as an instrument state.

- The time difference between calling an instrument state that includes multiple measurements and calling an instrument state that has only one measurement is small.
- Each call takes a corresponding amount of time, and the call time can be reduced by creating a set of measurements in groups to call them.

6.11 Accelerating Data Transmission

Using the fastest data transmission helps to improve the measurement efficiency. The following methods can be used to improve the data transmission speed:

- [Use Single Trigger Mode.....212](#)
- [Transmit As Little Data As Possible.....213](#)
- [Use Real Format.....213](#)
- [Use LAN.....213](#)
- [Use COM Programs.....213](#)

6.11.1 Use Single Trigger Mode

The use of single trigger mode ensures that measurements are completed before data transmission starts.

Menu path: [Trig]→[Trig...] to display the **Trigger** dialog box.

Select the [INTL] radio box in the Trigger Source zone.

Click the [Ch] box in the **Channel Trigger State** zone to select the measurement channel for data transmission.

Click the [CONT] radio box in the **Channel Trigger State** zone, and click the [OK] button to close the dialog box.

Update measurement results:

- 1) Click the corresponding trace status button to make the trace and channel to be updated active.
- 2) Click [Single] in [Trigger] interface.

6.11.2 Transmit As Little Data As Possible

For example, segment sweep is used to reduce the number of points in a trace, instead of transmitting the whole trace with many linear points.

6.11.3 Use Real Number Format

The fastest transmission speed can be obtained by selecting the real number format in the automatic measurement using SCPI.

6.11.4 Use LAN

The speed of data transmission by SCPI automatic measurement application can be improved via LAN.

6.11.5 Use COM Programs

The fastest data transmission speed can be obtained by using COM in the automatic measurement application.

6.12 Use Macro

Macros are executable files loaded into the analyzer and run on the analyzer. The 3657 series vector network analyzer supports up to 10 macros.

- [New Macro.....213](#)
- [Macro Settings Dialog Box.....214](#)

6.12.1 New Macro

Menu path: [SYS]→[Macro]→[Macro Set...] to display the **Macro Setting** dialog box.



Figure 6.24 Set macro

In **Macro Settings** dialog box, click to activate the blank line below the existing macro settings, and check **[Edit]** to edit the macro.



Figure 6.25 Macro Settings dialog box

- Enter the title text to describe the macro in the **[Macro Title]** box.
- Enter the detailed path of the executable file in the [EXE Macro] box, or click the [Browse...] button to find the executable file.
- Enter the string parameters passed to the executable file in the [Macro PARAM] box.

6.12.2 Macro Settings Dialog Box

1) **[Macro Title]** box

Display the macro title. When [Macro] is clicked in the System menu, the name of the macro

6.12 Use Macro

title is displayed in the Macro submenu item. In order to display the full Macro Title in the current item toolbar button, the title should preferably be no longer than 14 characters. The macro title is associated with the executable file, and it is desirable to describe the function of the macro so that the different macros can be easily distinguished. For example, a macro that allows access to the home page developed by Ceyear could be titled "Ceyear Home Page".

2) [Executable Macro] box

Display the full path of the executable file, such as the macro accessing the home page of Ceyear. The path of the executable file is: C:\Program Files\Internet Explorer\IEXPLORE.EXE.

3) [Macro Parameters] box

Display the parameters passed to the executable file and referenced by the executable file. For example, the parameter required for running the macro accessing the home page developed by Ceyear is <http://www.ceyear.com>.

4) [Browse...] button

Used to browse drives and directories, locate executable files, and establish the full path of the executable files.

5) [Edit] button

Set and modify the selected macro after being checked.

6) [Clear] button

Clear the selected macro when clicked.

7) [Up] button

Click the **[Up]** button to move the selected macro up one line and use it to reorder the macros. The order of macros in the dialog box is the same as that in the Macro current item toolbar.

8) [Down] button

Click the **[Dn]** button to move the selected macro down one line.

7 Calibration

Calibration reduces measurement errors. This chapter includes the following contents:

● Calibration Overview.....	217
● Select Calibration Type.....	219
● Calibration Wizard.....	222
● Calibration Set.....	231
● Measurement Error.....	236
● Edit Calibration Kit Definition.....	242
● Calibration Standard.....	254
● TRL Calibration.....	256
● Fixture Compensation Calibration.....	258

7.1 Overview of Calibration

Measurement calibration is a process to determine the system error through the standard with known measurement characteristics, and then eliminate the influence of these system errors when measuring the DUT, which reduces the measurement errors and improves the measurement accuracy of the vector network analyzer.

● Definition of Calibration.....	217
● Significance of Calibration.....	218
● Applications of Calibration.....	218
● Simple Process of Calibration.....	218

7.1.1 Definition of Calibration

Calibration means using an error model to eliminate one or more system errors. The analyzer solves error terms in the error model by measuring high quality calibration standards such as open circuit breaker, short circuit breaker, loads and through devices. See “7.5.3 System error” of “7.5 Measurement error” for the detailed information of system error.

During measurement, appropriate calibration methods shall be selected according to the requirements of measurement type and measurement accuracy. For details, see “7.2 Selecting calibration type”.

Various types of calibrations can be completed through the calibration wizard of the instrument. For details, see “7.3 Calibration wizard”.

After the calibration is completed, all calibration data will be saved to the calibration set file. The calibration set file can be applied to any channel with the same stimulus as this file, thus saving time for calibration again. For more information, refer to “7.4 Calibration Set”.

7.1 Overview of Calibration

The measurement accuracy after calibration depends on the quality of the calibration standard and the model definition accuracy of the calibration standard in the calibration kit definition file. The calibration kit definition file is saved in the analyzer, and the calibration kit used must be consistent with the calibration kit definition file to ensure the measurement accuracy. For details on performing accurate calibration, see ["7.5.4 High-precision measurement calibration"](#).

If users use their own customized calibration kit (such as for fixture measurement calibration), they must correctly define the calibration standard in the User Calibration Kit Definition File. For details, see ["7.6 Edit calibration kit"](#).

For the testing of DUT with non-standard connectors, such as on-chip, microstrip, etc., the test fixture is usually used for connection of DUT with analyzer and for measurement. The errors introduced by the test fixture, if any, can be eliminated by applying the fixture compensation technology. For details, please refer to ["7.9 Fixture Compensation Calibration"](#).

7.1.2 Significance of Calibration

Making ideal analyzers that do not require any error correction is impossible in hardware circuits, and even if these hardware circuits could be made exceptionally well to ignore the need for error correction, the cost would be extremely high. In addition, the measurement accuracy of the analyzer is largely affected by the external accessories of the analyzer. The amplitude and phase changes of the test components, such as connecting cables and adapters, will mask the real response of the DUT. Therefore, it is the best way to balance the performance and cost of the hardware, make the hardware as good as possible, and improve the measurement accuracy through calibration.

7.1.3 Calibration Applications

- It is desirable to obtain the highest measurement accuracy as far as possible.
- Different types of connectors or impedances are used.
- A cable is connected between the DUT and the analyzer test port.
- The DUT or the long electric delay device is to be measured over a wide frequency range.
- An attenuator or other similar device is connected to the input or output port of the DUT.

7.1.4 Simple Procedures for Calibration

- 1) Connect the analyzer according to the measurement requirements.
- 2) Select the appropriate analyzer settings to optimize the measurement.
- 3) Remove the DUT and select the calibration type and calibration kit by using the calibration wizard.
- 4) Follow the prompts in the calibration wizard to connect the calibration standard required in the selected calibration type for measurement. The analyzer calculates the error term by measuring the calibration standard and stores it in the analyzer's memory.

- 5) Reconnect the DUT for measurement. When error correction is used in device measurement, the influence of the error term will be removed from the measurement.

7.2 Select Calibration Type

There are 9 calibration types commonly used by 3657 series vector network analyzer. The details of each calibration type are as follows:

1) Open circuit response

- a) Calibration accuracy: low to medium
- b) Measurement parameters: S11, S22, S33 and S44
- c) Required calibration standard: open circuit breaker
- d) Corrected system error: reflection tracking
- e) Measurement application: reflection measurement at any port

2) Short-circuit response

- a) Calibration accuracy: low to medium
- b) Measurement parameters: S11, S22, S33 and S44
- c) Required calibration standard: short circuit breaker
- d) Corrected system error: reflection tracking
- e) Measurement application: reflection measurement at any port

3) Through response

- a) Calibration accuracy: medium
- b) Measurement parameters: transmission measurement S parameters
- c) Required calibration standard: through device
- d) Corrected system error: transmission tracking
- e) Measurement application: transmission measurement in any direction

Notice

Adapter is applied as a through device

The through device is defined as zero length and zero loss in the calibration kit definition file. If the adapter is used as the through device in the calibration process, the characteristics of the adapter must be characterized in the calibration kit definition file to allow accurate calibration. For details, see [“7.5.4 High-accuracy measurement calibration”](#).

4) Through response and isolation

- a) Calibration accuracy: medium
- b) Measurement parameters: transmission measurement S parameters

7.2 Select Calibration Type

- c) Required calibration standard: through device, two loads (one load per port)
- d) Corrected system error:
 - Transmission tracking
 - crosstalk
- e) Measurement applications:
 - transmission measurement in any direction
 - Isolation calibration is required to improve the dynamic range of the system.

Notice

Isolation calibration

If a load cannot be connected to each port at the same time, isolation calibration cannot be performed.

5) Single port (reflection)

- a) Calibration accuracy: high
- b) Measurement parameters: S11, S22, S33 and S44
- c) Required calibration standard: open circuit breaker, short circuit breaker and load
- d) Corrected system error:
 - Directivity
 - Source matching
 - Reflection tracking
- e) Measurement application: reflection measurement at any port

6) Enhanced frequency response

- a) Calibration accuracy: high
- b) Measurement parameters: unilateral reflection parameters and transmission parameters
- c) Required calibration standard: open circuit breaker, short circuit breaker and through device
- d) Corrected system error:
 - Directivity
 - Source matching
 - Reflection tracking
 - Load matching
 - Transmission tracking
- e) Measurement applications:
 - Large attenuation test such as long cable test
 - Amplifier gain and other large gain tests

7) Quick SOLT

- a) Calibration accuracy: high
- b) Measurement parameters: unilateral reflection parameters and transmission parameters
- c) Required calibration standard: open circuit breaker, short circuit breaker and through device
- d) Corrected system error:
 - Directivity
 - Source matching
 - Reflection tracking
 - Load matching
 - Transmission tracking
- e) Measurement applications:
 - Only one-side port and through standard measurements are performed to complete all error correction of full dual ports.
 - Simplify reflection standard measurement

8) Dual-port SOLT

- a) Calibration accuracy: high
- b) Measurement parameters: all
- c) Required calibration standard: open circuit breaker, short circuit breaker and through device
- d) Corrected system error:
 - Directivity
 - Source matching
 - Reflection tracking
 - Load matching
 - Transmission tracking
- e) Measurement applications:
 - Measurement of all S parameters
 - It is necessary to improve the measurement accuracy through 12 error corrections.

9) Dual-port TRL

- a) Calibration accuracy: high
- b) Measurement parameters: all
- c) Required calibration standard: reflectors, through device and air line
- d) Corrected system error:
 - Directivity
 - Source matching
 - Reflection tracking
 - Load matching

7.3 Calibration Wizard

- Transmission tracking
- e) Measurement applications:
 - High accuracy test
 - Fixture/waveguide test

7.3 Calibration Wizard

The vector network analyzer provides not only the types of calibration described in 7.2 (by programmed control), but also a user-friendly calibration wizard. The calibration wizard has the following features:

- 1) Simplified calibration setting interface, and two measurement modes (step-by-step and simplified) available;
- 2) Source and receiver calibration supported;
- 3) Automatic identification of SOLT, SOLR, SSLT, SSST, TRM, TRL and other calibrations;
- 4) Multi-port simplified calibration (extended to multi-port devices) and automatic simplified connection optimization supported;
- 5) 2040X series electronic calibration kit supported;
- 6) Waveguide calibration kit supported;
- 7) Parameter type and data type mechanical calibration kits supported;
- 8) Mechanical calibration kits of some foreign manufacturers (Keysight, R&S, Maury) supported.

Through the calibration wizard, users can freely configure the calibration port, calibration kit and calibration type. The specific operation is as follows.

- [Run Calibration Wizard.....222](#)
- [Perform Calibration Settings.....223](#)
- [Modify Calibration Method.....225](#)
- [Perform Source Calibration Settings.....227](#)
- [Measurement Standard.....228](#)
- [Complete Calibration.....230](#)
- [Save As User Calibration Set.....230](#)

7.3.1 Run Calibration Wizard

Menu path: Click **[Cal]**→**[Calibration...]** to display the Calibration Wizard interface.



Figure 7.1 Run calibration wizard

7.3.2 Perform Calibration Settings

For the first running of the calibration wizard, after the analyzer is turned on, the software will automatically update the settings according to the analyzer configuration, with the number of calibration ports selected as the maximum number of ports on the analyzer, and other settings as follows:

If the analyzer is connected to an electronic calibration kit, the DUT connector type field in the port calibration column and the calibration kit drop-down menu will display “Electronic calibration kit”; the More column will display **[ECal Port Detection]** option, which is checked by default.



Figure 7.2 Cal Settings interface (with an electronic calibration kit connected)

If the analyzer is not connected with an electronic calibration kit, the DUT connector type field in the port calibration column and the calibration kit drop-down menu will display the connector type and calibration kit matching the analyzer connector, for example, the interface of 67GHz vector network analyzer (with 1.85mm connector) will display 1.85mm connector and 85058EP calibration kit; the More column will display **[Step-by-step Standard Measurement]** option, which is not checked by default.



Figure 7.3 Cal Settings interface (with no electronic calibration kit connected)

For running of calibration wizard after the first running, the last calibration setting state will be activated.

[Calibration Ports]: select the number of ports to be calibrated based on the test needs. For through response calibration, the number of ports should be 2.

[VNA Port]: select the VNA port to be calibrated. When the setting of **[Calibration Ports]** is maximum, no modification is allowed.

[DUT Port]: the connector type of the existing calibration kit can be selected. Calibration kit and connector type can be edited and viewed as described in ["7.6 Edit Calibration"](#).

7.3 Calibration Wizard

[Cal Kit]: the calibration kit available under the corresponding **[DUT Connector]** can be selected.

[Source and Receiver Calibration]: when this option is checked, the source power will be calibrated and the source power and receiver measurement curves will be corrected when the S-parameters are being calibrated.

[ECal Port Detection]: when this option is checked, the connection port of the electronic calibration kit will be detected during the electronic standard measurement; otherwise, the connection port of the electronic calibration kit during the electronic standard measurement will be specified.

[Step-by-step Standard Measurement]: when this option is checked, the standard will be measured as per the guidances steps; otherwise, the standard to be measured will be displayed on one page.

[Couple Port Configuration]: when this option is checked, the DUT connector type and calibration kit in other rows in the Port Configuration column are the same as those in the first row, and only the information in first row can be modified; otherwise, all the DUT connector types and calibration kits in the Port Configuration column can be modified. When **[Calibration Ports]** is set to 1, this option is hidden.

[Modify Cal Method]: when this option is checked, the "Modify Cal Method" page shown in Figure 7.5 will be added; otherwise, the software will create a default calibration method for calibration according to the calibration configuration information. When **[Calibration Ports]** is set to 1, this option is hidden and only SOL single-port calibration is available.

Notice

1、 Selection of calibration kit

Only when the corresponding connector type is selected will the optional calibration kits under this connector type be shown in the calibration kit drop-down menu.

2、 Selection of step-by-step standard measurement

When an electronic calibration kit is present, the subsequent standard measurement will be carried out according to the steps;

When only the mechanical calibration kit is used, the step-by-step standard measurement will be subject to the status of the **[Step-by-step Standard Measurement]** check box.

Tips

Electronic calibration kits and electronic calibration:

For more information about this part, see the User's Manual for 2040X Electronic Calibration Kits.

After setting, click **[Next]**.



Figure 7.4 Change frequency interface

If the frequency of the calibration channel is out of the frequency range of the selected calibration kit or the standard information of the selected calibration kit is incomplete, the Change Frequency interface shown in Figure 7.4 will pop up. After resetting the frequency, click **[OK]** to modify the frequency of the calibration channel.

7.3.3 Modify Calibration Method

When the **[Modify Cal Method]** option is checked, click **[Next]** to enter the Modify Cal Method interface. The left side of the Modify Cal Method interface displays the information of analyzer corresponding to the measured through number including **[1st Port]**, **[2nd Port]**, **[Thru Method]**, **[Cal Method]** and **[Isolations]**, and the right side displays **[Add Thru]** option and **[Delete Thru]** option.

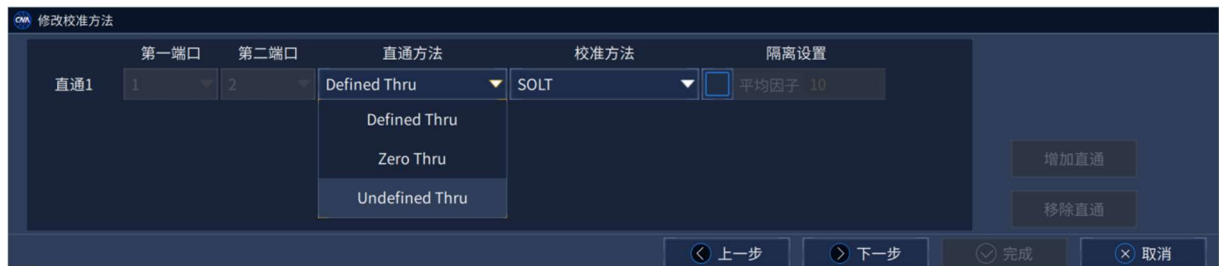


Figure 7.5 Modify through method



Figure 7.6 Modify calibration method

7.3 Calibration Wizard

When **[Calibration Ports]** is set to N, the minimum number of measurement through is N-1, the maximum number of measurement through is $N \times (N-1) / 2$, and the minimum number of measurement through is set to be the current number of measurement through by default.

[1st Port] and **[2nd Port]** drop-down menus: the drop-down menu is for selecting the port to be modified after it is clicked. If the current number of measurement through is equal to the maximum number of measurement through, the drop-down menu cannot be modified.

[Thru Method] drop-down menu: this drop-down menu is for selecting the desired through method when clicked. After modifying the through method, the content in the **[Cal Method]** drop-down menu will be updated. There are five through methods:

Zero Thru: zero-length through, which occurs only when **[DUT Connector]** of the two ports can be directly connected.

Defined Thru: defined through, which occurs only when the through class of corresponding standard class of the calibration kit contains through standard, and the through standard well matches the **[DUT Connector]** of the two ports.

Undefined Thru, unknown through, which occurs only when independent reference receivers are used for the two ports.

Defined Thru(ESol): defined through (electronic), which occurs only when at least one port uses an electronic calibration kit, and the two ports can be directly connected by the electronic calibration kit.

Undefined Thru(ESol): undefined through (electronic), which occurs only when the device has multiple ports, at least one port uses an electronic calibration kit, and the corresponding two ports can be directly connected by the electronic calibration kit.

[Cal Method] drop-down menu: click to select the desired calibration method. There are five calibration methods:

TRL: through-reflection-air line calibration, which occurs only when **[Thru Method]** is set to Defined Thru, and the corresponding two ports use the same TRL calibration kit, where TRL calibration kit means that the information in the calibration kit TRL standard class is complete.

SOLT: open-short-load-through calibration, which occurs only when the corresponding two ports use the SOL calibration kit, where SOL calibration kit means that the information in the calibration kit SOLT standard class, except for the through standard, is complete, and the electronic calibration kit is a SOL calibration kit.

QSOLT: quick open-short-load-through calibration, which occurs only when **[Thru Method]** is set to Defined Thru, Zero Thru or Defined Thru(ESol), and the corresponding two ports use the SOL calibration kit. For QSOLT calibration method, the number following it represents the port subject to SOL calibration.

EnhResp: enhanced response calibration, which occurs only when **[Calibration Ports]** is set to 2 and **[Thru Method]** is set to Defined Thru, Zero Thru or Defined Thru(ESol). For EnhResp calibration method, the number following it represents the port subject to SOL calibration.

TransResp: through response calibration, which occurs only when **[Calibration Ports]** is set to 2 and **[Thru Method]** is set to Defined Thru, Zero Thru or Defined Thru(ESol). For the TransResp calibration method, the number following it represents the stimulus port.

[Isolation]: the isolation between two ports is not calibrated by default. When it is necessary

7.3 Calibration Wizard

to measure and correct the isolation between ports, please check the check box in front of **[Averaging Factor]**, and after the averaging is enabled, the average times during isolation can be set.

[Add Thru]: click to add measurement through. The button is not available when the current number of measurement through is equal to the maximum number of measurement through.

[Delete Thru]: click to delete the currently selected through. The button is not available when the current number of measurement through is equal to the minimum number of measurement through.

After setting, click **[Next]**.

7.3.4 Perform Source Calibration Settings

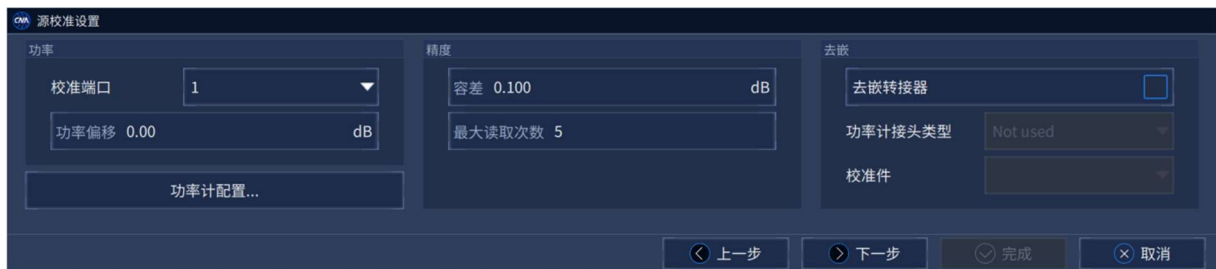


Figure 7.7 Source calibration settings interface

If **[Source and Receiver Calibration]** is checked on the Cal Settings interface, the Source Calibration Settings interface will be shown on the subsequent pages. On the Source Calibration Settings interface, the user can perform such settings as **[VAN Port]**, **[Power Offset]**, **[Tolerance]**, **[Max. Readings]**, **[De-embedding Adapter]**, **[Connector]**, **[Cal Kit]** and **[Power Meter Settings...]**.

[VAN Port]: it is allowed to select the **[Instrument Port]** in the Cal Settings interface. When the selected calibration method is EnhResp or TransResp, the calibration port cannot be modified.

[Power Offset]: set the specified power offset to obtain the required port output power, where port output power = power level + power offset.

[Tolerance]: tolerance is the limit range for judging source calibration.

[Max Readings]: the port power output will be adjusted when the port output power exceeds “power meter reading ± tolerance” during the source calibration. The maximum reading is the limit on the repeated adjustment times of the port power output.

[De-embedding Adapter]: when the S-parameter calibration end face and the power meter end face need to be connected through the adapter, this option can be checked to add the calibration process of the adapter and eliminate the influence of the adapter.

[Connector]: after the **[De-embedding Adapter]** is checked, the connector type of the existing calibration kit can be selected. The default value is Not used. When Not used is selected, de-embedding calibration will not be performed to the adapter even if the **[De-embedding Adapter]** is checked.

[Cal Kit]: When **[Connector]** is set as the connector type of the existing calibration kit, the

7.3 Calibration Wizard

calibration kit under the corresponding connector type can be selected.

The power meter used can be changed via **[Power Meter Settings...]**.

After setting, click **[Next]**.

7.3.5 Measurement Standard

If **[Source and Receiver Calibration]** is checked on the Cal Settings interface, the Source Calibration Measurement interface shown in Figure 7.8 will appear in the measurement standard.

[Power]: set the output power of specified port, where port output power = power level + power offset.



Figure 7.8 Source calibration measurement interface

Click **[Measure]** to perform measurement for source calibration. After the measurement is completed, the system will automatically enter the measurement interface of the next standard.

After the measurement is started, the **[Abort]** button is turned on, and you can click the **[Abort]** button to suspend the current measurement process, as shown in Figure 7.9.



Figure 7.9 Source calibration measurement interface (measurement in progress)



Figure 7.10 Calibration kit measurement (step-by-step) interface

For the measurement process of the calibration kit, if **[Step-by-step Standard Measurement]** is checked on the Cal Settings interface, the Calibration Kit Measurement (Step-by-step) interface as shown in Figure 7.10 will pop up. The sequence number of current measurement step and the total number of measurement steps will be shown in the upper left corner of the interface. When all standard measurements are completed, the Calibration Finish interface will automatically pop up.

You can select the calibration step by clicking **[Next]** or **[Back]**.

For the measurement process of the calibration kit, if **[Step-by-step Standard Measurement]** is not checked on the Cal Settings interface, the Calibration Kit Measurement (Simplified) interface as shown in Figure 7.11 will pop up. The first figure shows the SOLT Calibration Measurement interface with the active calibration adapter de-embedded. The second figure shows the TRL Calibration Measurement interface.



Figure 7.11 Calibration kit measurement (simplified) interface

In the Calibration Kit Measurement (Simplified) interface, you can click the button corresponding to the standard to finish the measurement. When all standard measurements are completed, the **[Finish]** button and **[Next]** button become effective, but the Calibration Finish interface will not automatically pop up.

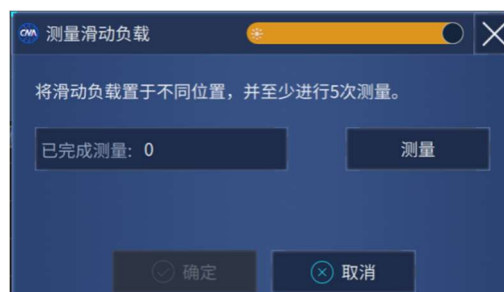


Figure 7.12 Calibration kit measurement (sliding load) interface

When a Calibration Kit Measurement interface contains a sliding load standard, the

7.3 Calibration Wizard

Calibration Kit Measurement (Sliding Load) interface will pop up as shown in Figure 7.13 during measurement. For the measurement of the sliding load, it is necessary to place the sliding load in different positions and make at least 5 measurements. When the 5 or more measurements are done, the **[OK]** button is enabled. You can click **[OK]** to end the sliding load measurement.

After all standard measurements are completed, click **[Next]** to enter the Calibration Finish interface.

7.3.6 Finish Calibration

After all standard measurements are completed, the Calibration Finish interface pops up. Click **[Finish]** to automatically save the current calibration data to the register.

To save the calibration data to the user calibration set, click **[Save as User Cal Set...]**. For more information on the calibration set, see "7.4 Calibration Set".



Figure 7.13 Calibration completion interface

If Undefined Thru method is used during calibration, when you click **[Finish]**, the Adapter Delay Confirmation interface will pop up. After confirming the delay of the adapter, click **[OK]** to complete the calibration calculation with the delay displayed in the input box; or click **[Cancel]** to complete the calibration calculation with the original delay in the input box.



Figure 7.14 Adapter delay confirmation interface

When calibration is completed, the calibration wizard will exit.

7.3.6 Save as User Calibration Set

When you click **[Save as User Cal Set...]** in the Calibration Finish interface, the Save as User Cal Set interface will pop up as shown in Figure 7.15.

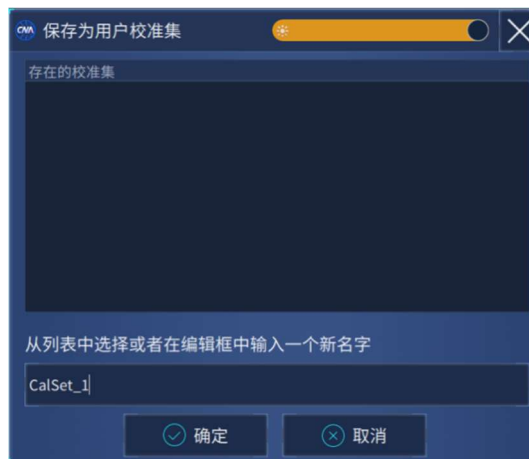


Figure 7.15 Save as user calibration set interface

[Existing Cal Sets] list shows the existing user calibration sets. Double-click to select an existing calibration set, and edit in the box to change the calibration set name. The initial default name displayed in the box is CalSet_x.

Click **[OK]** to save the calibration data to the calibration register and save it as the user calibration set with the name in the edit box. When the saving is completed, the Calibration Wizard will exit directly.

Click **[Cancel]** to exit the Save as User Cal Set interface and return to the Calibration Finish interface.

If the data is saved as an existing user calibration set, a prompt message as shown in Figure 7.16 will pop up.

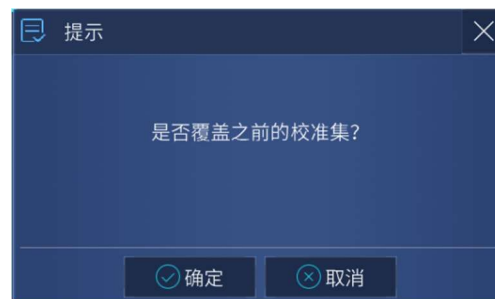


Figure 7.16 Prompt to overwrite previous calibration set

Click **[OK]** to overwrite the previous user calibration set; click **[Cancel]**, return to the Save as User Cal Set interface.

7.4 Calibration Set

The Calibration Set dialog box supports to View, Save As, Delete, and Apply the calibration set files, and supports to modify the file name and description of the calibration set. The Calibration Set dialog box can be opened by clicking Cal and then CalSet on the interface.

7.4 Calibration Set

- Definition of Calibration Set.....232
- Display Calibration Set Properties.....233
- Save Calibration Set As.....234
- Delete Calibration Set.....234
- Delete All Calibration Sets.....234
- Apply/Unapply Calibration Set.....234



Figure 7.17 Open calibration set

7.4.1 Definition of Calibration Set

After the calibration is completed, the calibration data will be saved to the calibration set file. The calibration set file can be applied to any channel with the same stimulus as this file, thus saving time for another calibration.

The calibration set includes channel calibration set (calibration register and user calibration set), between which the channel calibration set will be automatically saved after each calibration, and the user calibration set can be selected by the user to save the data in the user calibration set. Each channel has its own channel calibration set. When a calibration is completed, the channel calibration set will add or overwrite the calibration data according to the calibration method, and the channel calibration set is only applicable to the channel where it is located. The user calibration set can be applied to all channels.



Figure 7.18 Calibration set dialog box

As shown in Figure 7.18, calibration set CH1_CALREG is a channel calibration set, and the others are user calibration sets. The Calibration Set dialog box displays **[Cal Set Name]**, **[Class]**, **[Channels]**, **[Cal Type/Port]**, **[Data Time]** and **[Register]** (for channel calibration set). The measurement class includes Standard, Gain Compression, IM Spectrum, Swept MID, Noise

Figure and Scalar Mixer Converters/Vector Mixer Converter, etc.

[Cal Type/Port] displays the calibration type and the corresponding calibration port. When full n-port calibration is set, the calibration type is displayed as "nP/1,..., n", which means that the calibration type is n-port calibration, and the numbers after the slash are the sequence number of ports involved in the calibration. The enhanced response calibration is displayed as "ER/m,n", where m and n are the sequence numbers of the two ports involved in the calibration. The response and isolation calibration is displayed as "RI(Smn)", where Smn is the S-parameter involved in the calibration. The response calibration is displayed as "R(n)", where n is the sequence number of port involved in the calibration. When the calibration includes source and receiver calibration, a "+" symbol will be displayed after the calibration method, such as 1P /1, which means that the source and receiver calibration is carried out to port 1 together with the single-port calibration.

Functions include **[Save as...]**, **[Properties...]**, **[Delete]**, **[Clear Cal Sets]** and **[Apply/Unselect]**.

7.4.2 Display Calibration Set Properties

As shown in Figure 7.18, select the calibration set and click **[Properties...]** to open the Display/Edit Cal Set Properties dialog box. As shown in Figure 7.19, the Cal Set Properties interface supports to modify the name and description of the calibration set file in the input box, and when a calibration set of standard class is opened, the name, description, GUID, date, active channel number, sweep point, frequency range, sweep type, IF bandwidth, port power and port impedance of the calibration set file are displayed in the Detailed Information column.



Figure 7.19 Calibration set properties dialog box

7.4 Calibration Set

7.4.3 Save Calibration Set As

As shown in Figure 7.18, select the calibration set and click **[Save as...]** to open the Save As dialog box. As shown in Figure 7.20, the user can type the name of the new calibration set to be saved as in the following edit box, or the user can click the existing calibration set, and then the system will automatically fill the name of the selected calibration set in the input box. If the name in the input box is consistent with the name of the existing calibration set, the user can choose to overwrite the existing calibration set or not when clicking OK.

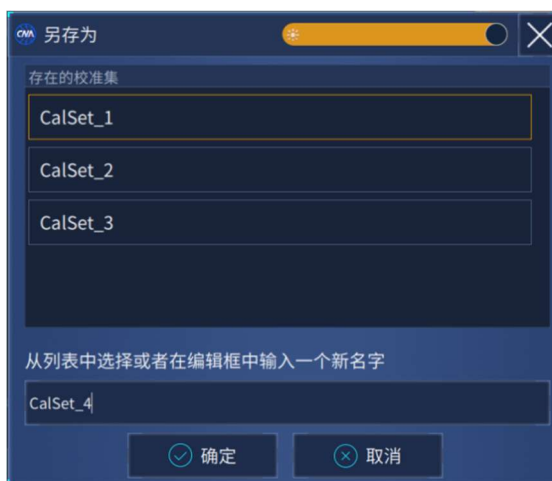


Figure 7.20 Save calibration set as

7.4.4 Delete Calibration Set

As shown in Figure 7.18, select the calibration set and click **[Delete]** to delete the selected calibration set. If the calibration set is being applied, the system automatically cancels the calibration set and removes the calibration set file from the system.

7.4.5 Delete All Calibration Sets

As shown in Figure 7.18, select the calibration sets and click **[Clear Cal Sets]**, and then the system will pop up a prompt reading "Delete all calibration sets?". If the user clicks OK, all calibration sets will be deleted.

7.4.6 Apply/Unapply Calibration Set

As shown in Figure 7.18, when the calibration set is not applied to the current channel, the Channel column of the dialog box will not display the channel number. When you click **[Apply Cal]**, the following four situations may be expected.

- 1) The stimulus settings of the calibration set and the stimulus settings of the current channel are identical or unaffected. The calibration set is applied directly to the current channel, and the currently active channel number is displayed in the Channel column.
- 2) The measurement class of calibration set is inconsistent with that of the channel. The

7.4 Calibration Set

system will directly display an error message reading "The current measurement class is inconsistent with the measurement class of the calibration set, and the calibration set cannot be applied!", and the calibration set will not be applied.

- 3) The stimulus setting of the calibration set is inconsistent with that of the channel, and the frequency range of calibration set is greater than that of the channel. As shown in Figure 7.21, the user can choose not to change the settings of the active channel, at which point interpolation will be performed if necessary. In this case, the user can also choose to change the environment configuration of the active channel to match the calibration set, at which point the stimulus settings of the calibration set will be applied to the active channel.
- 4) The stimulus setting of the calibration set is inconsistent with that of the channel, and the frequency range of calibration set is less than that of the channel. In this case, option 1 shown in Figure 7.21 becomes grayed and cannot be selected, and the user can only change the configuration of the active channel .

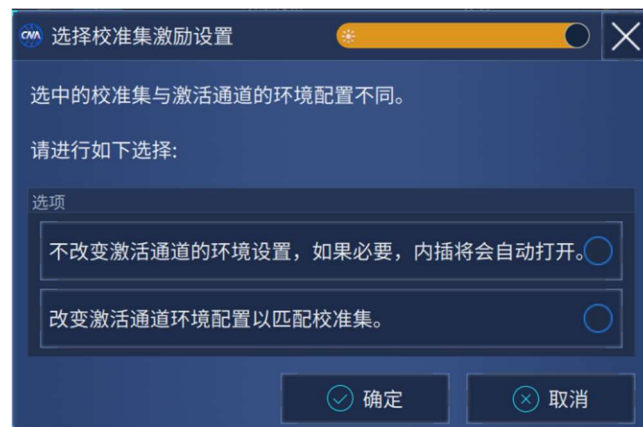


Figure 7.21 Choose stimulus settings

After the calibration set is applied to the active channel, the Channel column of the applied calibration set in Figure 7.18 will display the sequence number of the channel to which the calibration is applied, the calibration set will be selected, and the original Apply button will change to Cancel button. As shown in Figure 7.22, click **[Cancel]**, and the system will clear the calibration data and automatically correct the relationship. When multiple calibration sets are applied repeatedly to the same channel, the last activated calibration set will completely overwrite the previous calibration data.

7.5 Measurement Error



Figure 7.22 Unapply calibration set

7.5 Measurement Error

Understanding the sources of measurement errors and how to correct them will help to improve the accuracy of measurement, because no matter how careful the measurement is, there is still a certain amount of uncertainty. When the analyzer is used for measurement, there are three kinds of errors: drift error, random error and systematic error.

- [Drift Error.....236](#)
- [Random Error.....236](#)
- [Systematic Error.....237](#)

7.5.1 Drift Error

- Drift error results from changes in analyzer or test system performance after calibration.
- The thermal expansion characteristics of the interconnecting cables inside the analyzer and the change of the microwave mixer characteristics are the main causes of drift errors, which can be eliminated by recalibration.
- The test environment determines the holding time of accurate calibration, and a stable ambient temperature can minimize the drift error.

7.5.2 Random Error

Random errors are unpredictable and cannot be eliminated by calibration, but some methods can be used to reduce their impact on the measurement results. Random errors mainly include the following three types:

- 1) **Random noise error of instrument**
 - a) Random noise will be generated due to electrical disturbance of components inside

the analyzer. These electrical disturbances mainly include:

- Low level noise caused by background noise of receiver broadband.
- High level noise or jitter in data trace, mainly caused by the analyzer's internal background noise and LO phase noise.

b) Random noise error can be reduced by the following methods:

- Increase the source power input to the DUT.
- Decrease the IFBW.
- Use the Sweep Average.

2) Switch repeatability error

A switch is used in the analyzer to change the setting of the source attenuator. When the switch acts, the contact closure will be different from the last action closure at times. In this case, the measurement accuracy will be seriously affected. Therefore, in high-accuracy measurement, it is necessary to avoid changing the attenuator setting to reduce the switch repeatability error.

3) Connector repeatability error

The wear of the connector will lead to the change of its electrical performance, and correct maintenance can reduce the repeatability error of the connector.

7.5.3 System Error

Systematic error is caused by the unsatisfactory hardware characteristics of the analyzer. This error is repeatable (therefore predictable) and is assumed not to change over time. Systematic errors can be determined through calibration, and these errors can be eliminated through mathematical calculation during measurement.

The systematic error cannot be completely eliminated. Due to the limitations of the calibration process, there are always some residual errors. The systematic errors left from calibration mainly come from:

- Unsatisfactory calibration standard
- Connector connection
- Interconnecting cable
- Analyzer itself

All measurements are affected by dynamic accuracy and frequency error. For reflection measurement, the relevant residual errors include:

- Valid directivity
- Valid source matching
- Effective reflection tracking

7.5 Measurement Error

For transmission measurements, the relevant residual errors include:

- crosstalk
- Effective load matching
- Effective transmission tracking

1) Directionality error

The analyzer uses a directional coupler or an electric bridge for reflection measurement. Only the reflection signal from the coupling end of the ideal coupler is output to the receiver for measurement. In fact, a small amount of incident signal will leak to the coupling port through the main circuit of the coupler, which will cause directional error during measurement.

The analyzer determines and reduces the directional error by the following methods:

- During calibration, connect the load to the measurement port and assume that the load port is free of reflections.
- The output signal from the coupling port is the leaked error signal.
- The directional error signal is subtracted in the reflection measurement.

2) Crosstalk error

Ideally, only the transmission signal passing through the DUT reaches the receiver. In fact, a small amount of signals reach the receiver through other paths in the analyzer. This part of signals is called crosstalk signals. The analyzer determines and reduces crosstalk errors through the following methods:

- Connect a load to both port 1 and port 2 during calibration.
- The signal measured in the measurement receiver is the leakage signal in the analyzer.
- The crosstalk error is removed by error correction during transmission measurement.

3) Source matching error

Ideally, during reflection measurement, the measurement receiver receives all signals reflected from the DUT. In fact, part of the signals reflected from the DUT are reflected back to the DUT by the measurement port. This part of the signals cannot be measured by the measurement receiver, which will cause source matching error. The analyzer determines and reduces the source matching error by the following methods:

- During calibration, connect a short circuit breaker to the measurement port, the receiver measures the signal reflected from the short circuit breaker, and saves the measured value to the analyzer.
- Connect an open circuit breaker to the measurement port, the receiver measures the signal reflected from the open circuit breaker, and saves the measured value to the analyzer.
- The analyzer compares the measured values with known values of the open circuit

and short circuit breakers to determine the source matching error term.

- The source matching error is removed by error correction during reflection and transmission measurement.

4) Load matching error

Ideally, during transmission measurement, the measurement receiver receives the transmission signals passing through the DUT. In fact, part of the signals are reflected by the test port and cannot be measured, which will cause load matching error. The analyzer determines and reduces the load matching error by the following methods:

- Connect port 1 and port 2 together for zero-length through.
- When the source is at port 1, the measurement signal in receiver A includes the reflected signal of port 2. When the source is at port 2, the measurement signal in receiver B includes the reflected signal of port 1. In this way, the load matching error can be determined.
- The source matching error is removed by error correction during reflection and transmission measurement.

5) Reflection tracking error

Reflection measurement is made by comparing the signals in receiver A to those in receiver R1 or by comparing the signals in receiver B to those in receiver R2, which is called ratio measurement. For an ideal reflection measurement, the frequency response of receivers A and R1 or of receivers B and R2 should be exactly the same. In fact, this is impossible, because this will cause reflection tracking error, which is the vector sum error resulted from test deviations. The magnitude and phase of the error vary with the frequency. The test deviations are mainly caused by the following reasons:

- Signal separation device.
- Test Cables and Adapters.
- Difference between reference and test signal paths.

The analyzer determines and reduces the reflection tracking error by the following methods:

- During calibration, connect a short circuit breaker to the measurement port, the receiver measures the signal reflected from the short circuit breaker, and saves the measured value to the analyzer.
- Connect an open circuit breaker to the measurement port, the receiver measures the signal reflected from the open circuit breaker, and saves the measured value to the analyzer.
- The analyzer compares the measured values with known values of the open circuit and short circuit breakers to determine the source matching error term.
- The reflection tracking error is removed by error correction during reflection and transmission measurements.

7.5 Measurement Error**6) Transmission tracking error**

Transmission measurement is made by comparing the signals in receiver A to those in receiver R2 or by comparing the signals in receiver B to those in receiver R1, which is called ratio measurement. For an ideal transmission measurement, the frequency response of receivers A and R2 or of receivers B and R1 should be exactly the same. In fact, this is impossible, so transmission tracking error will be caused, which is the vector sum error resulted from test deviations. The magnitude and phase of the error vary with the frequency. The test deviations are mainly caused by the following reasons:

- Signal separation device.
- Test Cables and Adapters.
- Difference between reference and test signal paths.

The analyzer determines and reduces the transmission tracking error by the following methods:

- Connect port 1 and port 2 together for zero-length through.
- Measure signals in receivers A and R2 or in receivers B and R1.
- The transmission tracking error is determined by comparing the signals in the two receivers.
- The transmission tracking error is removed by error correction during transmission measurement.

7.5.4 High-accuracy Measurement Calibration

The accuracy of calibration is determined by the selected calibration type, the quality of calibration kits and the calibration process. This section mainly discusses how to carry out high-accuracy calibration.

1) Measurement reference plane

In most of the measurements, the DUT is not directly connected to the port of the analyzer, but are more likely to be connected through the test fixture or cable. To obtain the highest measurement accuracy, you must calibrate at the connection point of the DUT, which is called the measurement reference plane. If calibration is performed on the measurement reference plane, errors related to measurement components (such as cables, test fixtures, and adapters between analyzer ports and the reference plane) are measured and removed by calibration.

2) Effect of using wrong calibration kits

Under normal conditions, the Cal Std in the calibration kit is the same as that of the connector type of the DUT. However, in some cases, there may be no calibration kit of the same type as the connector of the DUT. For example, the specification of DUT port is 2.4mm, but the specification of connector of analyzer and calibration kit is 3.5mm. If the 2.4mm

7.5 Measurement Error

calibration kit is used for calibration and a 2.4mm/3.5mm adapter is connected for measurement, obvious measurement error will be introduced during measurement, especially during reflection measurement, as no adapter is applied during calibration. If the calibration kits used are different from those specified in the calibration process, the calibration accuracy will also be reduced. The degree of reduction depends on the difference between the specified calibration kits and the actual calibration kits.

3) Accuracy of interpolation measurement

When the set status of the VNA is different from that of calibration, the VNA can automatically interpolate calibration data, which will lead to unpredictable measurement accuracy. The measurement accuracy may decline significantly or may not be affected. The measurement error must be determined according to the actual situation. When the phase offset of the two measurement points increases by more than 180° , the measurement accuracy will be significantly reduced because the VNA cannot interpolate the correct phase data. In general, the chance of interpolation leading to a loss of measurement accuracy will increase in the following case:

- When the frequency span between measurement points is increased.
- When the frequency span between measuring points is particularly large.
- When the measurement frequency is very high, especially above 10 GHz.

4) Effect of power level

For the highest error correction accuracy, do not change the power level after performing the calibration. However, by changing the power level in the same state as the attenuator setting during calibration, the accuracy of S parameter measurement will be reduced very little. If the attenuator setting is changed, the accuracy of error correction will be further reduced.

5) System impedance

Menu path: **[Cal]→[System Impedance]**.

Enter the system impedance value in the **Impedance** box.

Notice

When measuring non 50 Ω impedance devices such as waveguide devices, you must select a calibration kit corresponding to the impedance. During calibration, the system sets and calibrates the port impedance according to the impedance of the calibration kit. After calibration, the system impedance is bound to the calibration kit impedance, and the impedance in the System Impedance Setting dialog box is not valid any more.

6) Port Extensions

Connecting additional cables, adapters or fixtures after calibration will lead to the change

7.6 Edit Calibration Kit

of the measurement reference plane and the introduction of additional phase offset. In this case, the port extension function can be used to compensate for the additional phase offset. Port extension is the simplest method to compensate for the additional phase offset between the calibration plane and the DUT plane, but it does not compensate for the loss and mismatch of the path between the calibration plane and the DUT plane. Therefore, the loss and mismatch should be minimized to obtain the highest measurement accuracy. See the “Set port extension” section in “6.5 Improving phase measurement accuracy” for the method of setting the port extension.

7) Perform isolation calibration correctly

In the isolation calibration portion of full dual-port calibration, the crosstalk error between ports is corrected. Isolation calibration is required only when high insertion loss measurements such as filter out-of-band rejection and switch isolation are necessary. When the crosstalk signal is very close to the noise base of the VNA, noise will be introduced into the error model during the isolation calibration measurement. Therefore, in order to improve the calibration accuracy, it is necessary to:

- Perform isolation calibration only if necessary.
- Use narrow IFBW.
- Use sweep averaging to reduce noise.

When carrying out isolation calibration measurement, it is necessary to connect a load at the test port of VNA. In order to obtain the highest calibration accuracy, it is best to connect load at both measurement ports for isolation calibration measurement. If there is only one load, a device with good compatibility can be connected at the non measurement port.

7.6 Edit Calibration Kit

The Edit Calibration Kit dialog box is used to edit the definition of calibration kits or create user-defined calibration kits.

● Define Calibration Kit.....	243
● User-defined Calibration Kit.....	243
● Create Calibration Kit.....	243
● Edit Calibration Kit.....	244
● Edit Calibration Kit Dialog Box.....	246
● Edit Connector Dialog Box.....	247
● Class Information Dialog Box.....	248
● Add Standard Dialog Box.....	249
● Open Circuit Breaker Dialog Box.....	250
● Short Circuit Breaker Dialog Box.....	251
● Load Dialog Box.....	252

- [Thru/Line Dialog Box.....253](#)
- [Databased Standard.....254](#)

7.6.1 Definition of Calibration Kit

Each standard in the calibration kit (represented by a standard ID) is a specific and precisely defined physical device. For example, No.1 in 31121 calibration kit is a 3.5mm male open circuit breaker. There are five basic types of standard devices: open circuit breaker, short circuit breaker, load, through device/air line and data model standard. Each type has a specific model structure, where the digital and physical characteristics of standard parts are described. Most calibration kits have a set of adapters that are phase matched precisely. Calibration standards other than calibration kits can be used, but the characteristics of the standard devices must first be defined.

7.6.2 Self-define Calibration Kit

Sometimes it may be necessary to edit a calibration kit definition file or create a user-defined calibration kit. Although for most measurement applications, the default calibration kit model is sufficient for accurate calibration, but in the following measurement applications, the user needs to create a custom calibration kit:

- The connector type is inconsistent with the definition in the calibration kit model.
- Use standards different from those in the calibration kit definition, for example, a single-port calibration can be performed by three different offset short circuit breakers instead of open circuit breaker, short circuit breaker, and load.
- Improve the accuracy of the predefined calibration kit model. When the model can better describe the actual characteristics of the standard, the calibration will be more **accurate**. For example, the impedance value of the load should be the actual value of 50.1Ω instead of 50Ω in the definition.

When creating user-defined calibration kits, you must first define the types of connectors, such as type N, 3.5mm, 2.4mm, etc. Although more than one connector type is allowed to be defined, it is best to limit each calibration kit to only one connector type. If the connectors are distinguished from female and male connectors, they must be defined separately.

7.6.3 Create Calibration Kit

- 1) Click **[Cal]**, and in **Cal** submenu, click **[Edit Cal...]** to display the **Edit Cal Kit** dialog box.
- 2) Click **[Add...]** to display the Edit Cal Kit dialog box:
 - a) Enter the name of the user-defined calibration kit in the **Cal Kit Name** box.
 - b) Click **[Add Connector...]** in the Connector interface to display **Add Connector** dialog box.
- 3) In the **Add Connector** dialog box, enter the connector name and click OK:

7.6 Edit Calibration Kit

- a) In **Gender** area, select the connector type of the calibration kit to be **[Gendered]** or **[Genderless]**.
 - b) Set the **Min Freq** and **Max Freq** of the connector.
 - c) Enter the characteristic impedance value of the connector in the **Impedance** box, such as 50Ω.
 - d) Select the media type of the connector in the **Connector Media** box: **Coaxial** or **Waveguide**.
 - e) To add a connector of another gender, click **[Add Connector...]** in the connector interface of **Edit Cal Kit** dialog box, and repeat the steps mentioned above to complete the adding of connector.
- 4) In the **Edit Cal Kit** dialog box, click **[Add Std...]** under the **Standard** box to display the **Add Std** dialog box, and select the standard to be added: **[Open Circuit Breaker]**, **[Short Circuit Breaker]**, **[LOAD]**, **[THRU]**, **[LINE]**, **[DATA-BASED]**, and after selection, click **[OK]** to close the **Add Std** dialog box and display the edit dialog box for the selected standard.
 - 5) Complete the input of user-defined data of the standard in the edit dialog box of the **Std** (such as **OPEN**). For the band-limited standard part, its frequency range may be different from the working frequency range of the connector. According to the minimum and maximum frequencies of the standard part, which are input in practice, complete the setting and then click **[OK]** to close the dialog box.
 - 6) Repeat steps 4) and 5) to complete the definition of all standards in the user-defined calibration kits.

7.6.4 Edit Calibration Kit

- 1) Click **[Cal]**, and in **Cal** submenu, click **[Edit Cal...]** to display the **Edit Cal Kit** dialog box.
- 2) Click the calibration kit to be modified in the dialog box.



Figure 7.23 Edit calibration kit dialog box

3) **[Import...]** button

Calls the **Open** dialog box to import the calibration kit definition on the hard disk or other drive.

4) **[Save...]** button

Open the **Save As** dialog box to save the selected calibration kit definition.

5) **[New...]** button

Open the **Edit Calibration Kit** dialog box to create a new calibration kit definition.

6) **[Edit...]** button

Display the **Edit Calibration Kit** dialog box to modify the selected calibration kit definition.

7) **[Delete]** button

Delete the selected calibration kit definition.

8) **[Reset All Calibrators]**

Reinstall all default calibration kits of the vector network analyzer.

7.6 Edit Calibration Kit

7.6.5 Edit Calibration Kit Dialog Box



Figure 7.24 Edit calibration kit dialog box

1) Calibration kit identification zone

- a) **[Calibration Kit Name] box**
Display or edit calibration kit SN and name.
- b) **[Calibrator Description] box**
Display or edit the characteristic description of the calibration kit.

2) Connector type zone

- a) **[Connector] box**
Click the **[Connector]** box or the **Arrow** button to select the connector type.
- b) **[Add Connector...] button**
Click the **[Add Connector...]** button to display the **Add Connector** dialog box to add a new connector type for the calibration kit.



Figure 7.25 Add Connector dialog box

3) Type zone

- a) **[Calibration Method]** box
Click **[SOLT]** box or **[TRL]** box to select the calibration method assigned by class.
 - b) **[Calibration Class]** button
Click the **[Cal Class]** button to display the **Class Information** dialog box to set the class of calibration kits.
- 4) **Standard zone**
- a) **[Add Standard...]** button
Click this button to display the **Add Std** dialog box to define the new calibration standard.
 - b) **[Edit Standard...]** button
Click the **[Edit Standard...]** button to display the Cal Std (**such as open circuit breaker, short circuit breaker, Load, THRU/Air Line**) dialog box to modify the definition of the selected calibration standard.
 - c) **[Delete Criteria]** button
Delete the definition of the selected calibration standard.

7.6.6 Edit Connector Dialog Box



Figure 7.26 Add or Modify Connector dialog box

- 1) **Characteristics zone**
 - a) **[Connector]** box
It is used to define the name of calibration kit connector.
- 2) **Frequency range zone**

7.6 Edit Calibration Kit

- a) **[Minimum Frequency]** box
Define the minimum frequency allowed by the calibration standard during calibration.
 - b) **[Max frequency]** box
Define the maximum frequency allowed by the calibration standard during calibration.
- 3) **Connector type zone**
- Select the gender of the calibration kit connector between **[Gendered]** and **[Genderless]**.
- 4) **Impedance zone**
- Define the characteristic impedance of the calibration standard.
- 5) **Media zone**
- Define the media of the calibration standard: **[Coaxial]** or **[Waveguide]**.
- 6) **Waveguide characteristics**
- Define the cutoff frequency and aspect ratio of the waveguide.

7.6.7 Class Information Dialog Box



Figure 7.27 Class Information dialog box

- 1) **Calibration kit type zone**
- Select the calibration class to edit or view. Classes SA, SB and SC are used for reflection calibration of a single port to calculate directional error, source matching error and reflection tracking error. In general, these three classes will set an open standard, a short standard or a load standard for each port. Classes REVT, UTHRu, FWDT, and REVM are used for

7.6 Edit Calibration Kit

calibration between two ports to calculate the transmission tracking error and load matching error between two ports, and the standards included in these classes are usually through standard or delay standard. Class ISOL is used to calculate the crosstalk error term in two-port calibration.

2) [>>], [<<] buttons

The [>>] and [<<] buttons are used to add or delete the calibration standard for the selected calibration class. Select the calibration standard in the **Unselected Std** box, click the [>>] button to add the selected standard to the **Selected Std** box, select the calibration standard in the **Selected Std** box, and click the [<<] button to delete the calibration standard for the calibration class.

7.6.8 Add Std Dialog Box



Figure 7.28 Add Standard dialog box

Standards supported by the calibration kit include **[Open Circuit Breaker]**, **[Short Circuit Breaker]**, **[LOAD]**, **[THR]**, **[LINE]** and **[DATA-BASED]**.

7.6 Edit Calibration Kit

7.6.9 Open Circuit Breaker Dialog Box

开路标准件

No. 2 名称 OPEN-M-

描述 7-16 male open

接头

接头 7-16 male

频率范围

最小频率 0.000 Hz 最大频率 999.000 GHz

延时特性

延时 0.067 ns 损耗 630.000 Mohms/s

阻抗 50.000 Ω

开路特性

C0 32.000 F(e-15) C1 100.000 F(e-27)/Hz

C2 -50.000 F(e-36)/Hz² C3 100.000 F(e-45)/Hz³

确定 取消

Figure 7.29 Open Circuit dialog box

1) Characteristics zone

- a) **[No] box**
It is used to display or edit the serial number of the standard.
- b) **[Name] box**
It is used to display or edit the name of the standard.
- c) **[Description] box**
It is used to display or edit the characterization of the standard.

2) Frequency range zone

- a) **[Minimum Frequency] box**
Define the minimum frequency at which the open circuit breaker can operate during calibration.
- b) **[Max frequency] box**
Define the maximum frequency at which the open circuit breaker can operate during calibration.

3) Connector zone

Define the open circuit breaker connector, which can be selected in the drop-down box.

4) **Open circuit characteristic zone**

[C0], [C1], [C2] or [C3] are used to define the fringe capacitance of the open circuit breaker.

5) **Delay characteristic zone**a) **[Time Delay] box**

Define the one-way transmission time from the calibration plane to the standard.

b) **[Loss] box**

Define the energy loss (in ohms/S) caused by the skin effect of the calibration standard, and the corresponding frequency is 1 GHz. When calculating the loss, measure the loss and delay of the calibration standard at 1 GHz respectively, and calculate according to the following formula:

$$Loss(\frac{\Omega}{S}) = \frac{Loss(dB) \times Z_0(\Omega)}{4.3429(dB) \times Delay(S)}$$

c) **[Impedance] box**

Define the impedance of the standard.

7.6.10 Short Circuit Breaker Dialog Box

The dialog box is titled "短路标准件" (Short Circuit Standard). It contains the following fields:

- No. 3
- 名称 SHORT-M-
- 描述 7-16 male short
- 接头 7-16 male
- 频率范围: 最小频率 0.000 Hz, 最大频率 999.000 GHz
- 延时特性:
 - 延时 0.067 ns
 - 损耗 630.000 Mohms/s
 - 阻抗 50.000 Ω
- 短路特性:
 - L0 0.000 H(e-12)
 - L1 0.000 H(e-24)/Hz
 - L2 0.000 H(e-33)/Hz^2
 - L3 0.000 H(e-42)/Hz^3

Buttons at the bottom: 确定 (OK), 取消 (Cancel).

Figure 7.30 Short circuit breaker dialog box

7.6 Edit Calibration Kit

1) Short circuit characteristics zone

[L0], [L1], [L2] and [L3] are used to define the residual inductance of the short circuit breaker.

7.6.11 Load Dialog Box

负载标准件

No. 1 名称 LOAD-M-

描述 7-16 male broadband load

接头

接头 7-16 male

频率范围

最小频率 0.000 Hz 最大频率 999.000 GHz

延时特性

延时 0.000 ns 损耗 0.000 ohms/s

阻抗 50.000 Ω

负载类型

固定阻抗 ☐

特定阻抗 ☒

滑动阻抗 ☐

偏移阻抗 ☐

复阻抗

实部 50.000 Ω

虚部 0.000 Ω

确定 取消

Figure 7.31 Load dialog box

1) Load type zone

a) [Fixed Impedance] radio box

When selected, the specified load type is dead load.

b) [Sliding Impedance] radio box

When selected, the specified load type is sliding load.

c) [Specific Impedance] radio box

When selected, the specified load has an impedance value different from the system impedance Z_0 . Only when [Specific Impedance] is selected, the setting of **Complex Impedance** zone is effective.

2) Complex impedance zone

a) [Real] box

Define the real part of the impedance.

b) **[Imaginary]** box

Define the imaginary part of the impedance.

7.6.12 THRU/TL Dialog Box

直通标准件

No. 7 名称 THRU

描述 Insertable thru standard

接头

接头1 7-16 female

接头2 7-16 male

频率范围

最小频率 0.000 Hz 最大频率 999.000 GHz

延时特性

延时 0.000 ns 损耗 2.200 Gohms/s

阻抗 50.000 Ω

确定 取消

Figure 7.32 THRU/Air Line dialog box

1) **Connector zone**

[Connector1] box and **[Connector2]** box define the types of the connectors of the two standard ports.

7.6.13 Databased Standard



Figure 7.33 Databased standard

1) Update standard data zone

[Import File] realizes the reading of standard data files.

7.7. Calibration Standard

This section mainly introduces the basic principle of the calibration kit and some terms in the calibration kit definition file.

1) Cal Kit

A calibration kit consists of a set of physical devices called calibration standards, each of which has a precisely known or predictable magnitude and phase response to frequency. In order for the analyzer to use these definitions of calibration standards, the response values of each standard must be defined mathematically and then organized into calibration classes that correspond to the error correction models used by the analyzer.

2) Calibration standard

The calibration standard provides a reference for the analyzer make error-corrected

7.7. Calibration Standard

measurements. The electrical delay, impedance, loss and other characteristics of each standard have been accurately defined. The analyzer stores these definitions and uses them to calculate the error terms in the error model. During calibration, the analyzer measures the response of these calibration standards. By comparing the measured values with the known values of the model, each error term can be obtained. During measurement, the influence of these error terms on the measurement results is removed by a mathematical operation process called error correction.

3) Calibration standard type

There are four types of calibration standards, corresponding to different standard definition structures or forms. The four types of calibration standards are as follows:

Calibration standard	Terminal impedance
Open circuit breaker	$\infty\Omega$
Short circuit breaker	0Ω
Load	System impedance
Through/Line/Adapter	No termination impedance

4) Definition of calibration standard

The definition of calibration standards describes the electrical characteristics of standards and the frequency range over which they can be used, including:

- a) Minimum frequency
Specifies the minimum frequency that a standard can use during calibration.
- b) Maximum frequency
Specifies the maximum frequency that a standard can use during calibration.
- c) Z0
Specifies the characteristic impedance of a standard, not the characteristic impedance of the system, nor the terminal impedance of the standard.
- d) Delay
Specifies the uniform length of the transmission line between the defined calibration standard and the actual calibration plane.
- e) Type
Specifies the type of calibration standard: open circuit breaker, short circuit breaker, Load, THRU/Air Line/Adapter.
- f) Loss
Specifies the energy loss due to the skin effect.
- g) Definition of loss model
The loss is expressed in ohms/S, and the corresponding frequency is 1 GHz. When calculating the loss, measure the loss and delay of the calibration standard at 1 GHz respectively, and calculate according to the following formula:

7.8 TRL Calibration

$$Loss(\frac{\Omega}{S}) = \frac{Loss(dB) \times Z_0(\Omega)}{4.3429(dB) \times Delay(S)}$$

h) Fringe capacitance model of open circuit breaker

In high frequency band, due to the influence of fringe capacitance, the open circuit breaker rarely has ideal reflection characteristics, and its phase offset changes with frequency, which can not be eliminated. However, the fringe capacitance is defined in the model of the open circuit breaker. The capacitance model is a cubic polynomial as a function of frequency. The coefficient of the polynomial is determined by the actual characteristics of the open circuit breaker. The equation of the fringe capacitance model is:

$$C=(C0)+(C1 \times F)+(C2 \times F^2)+(C3 \times F^3) \text{ (where F is the measurement frequency).}$$

i) Residual inductance model of short circuit breaker

In high frequency band, due to the influence of residual inductance, the short circuit breaker rarely has ideal reflection characteristics, and its phase offset changes with frequency, which can not be eliminated. However, the fringe inductance is defined in the model of the short circuit breaker. The inductance model is a cubic polynomial as a function of frequency. The coefficient of the polynomial is determined by the actual characteristics of the short circuit breaker. The equation of the residual induction model is:

$$L=(L0)+(L1 \times F)+(L2 \times F^2)+(L3 \times F^3) \text{ (where F is the measurement frequency).}$$

7.8 TRL Calibration

TRL (Thru-Reflect-Line) calibration includes a series of calibration techniques, such as TRM (Thru-Reflect-Match). TRL calibration determines the 12-term error coefficient by measuring two transmission standards and one reflection standard, while traditional SOLT calibration determines the 12-term error coefficients by measuring one transmission standard (T) and three reflection standards (SOL).

1) Reasons to do TRL calibration

TRL calibration is very accurate and in some cases more accurate than SOLT calibration, but there are few calibration kits supporting TRL. TRL calibration is often needed when high measurement accuracy is required and there is no calibration kit of the same type as the DUT connector, such as when using a fixture for measurement or using a probe for on-chip measurement. At this time, it is necessary to build and define calibration kits with the same media as the DUT. It is easier to manufacture three TRL standards than four SOLT standards. One disadvantage of TRL calibration is that multiple transmission line standards must be used for broadband calibration. For example, two transmission line standards are required for the 2 GHz ~ 26 GHz frequency band, and in low frequency bands, the transmission line standards will be particularly long.

2) TRL calibration standard

Three standards need to be defined in TRL calibration kits: through standard, reflection standard and transmission line standard.

a) Through Standard

The through standard can be zero length or non-zero length. Zero-length through is more accurate due to the absence of loss and characteristic impedance. The electrical delay of the through standard cannot be the same as that of the transmission line standard. If its phase and electrical length are precisely defined, the through standard can be used to establish a measurement reference plane during calibration.

b) Reflection Standard

The reflection standard can be any physical device with a high reflection coefficient. The characteristics of the reflection standard connected to the two measurement ports must be identical. During calibration, it is not necessary to know the reflection amplitude of the standard device, but the phase must be known, and its electrical length must be within $1/4$ wavelength. If the amplitude and phase of the reflection standard are precisely defined, it can be used to establish the measurement reference plane.

c) Transmission line standard

The transmission line standard is used to establish the measured reference impedance after calibration. TRL calibration has the following shortcomings due to the limitations of the transmission line standard:

- The transmission line standard must have the same impedance as the through standard.
- The electrical length of the transmission line standard cannot be the same as that of the through standard.
- The transmission line standard must have appropriate electrical length in the whole frequency range. At each frequency point, the phase difference between the transmission line standard and the through standard must be greater than 20° and less than 160° , so the actual frequency range that a single transmission line can cover is 8:1. To cover a wider frequency range, multiple transmission line standards are required.
- In the low frequency band, the transmission line standard will be particularly long. The optimal length of the transmission line standard is $1/4$ wavelength of the geometric mean frequency (the square root of the start frequency x the stop frequency) over the frequency span.

d) Matching standard

7.9 Fixture Compensation Calibration

The matching standard can be used in place of the transmission line when it is impossible to manufacture the transmission line according to the required length or loss.

- The matching standard is the low reflection terminal connected to the port.
- In the calculation of the error coefficient for TRL calibration, the matching standard is used as a transmission line with high loss and infinite length.
- The impedance of the matching standard becomes the reference impedance for the measurement.

7.9 Fixture Compensation Calibration

The fixture simulation function provides users with port impedance conversion, two-port fixture de-embedding, matching circuit embedding and four-port fixture embedding/de-embedding functions for networks in a mathematical sense. Users can use it to simulate a wide range of measurement scenarios based on measurement results in production applications, enabling fast, real-time simulation of the corresponding measurement results. Port impedance conversion converts a measurement with a port impedance of 50Ω to a measurement with any impedance value. The dual-port fixture de-embedding function is able to remove the characteristics of any network defined by the Touchstone data file between the measurement end face and the DUT end face, allowing the corresponding calibration face to be extended. The matching circuit embedding function converts the original measurement result into a specific analog result determined under the condition of inserting a matching circuit between the DUT and the test port (single-port). The four-port fixture embedding/de-embedding function enables simulated embedding/de-embedding operation of the four-port fixture to obtain simulation results of the DUT after embedding/de-embedding part of the network.

●	Fixture Simulation Terminologies.....	259
●	Fixture Enable Simulation.....	259
●	Fixture Processing Sequence.....	260
●	Port Extensions.....	261
●	2-Port De-embedding.....	265
●	Matching Circuit Embedding.....	267
●	Port Impedance Conversion.....	271
●	4-Port De/Embedding.....	272
●	AFR.....	275
●	Cal Plane Manager.....	280

The entire fixture simulation process is shown in 0.

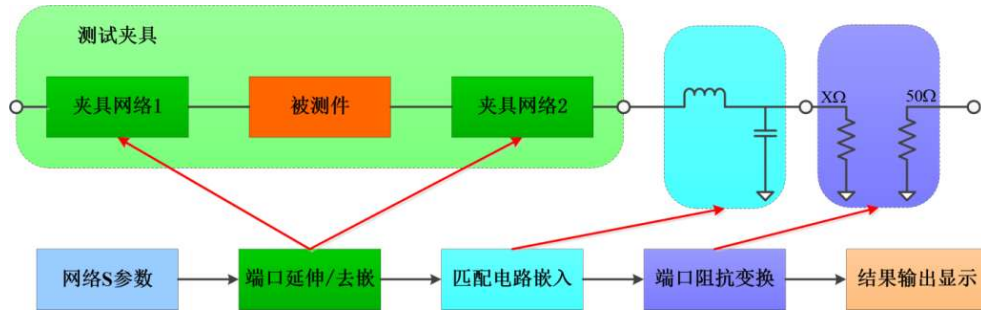


Figure 7.34 Fixture simulation process

Tips

S parameter

To achieve this function, it is required to first obtain the S-parameter of fixture or matching circuit. S-parameter is characterized through circuit model or Touchstone data file. By using the S-parameter of a known fixture, fixture embedding/de-embedding is possible..

7.9.1 Fixture Simulation Terminologies

➤ Fixture

Fixture refers to the device between the DUT and the cable to fix the DUT. The key to fixture embedding and de-embedding is to obtain the S-parameter of the fixture, which, however, is quite difficult. But, there are also some feasible methods currently available, for example, fixture modeling using linear simulation or 3D electromagnetic structure simulation, and application of port extension technology to determine the loss and delay of the fixture.

➤ Touchstone data file (SnP file)

Touchstone data file is a data format created by EESoft before it was acquired by HP, which is more widely known as S2P, and is usually called SnP file when it is extended to multi-port applications with n representing the number of ports, which saves S-parameter data of n ports.

➤ Circuit parameter model

In the microwave device and circuit design stage, it is usually required to perform circuit impedance matching design of the concerning device and circuit. Impedance matching design generally uses a conjugate matching circuit composed of resistance (R), inductance (L), capacitance (C), and conductance (G). In the fixture simulation, the parameter form of the conjugate matching circuit is referred to as the circuit parameter model.

7.9.2 Fixture Enable

Menu path: **[Cal]**→**[Fixture]**. In the **[Fixture]** submenu, click**[Fixturing]**, and check in the box before **[Fixturing]**. At this point, the **Fixture Simulation** function is turned on (only in this state will the subfunctions of the fixture simulation be effective); by default, the box before **[Fixturing]** is

7.9 Fixture Compensation Calibration

empty, indicating that the **Fixture Simulation** function is off.



Figure 7.35 Enable fixture simulation function

7.9.3 Fixture Processing Sequence

Function introduction: When fixture simulation is applied, multiple processing functions will be involved. The default processing flow is as follows: port extensions, 2-port fixture de-embedding, matching circuit embedding, port impedance conversion, and 4-port fixture de-embedding/embedding. The user can change the sequence in the Processing Sequence dialog box.



Figure 7.36 Fixture processing sequence

Menu path: **[Cal]**→**[Fixture]**. In **[Fixture]** submenu, click**[Processing Sequence...]** to display the Processing Sequence dialog box. The fixture processing sequence can be changed by changing the drop-down menu of the corresponding number. If necessary, the default fixture processing sequence can be restored via the Reset button.

7.9.4 Port Extension

Function introduction: The common practice to measure microwave devices is as follows: first calibrate to the coaxial reference surface, and then connect the DUT with adapter or fixture. Due to the electrical delay and loss of the adapter/fixture, the measurement result of the DUT involves error, and the port extensions function can effectively compensate for the measurement error caused by the fixture or adapter.

1) Port Extension dialog box



Figure 7.37 Set Port Extension

- a) **[Select Ports]** drop-down menu
Select the current extension port in the drop-down box. The Port Extension setting works for all measurements under the active channel on the current port.
- b) **[Port Extension (ON/OFF)]**
Turn on/off the port extensions function. The use of this function also depends on the ON/OFF state of **[Fixturing]**. Therefore, it is necessary to further turn on Fixturing after the parameter setting of port extensions function is completed.
- c) **[Time]**
Set the delay duration of port extensions for the current port.
- d) **[Distance]**
Set the physical length of port extensions for the current port.
- e) **[Unit]**
Set the distance calculation unit of port extensions for the current port among meters, feet and inches.
- f) **[DC Loss]**
Set the full band loss offset of the current port.

7.9 Fixture Compensation Calibration

g) **[Loss][@Frequency]**

Set the loss and frequency of the current port. When only [Loss 1] is selected, the network analyzer compensates using the following equation:

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

When both Loss 1 and Loss 2 are selected, compensation is performed using the following equation:

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

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

h) **[Velocity Factor]**

Set the velocity factor of current port. If Couple to System Velocity is enabled for the current port, the velocity factors of other coupled ports are also modified to the current setting.

i) **[Couple to System Velocity]**

Set to couple the velocity factor of current port to the velocity factory of the system

j) **[Coaxial]**

Set the medium characteristic of the current port extension as coaxial characteristic. If Couple to Current Trace Media Definition is enabled for the current port, the media of other coupled ports are also modified to the current setting.

k) **[Waveguide]**

Set the media characteristics of the current port extension to be waveguide. If Couple to Current Trace Media Definition is enabled for the current port, the media of other coupled ports are also modified to the current setting.

l) **[Couple to Current Trace Media Definition]**

Set to couple the media of current port to the media definition of the system

m) **[Cutoff Frequency]**

Set the cutoff frequency of the waveguide when the system media is set to be waveguide.

n) **[Reset]**

Restore the default setting of the port extension parameters for all ports, noting that the port extension enable state remains unchanged.

o) **[Automatic Port Extension...]**

Call the Automatic Port Extension dialog box.

p) **[OK]**

Click to complete the port extension parameter settings and close the Port Extensions dialog box.

q) **[Cancel]**

Click to close the Port Extensions dialog box.

2) **Steps for port extension**

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Port Extensions...]** to display the **Port Extensions** dialog box.

7.9 Fixture Compensation Calibration

Before the introduction of the port extension steps, the functions of port extension is to be introduced according to the following two situations.

- a) If a calibration has been performed, determine whether an additional section of cable is required in the measurement configuration, The port extension feature is used to "tell" the analyzer the length of cable that has been added to the specific port.
- b) When the DUT cannot be calibrated directly in the test fixture, the time delay (phase offset) and loss caused by the fixture can be compensated with port extension.

Steps:

Click **[Select Ports]** drop-down menu, and select the port requiring port extension.

If the physical length of the fixture under test or additional cable is known, simply enter in the **[DIS]** box;

If the electrical length of the fixture under test or additional cable is known, simply enter it in the **[TM]** box;

If both of the above are unknown, the extended reference surface must be available with an open circuit or short circuit breaker instead of the DUT. Obtain the distance or delay time through the automatic port extension function.

Click **[Loss1/2]** and **[@Freq 1/2]** box to enter the frequency point value and the loss value correspondingly.

Click **[Enable1/2]** to select the specific formula used by the network analyzer for compensation.

Click **[Velocity Factor]** input box to set the velocity factor. The default value is 1.

Select the media between **[Coax]** or **[Waveguide]** according to the type of media. When the media selected is waveguide, the cutoff frequency can be set.

Check **[Port Extensions ON/OFF]** in the dialog box to enable the port extension function.

Finally, click **[OK]** to complete the setting.

3) Automatic Port Extension dialog box

7.9 Fixture Compensation Calibration



Figure 7.38 Auto Port Extend

- a) **[Select Ports]:** select port to enable automatic port extension loss and delay time calculation.
 - [Port1]:** select port 1 to enable automatic extension calculation.
 - [Port 2]** Select port 2 to enable automatic extension calculation.
- b) **[OPEN]**
Click [Open Circuit] for measurement when the automatic extension port is connected to the open standard.
- c) **[SHORT]**
Click [Short Circuit] for measurement when the automatic extension port is connected to the short standard.
- d) **[Break]**
Click [Abort] to stop the measurement and calculation during **[OPEN]** or **[SHORT]** measurement of automatic port extension.
- e) **[Include Loss]**
Automatically calculate the loss introduced by the extension of current port.
- f) **[Compensate for Mismatch]**
Automatically compensate the error introduced by the mismatch.
- g) **[Prompt to Connect Calibration Kit]**
Automatically pop up a prompt box for connecting the calibration kit.
- h) **[Default Range]**
Automatically set the frequency range to match the frequency range of current measurement.

- i) **[Activate Cursor]**
Automatically set the frequency range to the frequency between the two active cursors.
- j) **[User Specified]**
The frequency range setting is input by the user. When **[User Span]** is selected, the User Span box is available.
- k) **[Start]**
This box is for user to specify the start frequency.
- l) **[Stop]**
This box is for user to specify the stop frequency.
- m) **[OK]**
Click to complete the automatic port extension setting and measurement, and close the Port Extensions dialog box.
- n) **[Cancel]**
Click to close the Automatic Port Extension dialog box.

4) Steps for automatic port extension

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Port Extensions...]** to display the **Port Extensions** dialog box. In the **[Port Extensions]** dialog box, click **[Automatic Port Extension...]**.

First move the calibration plane to the port plane before the connected adapter or fixture through **[Calibration]** function.

Connect the adapter, transmission line, or fixture, and connect the open circuit or short circuit breaker at the new measurement plane. Normally, not connecting a calibration kit corresponds to an open-circuit state.

In the **[Select Ports]** box, select the port that requires automatic port extension to enable.

Click **[OPEN]** or **[SHORT]** to start automatic port extension calculation. The calculated delay and loss will be automatically displayed in the **[Port Extensions]** dialog box.

Finally, click **[OK]** to complete the setting.

7.9.5 2-port De-embedding

Function introduction: As shown in 0, the dual-port de-embedding function is able to remove the characteristics of any network defined by the Touchstone data file between the measurement end face and the DUT end face, allowing the corresponding calibration face to be extended.

7.9 Fixture Compensation Calibration

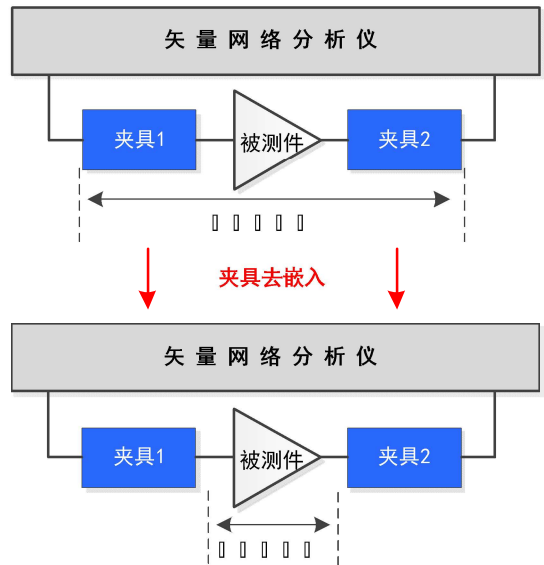


Figure 7.39 Example of fixture de-embedding

1) 2-port Fixture De-embedding dialog box

- a) **[Select Ports]** drop-down menu
Select the port number of the network analyzer requiring fixture de-embedding.



Figure 7.40 2-port fixture de-embedding interface

- b) **[De-embedding Type]** drop-down menu
In the drop-down menu to the right of the **Select Ports** drop-down menu, select the de-embedding type. This network analyzer provides **User** mode for the de-embedding type. When a different type is selected, the corresponding display in example diagram will changes.

- c) **[Load File] button**
This button is ineffective by default. When **[User]** is selected in the **[De-embedding Type]** drop-down menu, this button is enabled. When you click this button, a dialog box prompting you to load the **S2P** file of the fixture S-parameter will pop up. When a file is selected, the **[Load File]** box to the right of the button displays the name of the loaded file. At the same time, in the port and file range display zone of the network analyzer, the maximum/minimum frequency will be updated.
- d) **[Reverse Ports] check box**
This check box is ineffective by default. When **[User]** is selected in the **[De-embedding Type]** drop-down menu, this check box is enabled. When this check box is clicked, the port number of the loaded **S2P** file will be reversed.
- e) **[Fixture De-embedding ON/OFF] check box**
When checking, turn on the two-port fixture de-embedding function.
- f) **[Data Extension ON/OFF] check box**
When this check box is checked, the extrapolation function is turned on.
- g) **[OK] button**
Click **[OK]** to apply the settings in this dialog box to the currently active channels.
- h) **[Cancel] button**
Click **[Cancel]** to exit the 2-port de-embedding function.

2) Steps for 2-port fixture de-embedding

Menu path: Click **[Cal]→[Fixture]**, and in the **[Fixture]** submenu, click **[2-Port De-embedding...]** to display the 2-Port De-embedding dialog box.

Click **[Select Ports]** drop-down menu, and select the port requiring fixture de-embedding.

In the drop-down menu to the right of the **Select Ports** drop-down menu, select the de-embedding type. This network analyzer provides **User** mode for the de-embedding type.

When **[User]** is selected as the de-embedding type, the **[Load File]** button is enabled. Click **[Load File]** and select the **S2P** file to be used in the pop-up dialog box, and then the file name is displayed in the box next to the button. The network analyzer frequency and file frequency below it will be updated.

Click **[Reverse Ports]** check box to custom the reversion of ports in the **S2P** file, and its corresponding example diagram will change accordingly.

Check **[De-embedding ON/OFF]** in the dialog box to turn on the 2-port de-embedding function.

Check **[Extension ON/OFF]** in the dialog box to turn on the extension function.

Finally, click **[OK]** to complete the setting.

7.9.6 Matching Circuit Embedding

Function introduction: As shown in 0, the matching circuit embedding function can convert the

7.9 Fixture Compensation Calibration

original measurement result into a specific analog feature determined under the condition of inserting a matching circuit between the DUT and the test port (single-port). The matching circuit to be inserted is either selected from nine predetermined circuit models or among any circuit defined in the 2-port Touchstone file.

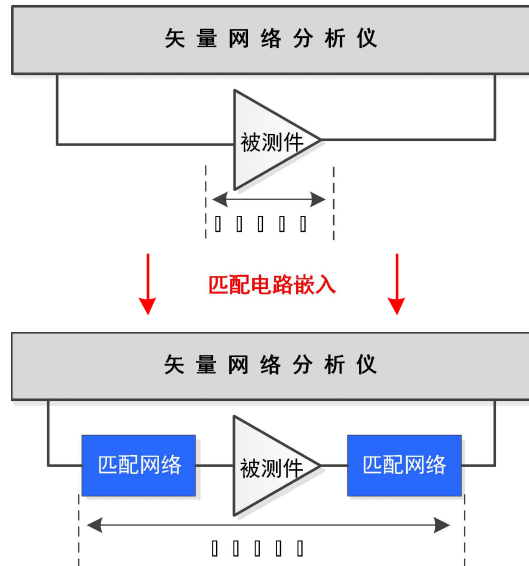


Figure 7.41 Matching Circuit Embedding diagram

1) Matching Circuit Embedding dialog box

- a) **[Select Ports]** drop-down menu
Select the sequence number of port to which the matching circuit is to be embedded.
- b) **[Load File]** button
This button is ineffective by default. When **[User]** is selected in the **[Matching Circuit Type]** drop-down menu, this button is enabled. Click this button, and select the **S2P** file of fixture S-parameter to be loaded in the pop-up dialog box. When a file is selected, the **[Load File]** box to the right of the button displays the name of the loaded file.



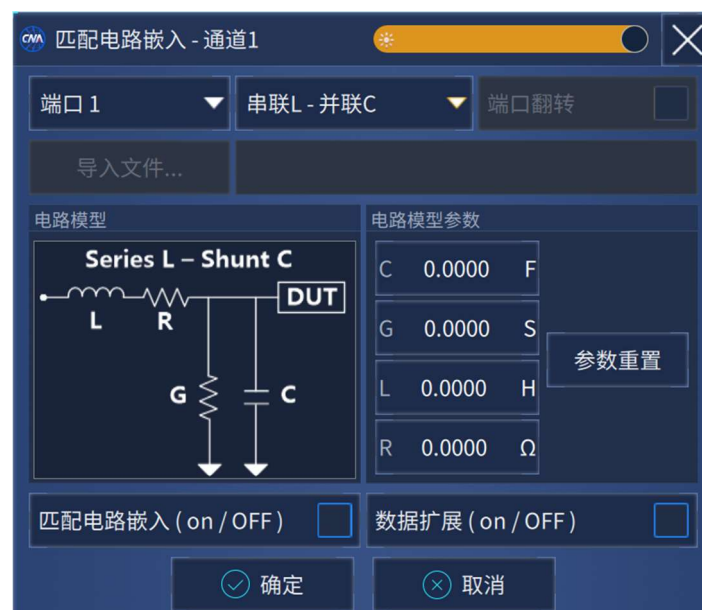
Figure 7.42 Matching Circuit Embedding interface

c) **[Matching Circuit Type]** drop-down menu

Select the type of matching circuit to embed. For the network analyzer, three options, namely **None** (with no circuit embedded), **9 Circuit Parameter Models** and **S2P File** are available. As shown in 0 and 0, when different matching circuit types are selected, the example diagram in the Circuit Model box will be different.

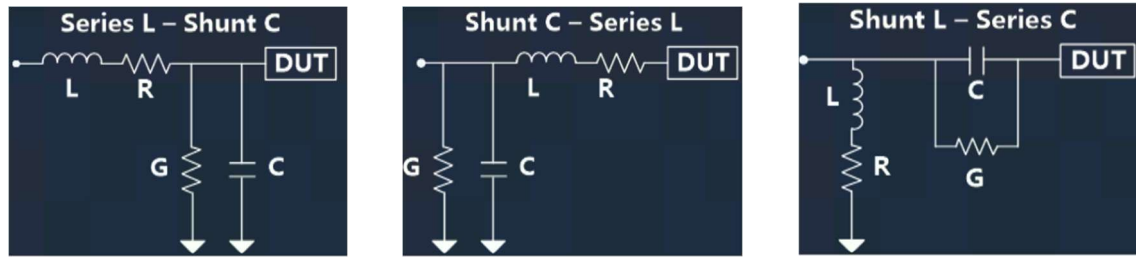
d) **[Reverse Ports]** check box

This check box is ineffective by default. When **[User]** is selected in check box **[Matching Circuit Type]** drop-down menu, this check box is enabled. When this check box is clicked, the port number of the loaded **S2P** file will be reversed.

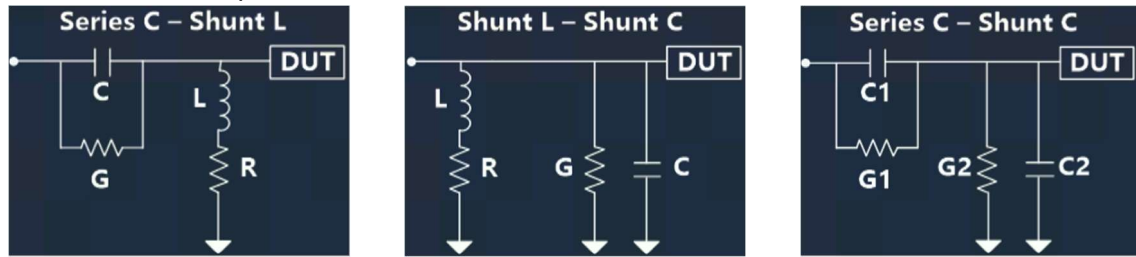


7.9 Fixture Compensation Calibration

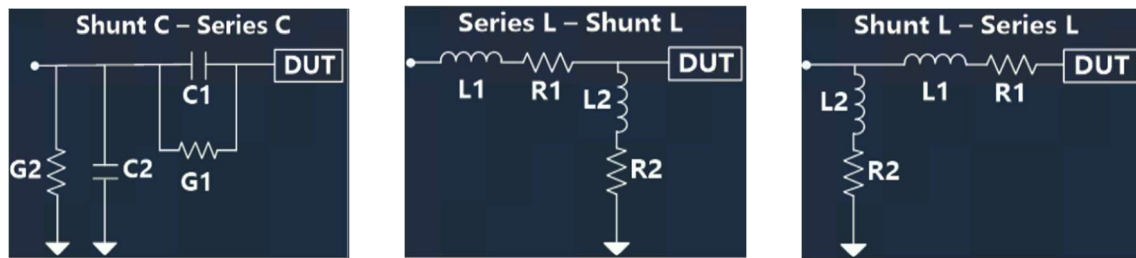
Figure 7.43 Matching Circuit Type drop-down menu



(1) Series inductance-shunt capacitance (2) Shunt capacitance-series inductance (3) Shunt inductance-series capacitance



(4) Series capacitance-shunt inductance (5) Shunt inductance-parallel capacitance (6) Series capacitance-shunt capacitance



(7) Shunt inductance-series capacitance (8) Series inductance-shunt inductance (9) Shunt inductance-series inductance

Figure 7.44 Nine circuit parameter models for matching circuit embedding

- e) **[Circuit Parameter Model Settings]** edit box
Those edit boxes are ineffective by default. When any one of the nine **Circuit Parameter Models** is selected in the **[Matching Circuit Type]** drop-down menu, such edit box is enabled. When the parameters in those edit boxes are changed, the relevant parameter values in the corresponding **circuit parameter model** will change accordingly.
- f) **[Parameter Reset]** button
This button is ineffective by default. When any one of the nine **Circuit Parameter Models** is selected in the **[Matching Circuit Type]** drop-down menu, this button is enabled. Click this button to reset the values in the **[Circuit Parameter Model Setting]** edit boxes.
- g) **[Matching Circuit Embedding ON/OFF]** check box

Check this box to turn on the matching circuit embedding function.

- h) **[Data Extension ON/OFF]** check box

When this check box is checked, the extrapolation function is turned on.

- i) **[OK]** button

Click [OK] to apply the settings in this dialog box to the currently active channels.

- j) **[Cancel]** button

Click [Cancel] to exit the matching circuit embedding function.

2) Steps for matching circuit embedding

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Matching Circuit Embedding...]** to display the Matching Circuit Embedding dialog box.

Click **[Select Ports]** drop-down menu, and select the port requiring matching circuit embedding.

In the drop-down menu on the right of the [Select Ports] drop-down menu, select the type of circuit to be embedded, and the network analyzer is provided with 9 **Circuit Parameter Models** and **User Mode**. The example diagram is shown in the **Circuit Model** box.

When any of the **Circuit Parameter Model** is selected as the circuit type, edit the **capacitance, inductance, resistance** and **conductance** in the corresponding **Circuit Parameter** column. When **[Reset]** is clicked, all parameters will be reset to the default. When **User** is selected as the circuit type, the **[Load File]** button becomes effectively. Click **[Load File]** and select the **S2P** file to be used in the pop-up dialog box, and then the file name is displayed in the box next to the button. At this point, the **[Reverse Ports]** check box becomes enabled, and you can click this check box to custom the reversion of ports in the S2P file.

Check **[Matching Circuit Embedding ON/OFF]** in the dialog box to turn on the matching circuit embedding function.

Check **[Extension ON/OFF]** in the dialog box to turn on the extrapolation function.

Click **[OK]** to complete the setting.

7.9.7 Port Impedance Conversion

Function introduction: Port impedance conversion converts the measured port impedance equal to system impedance to any impedance.



Figure 7.45 Port Impedance Conversion interface

1) Port Impedance Conversion dialog box

7.9 Fixture Compensation Calibration

- a) **[Select Ports]** drop-down menu
Select the port number of the network analyzer requiring port impedance conversion.
- b) **[Real/Imaginary]** edit box
Modifying the parameters in these edit boxes will set the impedance to be converted to.
- c) **[Port Impedance Conversion ON/OFF]** check box
Check this check box to turn on the port impedance conversion function.
- d) **[OK]** button
Click [OK] to apply the settings in this dialog box to the currently active channels.
- e) **[Cancel]** button
Click [Cancel] to exit the port impedance conversion function.

2) Steps for port impedance conversion

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Port Impedance Conversion...]** to display the **port impedance conversion** dialog box.

Click **[Select Ports]** drop-down menu, and select the port requiring impedance conversion.

In **R**(real) and **jX** (imaginary) edit boxes, edit the real and imaginary parts of the impedance to be converted.

Check **[Port Impedance Conversion ON/OFF]** in the dialog box to turn on the port impedance conversion function.

Click **[OK]** to complete the setting.

7.9.8 4-port De-embedding/Embedding

Function introduction: The 4-port fixture embedding/de-embedding function enables simulated embedding/de-embedding operation of the four-port fixture to obtain simulation results of the DUT after embedding/de-embedding part of the network.

1) 4-port De-embedding/Embedding dialog box

- a) **[Select Models]** drop-down menu
Select the fixture de-embedding/embedding model. As shown in 0, the vector network analyzer is designed with 3 network models (i.e. **Model A, B, C**).



Figure 7.46 4-port fixture de-embedding/embedding interface

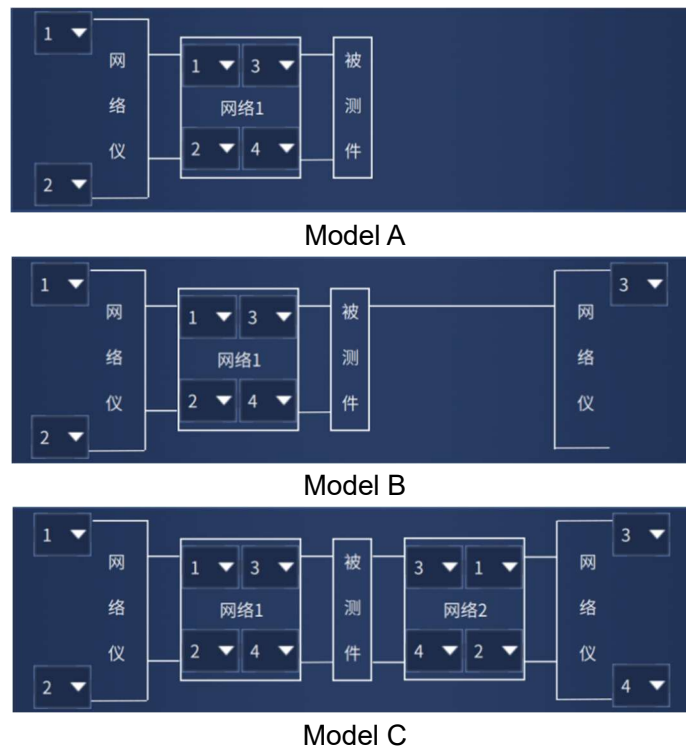


Figure 7.47 3 modes of 4-port fixture de-embedding/embedding

- b) **[Network De-embedding/Embedding Type]** drop-down menu
 The **Network1** (**Network2** is turned on when Model C is selected) (i.e. the first

7.9 Fixture Compensation Calibration

drop-down menu) is for selecting the network de-embedding/embedding type, and 3 options are available, namely **None**, **De-embed** and **Embed**.

c) **[Load File] button**

When you click this button, a dialog box prompting you to load the **S4P** file of the fixture S-parameter will pop up. When a file is selected, the **[Load File]** box to the right of the button displays the name of the loaded file.

d) **[Select Ports] drop-down menu**

Port selection covers ports of network analyzer and ports of fixture. By default, the port selection drop-down menu of fixture is not displayed. The selected network model and network de-embedding/embedding type will be displayed in the example diagram. By selecting the corresponding port correspondence, the de-embedding/embedding port number and the correspondence with the port for file loading are determined.

e) **[4 Port Embedding/De-embedding ON/OFF] check box**

Check the 4-port de-embedding/embedding function of fixture.

f) **[Data Extension ON/OFF] check box**

When this check box is checked, the extrapolation function is turned on.

g) **[Enable ON/OFF] check box**

Check the 4-port de-embedding/embedding function of fixture.

h) **[OK] button**

Click [OK] to apply the settings in this dialog box to the currently active channels.

i) **[Cancel] button**

Click [Cancel] to exit the 4-port de-embedding/embedding function.

2) Steps for 4-port de-embedding/embedding

Menu path: Click **[Cal]→[Fixture]**, and in the **[Fixture]** submenu, click **[4-Port De-embedding/embedding...]** to display the 4-Port De-embedding/Embedding dialog box.

Click **[Select Models]** drop-down menu to select the de-embedding/embedding network model. The vector network analyzer is designed with 3 network models (i.e. **Model A, B, C**).

Select the network de-embedding/embedding type in the **Network1** (**Network2** is turned on when Model C is selected).

When the network is in **Embedding** or **De-embedding** state, **[Load File]** button is enabled. Click **[Load File]** and select the **S4P** file to be used in the pop-up dialog box, and then the file name is displayed in the box next to the button.

Check **[Enable ON/OFF]** in the dialog box to enable the 4-port de-embedding/embedding function.

Check **[Extension ON/OFF]** in the dialog box to turn on the extrapolation function.

Click **[OK]** to complete the setting.

7.9.9 AFR

When the DUT contains non-coaxial connectors, a fixture is generally required to connect the DUT to the network analyzer. In this case, AFR function allows the user to extract and store the parameters of the fixture, and de-embed the fixture to obtain the real parameters of the DUT. During the test, the left and right sides of the DUT are respectively connected with the left and right fixtures and also with the network analyzer. Due to the introduction of the fixture, the measurement error is expected, so the fixture needs to be embedded. The fixture de-embedding function has been introduced in “Fixture Compensation Calibration” of the previous chapter, but the implementation of fixture embedding requires the loading of .snp file of the fixture, and the parameters of the fixture are usually difficult to obtain. AFR can, by measuring the time domain parameters of the fixture and applying the signal flow diagram, accurately extract the fixture parameters, and apply them to fixture embedding.

Notice

Prior to activation of automatic fixture removal, it is required to perform calibration at the coax port of the network analyzer.

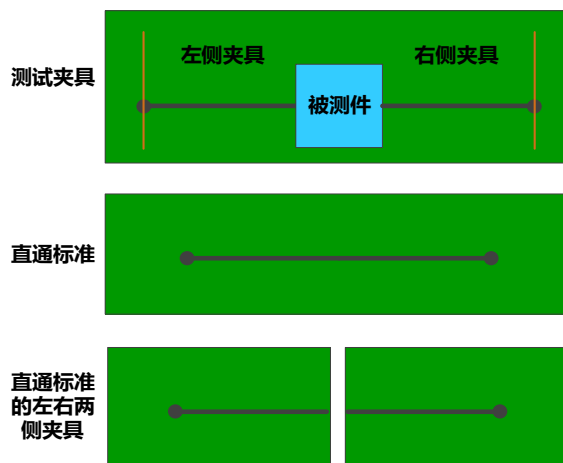


Figure 7.48 Single-ended fixture

7.9 Fixture Compensation Calibration

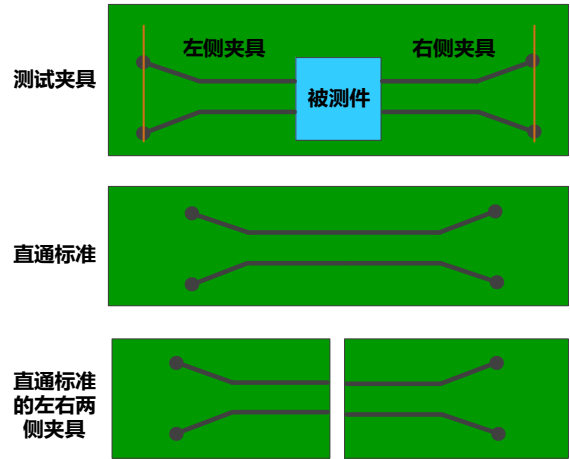


Figure 7.49 Differential fixture

The test fixture usually takes the through configuration or left-and-right configuration, as shown in 0 and 0. Besides, the test fixture and the through standard are usually made of the same material.

Menu path: click **[Cal]**→**[Fixture]**→**[AFR...]** to display the **ARF Wizard** dialog box.



Figure 7.50 Access to the automatic fixture removal function

The specific operation steps are as follows:

- 1) Describe fixture.



Figure 7.51 Fixture description interface

[Fixture Type] includes **[Single Ended]** and **[Balanced]**.

[Fixture Total Ports] includes **[1 Port]** and **[2 Ports]**.

[DUT Impedance] can be set to **[System Z0]**, **[Fixture Z0]** or **[Custom]**. **[System Z0]** means to calculate according to the system impedance of the network analyzer, **[Fixture Z0]** means to calculate according to the measured impedance of the fixture, and **[Custom]** means to calculate according to the impedance set by the user.

[Correct match: L≠R]: it is selected by default. When this option is selected, it suggests that the port matching of the left and right fixtures is inconsistent, and the reflection parameters are to be extracted for left and right fixtures respectively.

[Correct length L≠R]: it is selected by default. When this option is selected, it suggests that the length of the left and right fixtures is inconsistent, and the transmission parameters are to be extracted for left and right fixtures respectively.

After the fixture description is completed, click **[Next]** to enter the Specify Standards interface.

2) Specify the standards.

7.9 Fixture Compensation Calibration



Figure 7.52 Standard description interface

To extract fixture parameters, it is required to measure the fixture standards. In the Specify Standard interface, three fixture standards are available, namely Thru, Short and Open. The through standard is shown in 0 and 0, with the length equal to the sum of the left and right fixtures. When the fixtures connected on the left and right sides are OPEN or SHORT, it will constitute an open standard or a short standard.

After the standards are specified, click **[Next]** to enter the Measure Standards interface.

3) Acquire standard data.



Figure 7.53 Standard Data Acquisition interface

Click **[Load...]** to pop up the .snp file loading dialog box, and the user can read the .snp file data corresponding to the data into the program.



Figure 7.54 Measurement standard dialog box

Click **[Measure...]** to pop up the Measure Standard dialog box shown in 0. The user can connect the fixture standard to the corresponding port of the network analyzer and select the **[VNA Port]**, and click **[Measure]** to start the measurement of fixture standard. After the fixture standard is measured, click **[OK]** to save the measurement data of the fixture standard.

After the data for all fixture standards are obtained, click **[Next]** to enter the Fixture embedding interface.

4) Fixture de-embedding.



Figure 7.55 Fixture de-embedding interface

Connect the fixture and the DUT to the port of network analyzer and configure **[VAN Port]**. Check **[Remove Fixture ON/OFF]** to complete the de-embedding of the current fixture. Click **[Next]** to enter the Save Fixture interface.

5) Save the fixture data.

7.9 Fixture Compensation Calibration



Figure 7.56 Fixture data saving interface

Click **[Browse...]** to change the location where the file is saved, and edit the base name of the file in the **[Base Name]** edit box. The final name if the fixture files is designated as follows: **Base Name_L.snp** and **Base Name_R.snp**. Where: L/R stands for left or right fixture. When the fixture is a single-side fixture, the fixture file is named as **Base Name.snp**.

Three data formats are supported for .snp files: **[Real + Imag]**, **[LogMag + Phase]**, and **[LinMag + Phase]**. The data format can be changed by checking the radio box before the corresponding format, and the default format is **[Real + Imag]**.

[Port Assignment] shows the correspondence relationship of ports for file saving.

Click **[Save Fixture Files]** to save the fixture data.

Click **[Finish]** to exit the AFR setting.

7.9.10 Cal Plane Manager

Fixtures, adapters, or probes are typically used for measurements on a DUT with a non-coaxial interface. The use of additional fixtures makes traditional calibration more difficult. Therefore, it is recommended to choose to mathematically remove (de-embed) the characteristic parameters of the fixture, adapter or probe from the measurement using the calibration plane manager function.

Notice

Definition of adapter/fixture: Any two-port physical device or component that is mathematically removed from a channel measurement or calibration set.

The adapter/fixture to be characterized must be reciprocal ($S_{21} = S_{12}$).

Before characterizing the adapter/fixture, two calibrations must be performed before and after the adapter/fixture and the calibration result shall be saved to the calibration set.

Menu path: Click **[Cal]→[Fixture][Cal Plane Manager...]** to display the **Cal Plane Manager Wizard** dialog box.

As shown in 0, select one of the following options, and then click **Next**:

- Characterize Adapter/Fixture

If calibration has been performed before and after the adapter/fixture, an SNP file characterizing the adapter/fixture will be generated. These files will then be used to de-embed the fixture from the channel or the new calibration set.



Figure 7.57 Cal Plane Manager interface

- Apply Adapter/Fixture
If the SNP files have been saved, they will be used to embed the fixture from the channel or new calibration set.
- Other Actions
 - 1) Characterize Adapter/Fixture
 - a) Characterize Adapter/Fixture dialog box



Figure 7.58 Characterize Adapter/Fixture interface

Prerequisites:

- A tier 1 calibration must have been performed at the input of the adapter/fixture and a tier 2 calibration at the output of the adapter/fixture.
- The calibration results must be saved to the calibration set of network analyzer.

[Fixture Qty]

Click the **[Fixture Qty]** drop-down menu to select the number of fixtures.

If there is a single adapter/fixture on any input or output of the DUT, select One Fixture.

If there are two adapters/fixtures on any input or output of the DUT, select Two Fixtures.

[Select Calset...]

Click **[Select Calset...]** to open the Select Calset dialog box.

[DC Phase...]

Click **[DC Phase...]** to open the DC Phase Settings dialog box.

[Back]

Click to close the Characterize Adapter/Fixture dialog box and return to the Cal Plane Manager dialog box.

[Next]

After the settings of the number of fixture, the calibration set (Tier 1 Canset and Tier 2 Calset) and DC phase (general default value) are completed in sequence, the **[Next]** button is enabled, and clicking the **[Next]** button will enter the Save Fixture Parameters dialog box.

[Done]

In this interface, the **[Finish]** is disabled.

[Cancel]

Click to exit the Cal Plane Manager function.

b) **Cal Set Selection dialog box**



Figure 7.59 Cal Set Selection dialog box

[Filter Cal Set...]

Click **[Filter Cal Set...]** to open the Filter Cal Set dialog box. Select the row of the calibration set in the table, and the name of the selected calibration set will be displayed in the Tier 1 Cal Set zone and Tier 2 Cal Set zone below.

[Reset Cal Set]

After performing filtering of calibration set, the relevant calibration sets will be filtered out and listed in the table. You can click **[Reset Cal Set]** to display all calibration sets.

[Tier 1/2 Cal Set]

Select any calibration set in the Tier 1 Cal Set table at the upper part, and the row where the corresponding calibration set is located will be highlighted, and the calibration set name of the selected row will be displayed here.

[OK]

After the tier 1 calibration set and the tier calibration set are selected, click [OK] and close the Cal Set Selection dialog box.

[Cancel]

Close the Cal Set Selection dialog box.

c) **Filter Cal Set dialog box**

7.9 Fixture Compensation Calibration

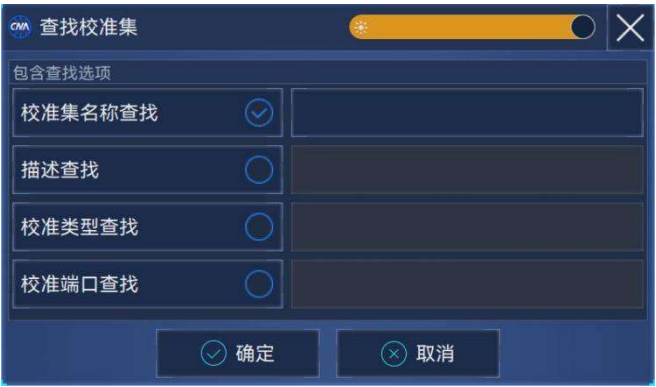


Figure 7.60 Filter Cal Set interface

[Cal Set Name Filter]

To find a calibration set by its name, click the **[Cal Set Name Filter]** radio button and enter the calibration set name in the text box.

[Description Filter]

To find a calibration set by the description, click the **[Description Filter]** radio box and enter the calibration set description in the text box.

[Cal Type Filter]

To find a calibration set by calibration type, click the **[Cal Type Filter]** radio button and enter the calibration set type in the text box.

[Cal Ports Filter]

To find a calibration set by calibration port, click the **[Cal Ports Filter]** radio box and enter the calibration port in the text box.

[OK]

After entering the filtering option, click **[OK]** and close the Filter Cal Set dialog box. The Cal Set Selection dialog box will automatically filter and display the calibration sets.

[Cancel]

Close the Filter Cal Set dialog box.

d) Port Selection dialog box

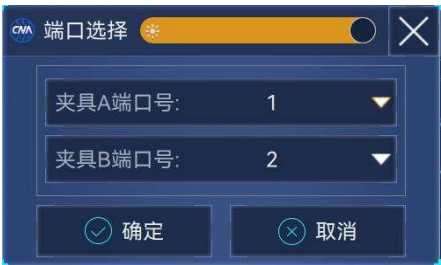


Figure 7.61 Port Selection interface

In the Cal Set Selection dialog box, if a multi-port calibration set is selected for characterizing of fewer ports, it is required to select the port for characterizing adapters/fixtures in the Port Selection dialog box.

[Port of Fixture A]

Select the port corresponding to Fixture A in the common port number of the two calibration sets.

[Port of Fixture B]

Select the port corresponding to Fixture B in the common port number of the two calibration sets.

[OK]

Click to confirm the selected ports of Fixture A and Fixture B and close the Port Selection dialog box.

[Cancel]

Close the DC Phase Pivot dialog box.

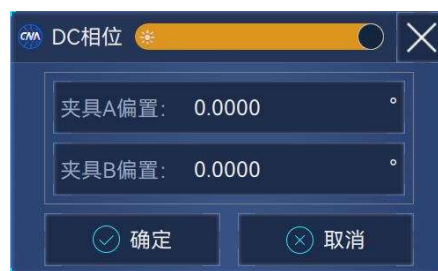
e) DC Phase Pivot dialog box

Figure 7.62 DC Phase Pivot interface

For most adapters/fixtures, the phase projection of S21 in the DC position passes through the X-axis between 0° and -180° . The default value of 0° is suitable for most adapters/fixtures, but in case of long cables, cables with significant mismatch, or high noise in the measurement, the phase projection may be above 0° , which will result in a 180° phase difference between the result calculated by the calibration plane manager and the result measured on the same adapter/fixture using the two-port calibration. In such case, the default value needs to be changed.

[Fixture A Pivot]

This is the input box for pivot of Fixture A.

[Fixture B Pivot]

This is the input box for pivot of Fixture B.

[OK]

Click to confirm the pivots of Fixture A and Fixture B, and close the DC Phase Pivot dialog box.

[Cancel]

Close the DC Phase Pivot dialog box.

f) Save Fixture Characterization dialog box

7.9 Fixture Compensation Calibration



Figure 7.63 Save Fixture Characterization interface

[Select File...]

For each fixture (A and B), click **[Select File...]** to open the Save File dialog box. Select the folder to which the S2P file is to be saved, enter the file name, and click **[Setting]** to select the format for data saving.

[Back]

Click to close the Save Fixture Characterization dialog box and return to the Characterize Adapter/Fixture dialog box.

[Apply]

Click to characterize the fixture and continue to de-embed the fixture.

[Done]

Click to characterize the fixture and close the Save Fixture Characterization dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

2) Apply Adapter/Fixture**a) Port Selection dialog box**

If a S2P file is available for each adapter/fixture, this dialog box allows to eliminate the effects of the adapter/fixture from any of the following locations.

- One or more calibration sets
- One or more channels

[Fixturing]

For each fixture (A and B), click the **[Fixturing]** check box to enable the fixture de-embedding.

[Port Number]

The port number drop-down box allows you to select the port for fixture de-embedding.

[Reverse]

Click the **[Reverse]** check box to reverse the port order of S2P file.

[Select File...]

For each fixture (A and B), click **[Select File...]** to open the Open File dialog box. Select the S2P file for fixture characterization.



Figure 7.64 Apply Adapter/Fixture interface

Then, the text box in the interface will display the file path of the S2P file.

[Back]

Click to close the Apply Adapter/Fixture dialog box and return to the Cal Plane Manager dialog box.

[Next]

After completing fixturing (Fixture A and Fixture B), selection of port number, selection of fixture S2P file, and selection between de-embedding from calsets/channels, the **[Next]** button is enabled, and you can click the **[Next]** button to enter the De-embedding from Calsets/Channels dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

b) **De-embed from Calsets dialog box**

7.9 Fixture Compensation Calibration



Figure 7.65 De-embed from Calsets interface

[De-embed ON/OFF]

Select the specified calibration set to de-embed in the **[De-embed ON/OFF]** column of the table.

[Attribute...]

Click **[Properties...]** to view the relevant properties of the calibration set.

[Back]

Click to close the De-embed from Calsets dialog box and return to the Apply Adapter/Fixture dialog box.

[Next]

In this interface, this button is disabled.

[Done]

When **[De-embed ON/OFF]** of at least one calibration set is enabled, the **[Finish]** button is enabled, and you can click the **[Finish]** button to de-embed the calibration set, and close the De-embed from Calsets dialog box. The file name of the newly embedded calibration set is prefixed with CPM_ before the original calibration set name by default.

[Cancel]

Click to exit the Cal Plane Manager function.

c) **De-embed from Channels dialog box**



Figure 7.66 De-embed from Channels dialog box

[De-embed ON/OFF]

Select the specified channel to enable de-embedding in the **[De-embed ON/OFF]** column of the table.

[Back]

Click to close the De-embed from Channels dialog box and return to the Apply Adapter/Fixture dialog box.

[Next]

In this interface, this button is disabled.

[Done]

When **[De-embed ON/OFF]** of at least one channel is enabled, the **[Finish]** button is enabled, and you can click the **[Finish]** button to de-embed the channel, and close the De-embed from Channels dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

3) Other Actions**a) Other Actions dialog box**

Most of the functions in this interface are for simple processing of SNP files. The functions available in this dialog box mainly include:

- Reverse the Port Order of an S2P File
- Create a Transmission Only S2P File
- Cascade Two S2P Files
- Modify S4P Files
- Write to VNA power sensor loss table

7.9 Fixture Compensation Calibration



Figure 7.67 Other Actions interface

b) Reverse the Port Order of an S2P File dialog box



Figure 7.68 Reverse the Port Order of an S2P File dialog box

This functions enables reverse of port order in an existing S2P file.

- The data of S11 becomes the data of S22, and vice versa.
- The data of S21 becomes the data of S12, and vice versa.

[Select File...]

The **[Select File...]** button corresponding to the original file is used to load the S2P file to be modified. After the file is selected, the text box corresponding to the original file will display the file path of the selected S2P file. The **[Select File...]** button corresponding to the target file is used to save the newly generated S2P file after port reversion, and the text box corresponding to the target file will display the path to save the S2P file.

[Back]

Click to close the Reverse the Port Order of an S2P File dialog box and return to

the Other Actions dialog box.

[Next]

In this interface, this button is disabled.

[Done]

After setting the path of the original file and the target file, click **[Finish]** to reverse the port order of the original S2P file and close the Reverse the Port Order of an S2P File dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

c) **Create a Transmission Only S2P File dialog box**



Figure 7.69 Create a Transmission Only S2P File interface

This function causes the data of S11 and/or S22 column in the existing S2P file to be zeroed, while the original S21 and S12 data are preserved, which will be useful for enhanced response calibration/de-embedding function.

[Select File...]

The **[Select File...]** button corresponding to the original file is used to load the S2P file to be modified. After the file is selected, the text box corresponding to the original file will display the file path of the selected S2P file. The **[Select File...]** button corresponding to the target file is used to save the S2P file of transmission parameters, and the text box corresponding to the target file will display the path to save the S2P file.

[Back]

Click to close the Create a Transmission Only S2P File dialog box and return to the Other Actions dialog box.

[Next]

In this interface, this button is disabled.

[Done]

After setting the path of the original file and the target file, select the S11 check

7.9 Fixture Compensation Calibration

box and/or the S22 check box to reset the corresponding data column to zero. Click **[Finish]** to create the transmission only S2P file and close the Create a Transmission Only S2P File dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

d) **Cascade Two S2P File dialog box**



Figure 7.70 Cascade Two S2P Files interface

This function combines the loss and phase of two S2P files into one S2P file. The stimulus of the two input S2P files are not necessarily to be the same. The frequency range of the cascade files is the intersection of the two input files, and the number of data points is taken as the greater number of data points between the two files. For example:

- S2P #1: Frequency range: 1G ~ 5G, frequency points: 201
- S2P #2: Frequency range: 3G ~ 10G, frequency points: 401
- Cascaded S2P: frequency range: 3G ~ 5G, frequency points: 401

[Select File...]

The **[Select File...]** button corresponding to S2P #1 is used to load the first S2P file to be cascaded. After the file is selected, the text box corresponding to S2P #1 will display the file path of the selected S2P file. The **[Select File...]** button corresponding to S2P #2 is used to load the second S2P file to be cascaded. The **[Select File...]** button corresponding to the cascade S2P file is used to save the cascaded S2P files, and the text box corresponding to the cascaded S2P file displays the path to save the S2P file.

[Back]

Click to close the Cascade Two S2P Files dialog box and return to the Other Actions dialog box.

[Next]

In this interface, this button is disabled.

[Done]

7.9 Fixture Compensation Calibration

After setting the paths of S2P #1, S2P #2, and Casc S2P, click **[Finish]** to cascade the two S2P files and close the Cascade Two S2P Files dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

e) **Modify S4P File dialog box**

This function causes any S-parameter data columns in an existing S4P file to be zeroed.

[Select File...]

The **[Select File...]** button corresponding to the original file is used to load the S4P file to be modified. After the file is selected, the text box corresponding to the original file will display the file path of the selected S4P file.



Figure 7.71 Modify S4P Files interface

The text box corresponding to the original file will display the file path of the S4P file selected. The **[Select File...]** button corresponding to the target file is used to save the modified S4P file, and the text box corresponding to the target file will display the path to save the S4P file.

[Back]

Click to close the Modify S4P File dialog box and return to the Other Actions dialog box.

[Next]

In this interface, this button is disabled.

[Done]

After setting the path of the original file and the target file, select any S-parameter check box (such as S11, S22, S33 and S44) to reset the corresponding data column to zero. Click **[Finish]** to modify the S4P file and close the Modify S4P File dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

7.9 Fixture Compensation Calibration

f) **Write to VNA power sensor loss table dialog box**

This function can write the frequency/loss pairs from the S2P file into the VNA power sensor loss table to compensate for the loss that occurs when the device is used to connect the power sensor to the measurement port during source power calibration.

[Compute Fixture from Calsets]

Select this radio button, and click **[Next]**, and then a S2P file that characterizes the fixture through the Characterize Adapter/Fixture function.

[Use an S2P File]

This radio button can be selected in the case of an S2P file with existing fixture characterization. Then click **[Select File...]** to select the S2P file to be loaded.

[Select File...]

Click this button to load the S2P file. The left edit box will display the S2P file path.

[Enable Power Loss Table]

This check box is for enabling the power loss table .



Figure 7.72 Write to VNA power sensor loss table interface

[Back]

Click to close the Write to VNA power sensor loss table dialog box and return to the Other Actions dialog box.

[Next]

This button is enabled when the **[Compute Fixture from Calsets]** radio button is selected. You can click **[Next]** to pop up the Characterize Adapter/Fixture dialog box.

[Done]

When the **[Use an S2P File]** radio button is selected, the **[Select file...]** button is enabled. After the S2P file selection is completed, Click **[Finish]** to load the S2P file into the Power Loss Table and close the Write to VNA power sensor loss table dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

8 Network Measurement Fundamentals

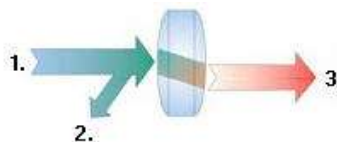
This chapter includes the following contents:

● Reflection Measurement.....	295
● Phase Measurement.....	298
● Amplifier Parameter Description.....	302
● Complex Impedance.....	305
● Group Delay.....	307
● Absolute Output Power.....	311
● AM-PM Transform.....	313
● Linear Phase Deviation.....	316
● Reserve Isolation.....	318
● Small Signal Gain and Flatness.....	320

8.1 Reflection Measurement

Reflection measurement is an important part of network measurement. We first discuss what is reflection measurement. In order to better understand reflection measurement, we use light waves to simulate the propagation of traveling waves along transmission lines:

When the incident light encounters an optical element such as a lens, part of the light is reflected away from the lens, but most of the light continues to transmit through the lens. If the surface of the optical element is a mirror, most of the light will be reflected, and only a small amount of light or no light continues to transmit.



1. Incident light 2 Reflected light 3 Transmitted light

Figure 8.1 Reflection and transmission of light

For RF signals, reflection occurs when the impedance of two connected devices is different. Reflection measurement is to measure the ratio between the reflected signal and the incident signal. The analyzer uses receiver R to measure the incident signal and receiver A to measure the reflected signal. Therefore, the reflection measurement is often expressed as the ratio of A to R (A/R). We can fully quantify the reflection characteristics of the DUT by using the amplitude and phase information of signals from receivers A and R. In the S-parameter terminology, S_{11} represents the reflection from DUT's port 1 (input port), and S_{22} represents the reflection from

8.1 Reflection Measurement

DUT's port 2 (output port). The purpose of reflection measurement is to ensure effective transmission of RF energy. If the energy is reflected, it means that very little energy is transmitted to the desired place. In addition, if the reflected energy is too large, it may burn devices, such as output power amplifier.

- [Expression of Reflection Measurement.....296](#)
- [Summary of Reflection Measurement Expression298](#)

8.1.1 Expression of Reflection Measurement

During reflection measurement, reflection data can be expressed in many ways according to what is desired. Different expressions are calculated from the same reflection measurement data. Each expression can be displayed in one or more graphic formats. For details, see “[4.7 Selecting data format and scale](#)”.

8.1.1.1 Return Loss

The easiest way to represent reflected data is return loss, which is a scalar and expressed in dB. The return loss is the dB value of the reflected signal and the incoming signal. When the impedance is completely matched, the return loss is infinite. For an open circuit, short circuit or lossless reactance circuit, the return loss is 0 dB. For example, when the logarithmic format is used for measurement on the analyzer, the reflection measurement data displayed is -18dB, which means that the device has 18dB return loss, ignoring the minus symbol.

8.1.1.2 VSWR

When two sets of waves are transmitted in the opposite direction on the same transmission line, they will cause a standing wave, which can be expressed by the voltage standing wave ratio (VSWR or SWR for short). SWR, a scalar, is defined as the ratio of the maximum RF envelope voltage to the minimum RF envelope voltage at a given frequency. If the impedance is perfectly matched, SWR is equal to 1. For open, short or lossless reactive circuits, SWR is infinite.

8.1.1.3 Reflection Coefficient

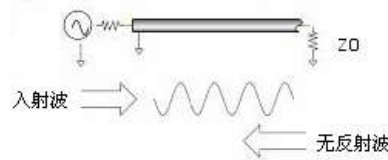
Another way to express reflection measurement is the reflection coefficient (Γ). Γ includes both amplitude and phase information, and the magnitude portion of Γ is called ρ , which ranges from 0 ~ 1. The reflection coefficient is the ratio of the reflected voltage to the incident voltage. If the transmission line is terminated with a characteristic impedance, all energy is transferred to the load, and no energy is reflected, $\rho=0$. If the transmission line is terminated with an open circuit breaker or a short circuit breaker, all energy is reflected, $\rho=1$, and ρ has no unit.

Now we will discuss the phase information of reflection. At high frequency, when the wavelength of the signal is smaller than the length of the conductor, the reflected wave can be

8.1 Reflection Measurement

understood as a wave traveling in the opposite direction to the incident wave. The combination of incident and reflected waves will produce a standing wave, causing the amplitude of the voltage envelope to change with the position of the transmission line.

When the transmission line is terminated with a characteristic impedance, there is no reflected signal, the energy is transmitted in one direction along the transmission line, and all the energy of



the incident signals is transmitted to the load, as shown in the following figure:

Figure 8.2 Signal transmission along a transmission line terminated with load

When the transmission line is terminated with a short circuit breaker, all energy is reflected back to the signal source, and the amplitude of the reflected wave is equal to that of the incident wave ($\rho = 1$), the voltage at both ends of the short circuit breaker is equal to 0, so the voltage wave reflected by the short point is 180° out of phase with the incident voltage wave, and the voltage counteracts each other.

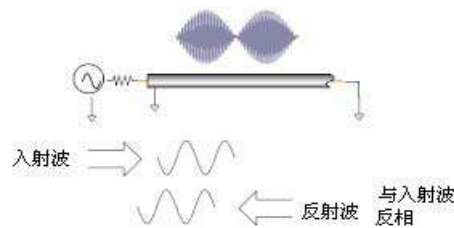


Figure 8.3 Signal transmission along a transmission line terminated with short circuit breaker

When the transmission line is terminated with an open circuit breaker, all energy is reflected back to the signal source, the amplitude of the reflected wave is equal to that of the incident wave ($\rho = 1$), no current flows through the open circuit breaker, and the reflected voltage wave is in phase with the incident voltage wave.

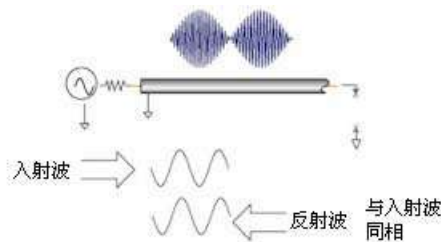


Figure 8.4 Signal transmission along a transmission line terminated with open circuit breaker

When the transmission line is terminated with a $25\ \Omega$ resistor, a portion of the energy is absorbed and a portion is reflected back to the signal source. The amplitude of the reflected

8.2 Phase Measurement

wave is 1/3 of the incident wave, the voltage is 180° out of phase at the resistor, and the phase relationship between the two changes along the transmission line with the distance from the terminating resistor. The trough of standing wave pattern no longer tends to zero, and the wave crest is smaller than those of open circuit breaker and short circuit breaker.

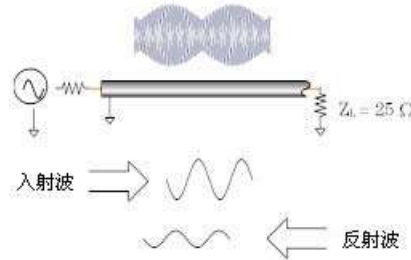


Figure 8.5 Signal transmission along a transmission line terminated with 25 Ω resistor

8.1.1.4 Impedance

Impedance is another way to represent reflection data. For more information about impedance, see "Smith chart format" section in "4.7 Selecting data format and scale".

8.1.2 Summary of Reflection Measurement Expression

Various data expressing reflection are calculated from the same measurement data, and the relationship between them is shown in Figure 8.6:

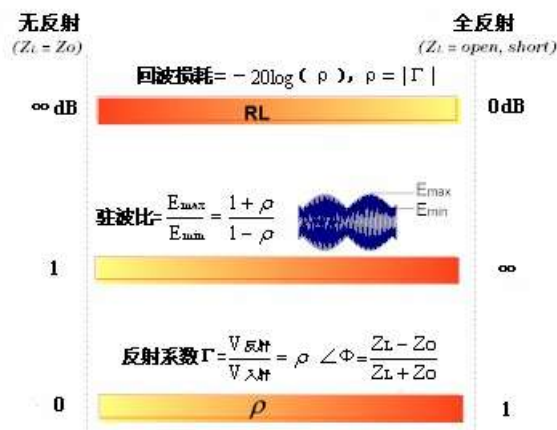


Figure 8.6 Relationship between reflection expressions

8.2 Phase Measurement

8.2.1 What is Phase Measurement

Knowing both the amplitude and phase information of the DUT is very important for a high level of device integration. As with amplitude measurements, phase measurements are also

8.2 Phase Measurement

performed using S-parameters. The phase measurement is a relative (ratio) measurement rather than an absolute measurement. The phase of the incoming device signal (incident signal) is compared with the phase of the device response signal, and the response signal can be either a reflected signal or a transmitted signal. Assuming that the microwave analyzer has been accurately calibrated, the phase difference (i.e. phase offset) between the two signals is a measure of the phase characteristics of the device to be tested. As shown in the following figure, the phase offset between incident signal and transmitted signal can be observed by the oscilloscope.

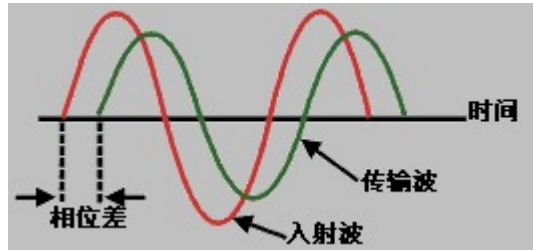


Figure 8.7 Phase offset between signals

8.2.2 Why Phase Measurement is Made

Phase measurement is an important measurement function of a vector network analyzer. Figure 8.8 lists the reasons for making accurate measurements of amplitude and phase.

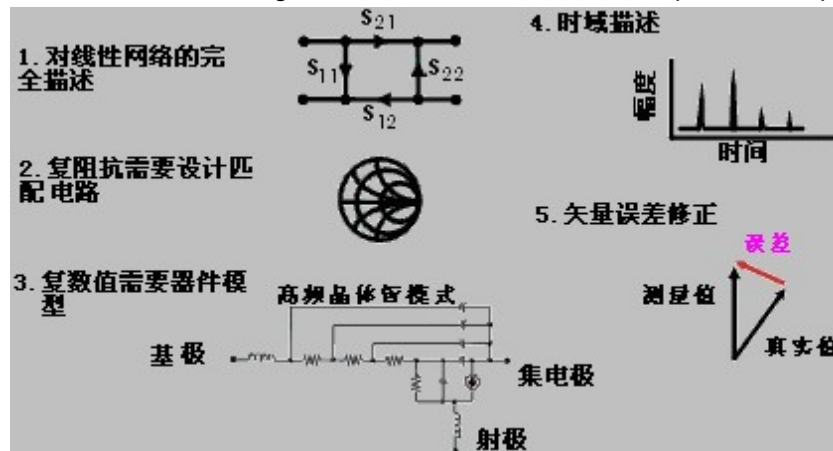


Figure 8.8 Reasons to make accurate measurements of amplitude and phase

When components and circuits are used to transmit signals in the communication system, signal distortions that exceed the normal limit are not allowed. Signal distortion includes the following two categories:

- 1) Linear distortion: It refers to the fact that the amplitude and linear phase offset cannot be maintained in the concerned frequency band as the change of frequency.
- 2) Nonlinear distortion: it means that the circuit will generate new spectral components, such as

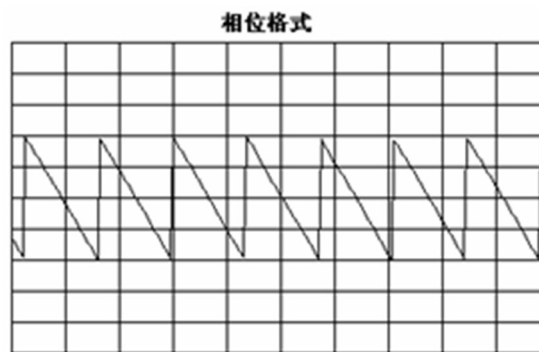
8.2 Phase Measurement

AM-PM conversion.

It is very important to accurately measure the amplitude and phase characteristics of components or circuits to ensure that these circuits can effectively transmit or absorb energy and prevent the transmitted signal from being distorted. Measuring the complex impedance of an antenna is a good example of preventing distortion.

8.2.3 Use the Analyzer's Phase Format

The phase format of the analyzer shows the change of phase with frequency or power. The analyzer cannot measure the phase difference greater than $\pm 180^\circ$ between the reference signal and the response signal. When the phase value varies between $+180^\circ$ and -180° , the analyzer displays the sawtooth phase measurement trace. The sawtooth waveform does not always reach $+180^\circ$ and -180° because the measurements are carried out at discrete frequency points, and the



data points at $+180^\circ$ and -180° may not be in the sweep measurement points.

Figure 8.9 Phase format of analyzer

8.2.4 Type of Phase Measurement

- 1) Complex impedance: complex impedance data such as resistance, reactance, amplitude and phase can be determined by measuring S11 and S22, and can be observed by using Smith chart and polar format. For details, see [“8.4 Complex impedance”](#).
- 2) AM-PM conversion: the AM-PM conversion is to measure the unwanted phase deviation caused by the system amplitude margin. It is defined as the change in output phase when the power input to the amplifier increases by 1 dB, expressed in $^\circ/\text{dB}$, and measured at the 1 dB gain compression point. For details, see [“8.7 AM-PM conversion”](#).
- 3) Linear phase deviation: it measures the phase distortion produced by the DUT. Ideally, the phase offset of the DUT is linear with the frequency, and the deviation from the theoretical phase deviation is called linear phase deviation (also known as phase linearity). For details, see [“8.9 Linear phase deviation”](#).
- 4) Group delay: as another method to measure the phase distortion of devices, group delay

8.2 Phase Measurement

is to measure the transit time of a frequency-specific signal passing through the device. The analyzer calculates the group delay according to the derivative of the measured phase response. For details, see “[8.5 Group delay](#)”.

8.2.5 Linear Phase Offset and Group Delay

Both linear phase deviation and group delay are to measure the phase information of devices, but they are used for different purposes.

- 1) Advantages of linear phase deviation measurement:
 - a) The noise is smaller than that produced in group delay measurement.
 - b) It can better characterize the characteristics of the device that transmits phase-modulated signals. In this case, it is more appropriate to take the phase as the unit than the second as the unit.
- 2) Advantages of group time delay measurements:
 - a) Compared to linear phase deviation measurement, it is easier to explain phase distortion.
 - b) It can most accurately characterize the DUT. Because the analyzer calculates the slope of the phase ripple when measuring the group delay, the slope of the phase ripple depends on the number of ripples per unit frequency. Phase responses with the same peak-to-peak phase ripple are compared. A larger phase slope response will lead to greater group delay variation and greater signal distortion.

8.3 Amplifier Parameter Description

This section discusses the following measurement parameters of amplifier:

8.3.1 Gain

The gain is the ratio of the amplifier's output power (to the characteristic impedance load) to the input power (from the characteristic impedance source), which can be calculated by the following formula:

$$\tau = \frac{V_{\text{trans}}}{V_{\text{inc}}}$$

$$\text{Gain (dB)} = -20 \log_{10} |\tau|$$

$$\text{Gain (dB)} = P_{\text{out}} \text{ (dBm)} - P_{\text{in}} \text{ (dBm)}$$

In the case of small signal, the amplifier's output power is proportional to the input power, and the small signal gain is the gain in the linear region. As the input signal power level gradually increases, the amplifier enters a saturation state, and the output power reaches the limit, which causes the gain to decrease. The large signal gain is the gain in the nonlinear region. For more details, please refer to "[Annex 5 Grain Compression Measurement of Amplifier](#)".

8.3.2 Gain Flatness

Gain flatness refers to the gain variation within the working frequency of the amplifier. For details, see "[8.10 Small signal gain and flatness](#)".

8.3.3 Reserve Isolation

The reverse isolation is to measure the transmission between the output and the input, which is similar to the gain measurement, except that the stimulus signal is applied to the output terminal of the amplifier. For details, see "[8.9 Reverse isolation](#)".

8.3.4 Gain Drift Change (Temperature, Bias) with Time

It refers to the maximum change in gain over time when all other parameters remain unchanged, and is a function of time. It can also measure the gain drift related to other parameters, such as the gain shift with temperature, humidity and bias voltage.

8.3.5 Linear Phase Deviation

It refers to the deviation from the linear phase offset. Ideally, the phase offset of the amplifier is a linear function of frequency. For details, see "[8.8 Linear phase deviation](#)".

8.3.6 Group Delay

The group delay is to measure the transit time of a signal passing through an amplifier and is a function of frequency. The ideal linear phase offset has a constant rate of change with frequency, and the group delay is a constant at this time. In actual measurement, the group delay is calculated by the following formula:

$$\begin{aligned}\tau_g (\text{sec}) &= - \frac{\Delta \theta}{\Delta \omega} \\ &= - \frac{1}{360} * \frac{\Delta \theta}{\Delta f}\end{aligned}$$

For details, see “8.5 Group delay”.

8.3.7 Return Loss (VSWR, ρ)

The return loss is to measure the reflection matching of the amplifier relative to the system impedance Z_0 at the input or output port, and it is calculated by the following formula:

$$\begin{aligned}\Gamma &= \frac{V_{\text{refl}}}{V_{\text{inc}}} = \rho \angle \theta \\ \text{Reflection coefficient} &= \rho \\ \text{Return loss (dB)} &= -20 \log_{10} \rho \\ \text{SWR} &= \frac{1+\rho}{1-\rho}\end{aligned}$$

8.3.8 Complex Impedance

The magnitude of the energy reflected by the device is related to the complex impedance of the device. The complex impedance is composed of resistance component and reactance component, which can be obtained from the system characteristic impedance and reflection coefficient, as shown in the following formula:

$$\begin{aligned}Z &= \frac{1+\Gamma}{1-\Gamma} * Z_0 \\ &= R + jX\end{aligned}$$

For details, see “8.4 Complex impedance”.

8.3.9 Gain Compression

The amplifier has a linear gain region in which the gain is independent of the input power level (small signal gain). When the power level increases to a certain value, the amplifier will enter a saturated state, thus reducing the gain.

8.3 Amplifier Parameter Description

The gain compression is determined by measuring the 1 dB gain compression point of the amplifier, which refers to the output power when the gain of the amplifier decreases by 1 dB relative to the small signal gain. Usually, this measured value is used to describe the power output capability of the amplifier. For more details, please refer to [“Annex 5 Grain Compression Measurement of Amplifier”](#).

8.3.10 AM-PM Conversion Coefficient

AM-PM conversion is to measure the change in output signal phase caused by the amplitude margin of amplifier input signal. The AM-PM conversion is expressed in “°/db”, which is measured at 1 dB gain compression point. For details, see [“8.7 AM-PM conversion”](#).

8.4 Complex Impedance

During S11 or S22 measurement of the tested part, the complex impedance data, such as series resistance, series reactance, and the phase and amplitude information of impedance, can be observed. The complex impedance data can be observed using Smith chart or polar format.

8.4.1 What is Complex Impedance

The complex impedance data can be determined by the S11 or S22 measurement results of the DUT. The magnitude of the reflected power of the DUT is related to the impedance value of the device and the measurement system. For example, the complex reflection coefficient (Γ) is 0 (that is, the power transmission efficiency from the source to the load is the highest) only when the device and system impedances are exactly the same. Each value of Γ corresponds to a unique complex impedance of the device, which is a function of frequency.

$$Z_L = [(1 + \Gamma) / (1 - \Gamma)] * Z_0$$

Where Z_L is the impedance of the DUT and Z_0 is the characteristic impedance of the measurement system. Smith chart and polar format are most suitable for impedance measurement. For details, see "Smith chart format" and "polar format" in "4.7 Selecting data format and scale".

8.4.2 Improve the Accuracy of Impedance Measurement

- 1) The Smith chart format is the easiest to understand when the full scale value is 1.
- 2) When the cursor is used in Smith chart or polar format, higher measurement accuracy can be obtained in discrete cursor mode.
- 3) The uncertainty of reflection measurement is affected by the following factors:
 - a) Directivity
 - b) Reflection tracking
 - c) Source matching
 - d) Load matching

The influence of these factors can be reduced when a full dual-port calibration is used, and a single-port calibration can provide the same measurement accuracy if the output port of the DUT is connected to a high-quality load.

8.4 Complex Impedance

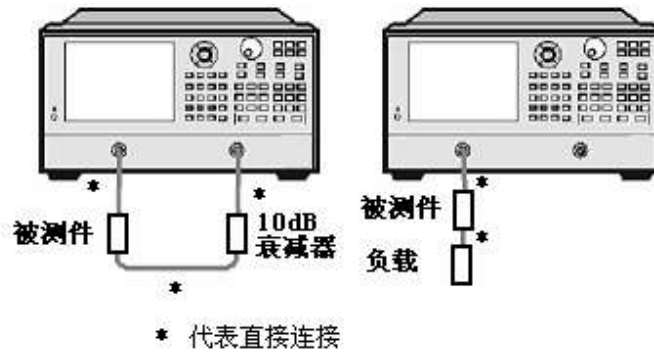


Figure 8.10 Impedance measurement connection

- 1) If the DUT is connected between the two ports of the analyzer after the single-port calibration, it is recommended to connect a 10dB attenuator at the output port of the DUT to improve the measurement accuracy. If full dual-port calibration has been carried out, it is not necessary to connect this attenuator, because the load matching error has been corrected in the full dual-port calibration.
- 2) If only a dual-port device is connected to one port of the analyzer after the single-port calibration, a high-quality load (such as the load standard in the calibration kit) must be connected to the output port of the device.

8.4.3 Measurement Steps of Complex Impedance

- 1) Reset the analyzer.
- 2) Set up and calibrate the analyzer.
- 3) Connect devices as shown in Figure 8.10 above and carry out S11 or S22 measurement.
- 4) Observe impedance data:
 - a) Select the Smith chart format.
 - b) Scale the displayed measurements for optimal viewing.
 - c) Turn on the cursor and move the cursor along the trace to read the resistance and reactance components of the complex impedance at the desired the data point.
 - d) Print or store data.
- 5) Observe admittance data (if necessary):
 - a) Turn on the cursor and select the cursor format as $G+jB$ to read admittance data.
 - b) Scale the displayed measurements for optimal viewing.
 - c) Move the cursor along the trace to read the conductance and susceptance components of the complex admittance at the desired data point. The conductance and susceptance are expressed in Siemens (S).
 - d) Print or store data.

6) Observe the amplitude and phase of the reflection coefficient:

- a) Select Smith chart or polar coordinate format.
- b) Turn on the cursor and select the cursor format as logarithmic/phase or linearity/phase.
- c) Scale the displayed measurements for optimal viewing.
- d) Move the cursor along the trace to read the amplitude and phase information of the desired data points.
- e) Print or store data.

8.5 Group Delay

Group delay is a method to measure the phase distortion of a DUT. It is the actual transit time of the signal passing through the DUT, which varies with frequency. When specifying group delay metrics, be sure to specify the aperture used for group delay measurements.

8.5.1 What is Group Delay

The group delay is:

- 1) A measure of device phase distortion.
- 2) The relationship between the transit time of signal passing through a device and the frequency.
- 3) The derivative of the phase characteristic of a device with respect to the frequency.

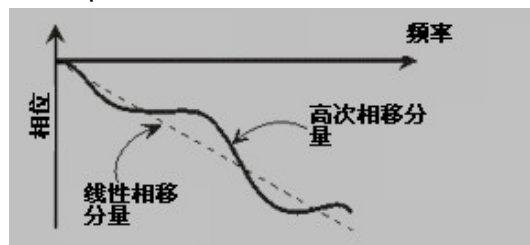


Figure 8.11 Phase offset and group delay

$$\text{Group Delay} = \frac{-d\phi}{d\omega} = \frac{-1}{360^\circ} \cdot \frac{d\theta}{df}$$

ϕ is in radians, ω is in radians per second, θ is in degrees, and f is in hertz.

The phase characteristics of the device include linear phase offset component and high-order phase offset component:

- 1) Linear phase offset component: represents the average transit time of the signal and indicates the electrical length of the device.
- 2) High-order phase offset component: represents the change of transit time with frequency, which is the cause of signal distortion.

8.5 Group Delay

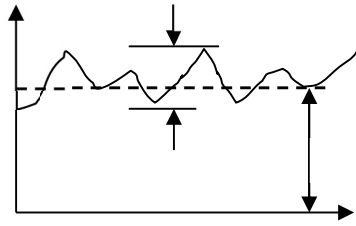


Figure 8.12 Group delay

In group delay measurement:

- 1) The linear phase offset component represents the average group delay.
- 2) The high-order phase offset component represents the deviation from the average group delay (group delay ripple).
- 3) Just as linear phase deviation will cause distortion, group delay margin will also cause signal distortion.
- 4) The group delay measurement trace shows the time taken for each frequency signal to pass through the DUT.

The analyzer calculates the group delay according to the following method:

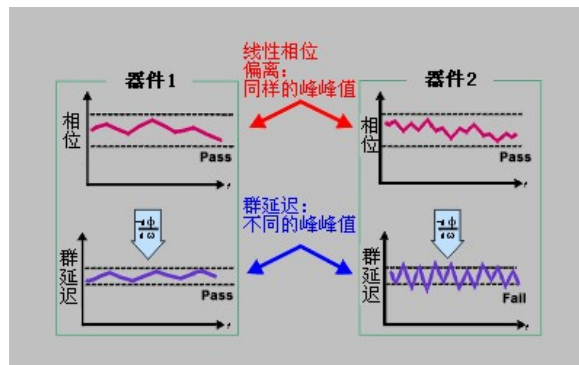
$$\text{Group Delay} = \frac{-d\phi}{d\omega} = \frac{-1}{360^\circ} \cdot \frac{d\theta}{df}$$

ϕ is in radians, ω is in radians per second, θ is in degrees, and f is in hertz.

- 1) Calculate the phase change ($-d\phi$) from the phase data.
- 2) The specified frequency aperture is a function of frequency ($d\omega$).
- 3) Using the above two values, calculate an approximation of the rate of phase change with frequency. This approximation is the group delay (assuming that the phase changes linearly within the specified frequency aperture).

8.5.2 Reason for Measuring Group Delay

Group delay can display phase distortion information more accurately than linear phase



deviation, as shown in Figure 8.13:

Figure 8.13 Comparison of group delay and linear phase deviation

- 1) The upper part of the figure shows the measurement results of the linear phase deviation of device 1 and device 2. Although the shape of the traces is different, the linear phase deviation of the two devices is the same.
- 2) The lower part of the figure shows the group delay measurement results of device 1 and device 2. The group delay of the two devices is different because the analyzer calculates the group delay from the slope of the phase ripple, which is determined by the number of ripples per unit frequency.

8.5.3 What is Group Delay Aperture

To measure group delay, the analyzer measures the phase at two adjacent frequency points, and then calculates the phase slope. The frequency interval (frequency difference) between two phase measurement points is called aperture. Different group delay values can be obtained by changing the aperture, which is why the aperture used in the measurement must be known when comparing the group delay data.

The default group delay aperture of the analyzer is the frequency interval between two adjacent sweep points. The aperture size can be changed by the following two methods:

- 1) Change the number of measurement points or frequency span
 - a) Increase the number of points or decrease the frequency span to narrow the aperture.
 - b) Reduce the number of points or increase the frequency span to widen the aperture.

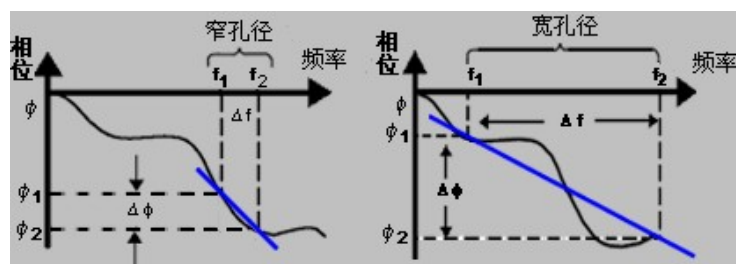


Figure 8.14 Effect of group delay aperture on measurement results

Notice

Relationship between aperture and group delay

If the aperture is too wide and the phase offset between adjacent frequency points is greater than 180° , the calculation of group delay will be wrong.

- 2) Use the smoothing function of the analyzer
Perform a single sweep, turn on the smoothing function, and change the percentage of

8.5 Group Delay

the value range in the smoothing setting, which has the same effect as changing the frequency interval between sweep points. This method allows a wider group delay aperture, and a phase offset of more than 180° can occur within the smooth aperture.

Group delay measurement can be carried out under the following sweep types:

- a) Linear frequency
- b) Logarithmic frequency
- c) Segment sweep

The group delay aperture depends on the frequency interval and the density of sweep points, so the group delay aperture varies at logarithmic frequency and segment sweep.

8.5.4 Improve the Measurement Accuracy of Group Delay

During group delay measurement, it is very important to ensure that the phase difference between adjacent measurement points is less than 180° , otherwise incorrect phase and group delay information will be obtained, as shown in Figure 8.15). Under-sampling may occur during the measurement of long electrical delay devices. The following settings can be adjusted until the measurement trace is no longer changed to ensure that the phase difference between two adjacent points is less than 180° .

- 1) Add the number of points
- 2) Decrease the frequency span.

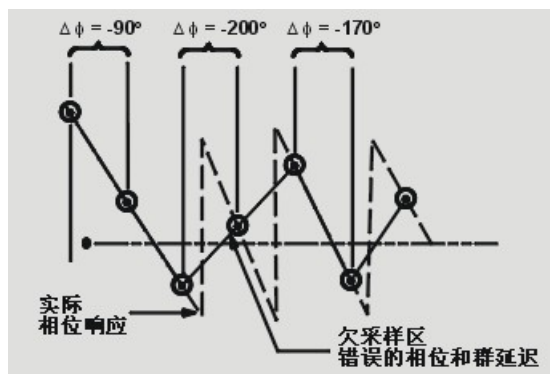


Figure 8.15 Effect of under-sampling on group delay measurement

Frequency response is the main error in group delay measurement. Through response calibration can significantly reduce this error. For higher accuracy, full dual-port calibration should be performed. For amplifiers, the response may vary with temperature, and the test shall be carried out with the amplifier at the required operating temperature.

8.5.5 Measurement Steps of Group Delay

- 1) Reset the analyzer. If the DUT is an amplifier, the source power of the analyzer must be adjusted:
 - a) Set the source power of the analyzer in the linear region of the amplifier response

8.6. Absolute Output Power

(usually at least 10 dB lower than the input 1 dB compression point).

- b) If necessary, an attenuator shall be connected to the output port of the amplifier to fully attenuate the output power of the amplifier so as to avoid compression or burnout of the receiver at the receiving port of the analyzer.
- 2) Connect the DUT, as shown in Figure 8.16;

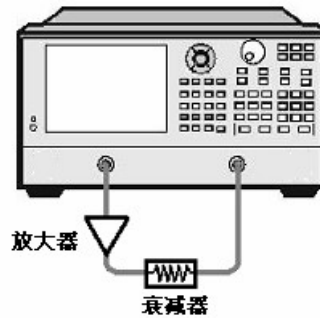


Figure 8.16 Measurement connection of group delay

- 3) Select S21 measurement.
- 4) Select settings for the DUT, including:
 - a) Format: phase;
 - b) Scale: auto scale
 - c) Number of measurement points: select the appropriate number of points to ensure that no under-sampling occurs.
- 5) Remove DUT and perform calibration.
- 6) Reconnect the DUT.
- 7) Set the display format to group delay, and set the scale for measurement display to obtain optimal viewing.
- 8) The smoothing function of the analyzer is used to increase the aperture, reduce the noise on the trace, and maintain meaningful details. To increase the aperture, the following steps can be taken:
 - a) Turn on the smoothing function of the analyzer;
 - b) Change the smooth aperture (maximum 25% of the frequency span).
- 9) Use the cursor to read the group delay at the frequency of interest.
- 10) Print or save data.

8.6. Absolute Output Power

Absolute output power is a measurement that shows the relationship between absolute power (dBm or W) and frequency.

8.6. Absolute Output Power

8.6.1 What is Absolute Output Power

It refers to the power present on the receiving port of absolute output power measurement analyzer. This power is absolute power, not referenced (or ratioed) to incident power or source power. At this time, in the logarithmic amplitude format, the value associated with the vertical axis of the grid is expressed in dBm, and dBm is the power measured with 1 mW as the reference:

$$0\text{dBm} = 1\text{mW}$$

$$-10\text{dBm} = 0.1\text{mW}$$

$$+10\text{dBm} = 10\text{mW}$$

In the linear amplitude format, the value associated with the vertical axis of the grid is expressed in W.

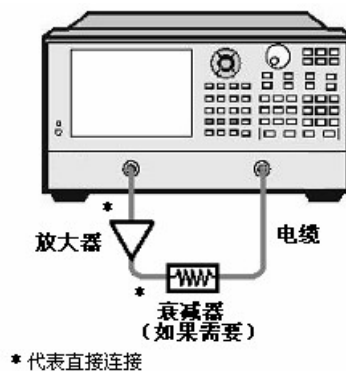
8.6.2 Why do We Measure Absolute Output Power

The absolute output power must be measured when the output of the amplifier must be expressed by absolute power rather than relative power. For example, in gain compression measurement, it is common to measure the absolute output power of the amplifier at 1 dB compression. To improve measurement accuracy, the following factors must be considered:

- 1) If necessary, the output power of the amplifier shall be fully attenuated. Too much output power may lead to the following results:
 - a) The input compression level of the analyzer receiver is exceeded, resulting in inaccurate measurement results;
 - b) The analyzer receiver is burned down.
- 2) Attenuators or couplers can be used to attenuate the output power of the amplifier.
- 3) The amplifier has different response values at different temperatures. The test can be carried out only when the amplifier is at the required operating temperature.

8.6.3 Measurement Steps for Absolute Output Power

- 1) Reset the analyzer.
- 2) Select non-ratio power measurement (source at port 1, receiver B as input).
- 3) Set the source power of the analyzer to 0 dBm.
- 4) Connect the amplifier and provide DC bias as shown in Figure 8.17. If necessary, use an external attenuator to fully attenuate the output power of the amplifier so as not to cause



compression or burnout of the receiver at port 2 of the analyzer.

Figure 8.17 Absolute output power measurement connection

- 5) Select the setting of the analyzer according to the amplifier under test.
- 6) Remove the amplifier, connect the attenuator and cable between the two ports, use the trace operation function to save the data trace to the memory, and use the data/storage function for normalization. If attenuators and cables are required for amplifier measurement, they must be included in normalization measurement.
- 7) Reconnect the amplifier.
- 8) To scale the measurement display for best viewing, use the cursor to read the absolute output power at the desired frequency.
- 9) Print or save data.

8.7 AM-PM Conversion

The AM-PM conversion of the amplifier is to characterize the magnitude of the phase deviation caused by the system amplitude margin.

8.7.1 What is AM-PM Conversion

AM-PM conversion is to characterize the unwanted phase deviation caused by the amplitude margin (AM) of the system. In the communication system, the unwanted phase deviation may be caused by the following reasons:

- 1) Unintentional amplitude margin (AM)
 - a) Power supply fluctuation
 - b) Thermal drift
 - c) Multipath fading
- 2) Intentional signal amplitude modulation
 - a) QAM (quadrature amplitude modulation)
 - b) Sudden pulse modulation

AM-PM conversion is usually defined as the change in output phase when the sweep power applied to the input port of the amplifier increases by 1 dB (at the 1 dB gain compression point), expressed in $^{\circ}/\text{dB}$. The phase response of an ideal amplifier is independent of the power level of the input signal.

8.7.2 Reason for Measuring AM-PM Conversion

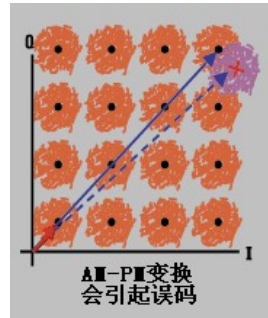
AM-PM conversion is a very critical parameter in the following phase (angle) modulation systems:

- 1) FM
- 2) QPSK

8.7 AM-PM Conversion

3) 16QAM

Undesired phase deviation will lead to the degradation of analog signal performance or the increase of bit error rate (BER) in digital communication systems. While it is easy to measure the bit error rate (BER) of a digital system, this measurement does not help to understand the root cause of the bit error. AM-PM conversion is one of the main factors causing bit error, so it is very



important to measure this parameter quantitatively in communication system. The I/Q Diagram in Figure 8.18 illustrates the causes why AM-PM conversion introduces bit error.

Figure 8.18 AM-PM conversion causes bit error

- 1) The desired state change is from a small solid line vector to a large solid line vector.
- 2) Due to AM-PM conversion, the large solid line vector may actually end up as shown by the dotted line, which is caused by the phase offset due to the change of input power level.
- 3) For the 64QAM signal as shown in Figure 8.18 (only one quadrant is drawn), the noise circles around each state overlap, which means that there will be a bit error in statistics.

8.7.3 Factors Related to Measurement Accuracy

The network analyzer is used to measure AM-PM conversion. The frequency of AM signal is approximately the reciprocal of sweep time. Even if the fastest power sweep is set, the modulation frequency is quite low (usually below 10 Hz). This may cause a slight temperature change of the DUT during sweep, especially if the amplifier has a small thermal mass (a typical example is an unpacked device). If the nonlinear characteristics of the amplifier are extremely sensitive to thermal changes, the measurement results obtained by this method may have errors. In order to improve the measurement accuracy, the following factors should be considered during measurement:

- 1) The response of the amplifier may be quite different at different temperatures, and the test shall be carried out at the required operating temperature of the amplifier.
- 2) If necessary, the output power of the amplifier shall be fully attenuated. Too much output power may cause the following consequences:
 - a) The input compression level of the analyzer receiver is exceeded, resulting in inaccurate measurement results;
 - b) The analyzer receiver is burned down.

8.7 AM-PM Conversion

- 3) Attenuators or couplers can be used to attenuate the output power of the amplifier.
- 4) The influence of attenuator and coupler frequency response must be considered during calibration, because they are part of the test system, and they must be included in the calibration. Sophisticated error correction techniques can reduce or remove the influence of these accessories.
- 5) Frequency response is the dominant error in the measurement composition of AM-PM conversion, and this error can be significantly reduced by through response calibration.

8.7.4 Measurement Steps for AM-PM Conversion

- 1) Reset the analyzer.
- 2) Select S21 measurement in power sweep mode.
- 3) Set the start and end power levels of the analyzer power sweep. The start power level should be in the linear region of the amplifier response (usually at least 10 dB lower than the input 1 dB compression point), and the end power level should be in the compression region of the amplifier response.
- 4) If necessary, use an external attenuator or coupler to fully attenuate the output power of the amplifier to avoid compression or burnout of the receiver at port 2 of the analyzer.
- 5) Connect the amplifier and provide DC bias as shown in Figure 8.19.

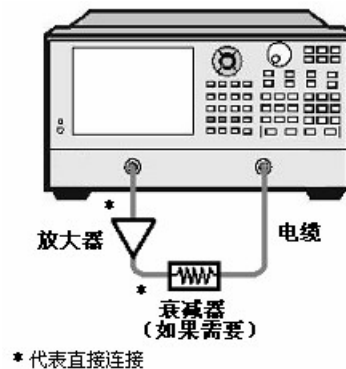


Figure 8.19 AM-PM transformation measurement connection

- 6) Select the analyzer setting according to the amplifier under test, so that the gain compression measurement in power sweep can be carried out at the selected frequency. For more details, please refer to "[Annex 5 Grain Compression Measurement of Amplifier](#)".
- 7) Remove the amplifier for calibration. If attenuators, couplers and cables are used to measure the amplifier, they must be included in the calibration.
- 8) Reconnect the amplifier, turn on the cursor R, place it at the 1 dB gain compression point of the amplifier, then turn on the second cursor, activate Δ cursor mode, and set the stimulus value of the second cursor to -1 dBm.
- 9) Change the S21 measurement from logarithmic amplitude format to phase format (no recalibration required).

8.8 Linear Phase Deviation

- 10) Read the phase difference between the cursors, which is the AM-PM conversion coefficient at the 1 dB gain compression point.
- 11) Print or store data.

8.8 Linear Phase Deviation

Linear phase deviation is to measure the phase distortion of the device. During measurement, the electrical delay function of the analyzer is used to remove the linear part of the phase offset, so as to display the nonlinear part of the phase deviation with high resolution: linear phase deviation.

8.8.1 What Is Linear Phase Offset

During signal transmission, as the frequency of incident signal increases, the wavelength of the signal becomes shorter and shorter, and obvious phase offset will occur. When the phase response of the device is proportional to the frequency, a linear phase offset occurs, and the phase versus frequency trace displayed on the analyzer is a sloping line with a slope proportional to the electrical length of the device. To transmit the signal without distortion, there must be a linear phase offset.

8.8.2 What Is Linear Phase Offset

In practice, many devices delay some frequencies more than others, resulting in nonlinear phase offset and distortion of signals including multiple frequency components. Measuring linear phase deviation is a method to determine the magnitude of such nonlinear phase offset.

Since only the linear phase deviation causes the phase distortion, it is desirable to remove the linear part of the phase response from the measurement. This can be achieved by using the electrical delay function of the analyzer. The electrical length of the DUT is removed through mathematical processing, and the remaining is the linear phase deviation or phase distortion.

8.8.3 Reason for Measuring Linear Phase Deviation

Linear phase deviation measurement has the following advantages:

- a) The measurement results are phase data rather than group delay data in seconds. For devices that transmit modulated signals, phase data may be more useful.
- b) A measurement method that is less noisy than the group delay is provided.

8.8.4 Use Electrical Delay Function

The electrical delay characteristic of the analyzer has the following functions:

- 1) Analog variable-length lossless transmission line can be easily added to or removed from the signal path.
- 2) Compensate for changes in the electrical length of the DUT.

8.8 Linear Phase Deviation

- 3) Make the phase measurement trace displayed by the analyzer flat and observe the trace with high resolution, so as to find the nonlinear details of the phase.
- 4) A convenient method for observing the linear phase deviation of the DUT is provided.

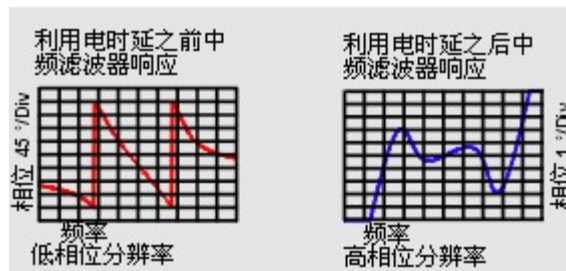


Figure 8.20 Observing linear phase deviation with electrical delay function

For details of electrical delay, see "6.5.1 Electrical delay".

8.8.5 Factors Related to Measurement Accuracy

The frequency response of the test component is the error that plays a major role in the linear phase deviation measurement. This error can be significantly reduced by through response calibration. In order to obtain higher accuracy, full dual-port calibration should be carried out.

8.8.6 Measurement Steps For Linear Phase Deviation

- 1) Reset the analyzer. If the DUT is an amplifier, it may be necessary to adjust the source power of the analyzer:
 - a) Set the source power of the analyzer so that the amplifier operates in the linear region (usually the power is at least 10 dB lower than the input 1 dB compression point).
 - b) If necessary, use an external attenuator or coupler to fully attenuate the output power of the amplifier to avoid compression or burnout of the receiver at port 2 of the analyzer.
- 2) Select S21 measurement.
- 3) Connect the DUT, as shown in Figure 8.21.



Figure 8.21 Linear phase deviation measurement connection

- 4) Set the analyzer according to the DUT, including setting the format as phase.

8.9 Reserve Isolation

- 5) Remove DUT and perform calibration.
- 6) Reconnect the DUT and scale the measurement display for optimal viewing.
- 7) Change the electrical delay to make the phase trace flat.
- 8) Read the linear phase deviation with the cursor.
- 9) Print or save data.

8.9 Reserve Isolation

The reverse isolation measures the reverse transmission response of the amplifier from the output to the input.

8.9.1 What is Reverse Isolation

Reverse isolation measures the degree of isolation from the device output to the input. The measurement of reverse isolation is similar to the measurement of forward gain, but has the following differences:

- 1) The stimulus signal is applied to the output port of the amplifier.
- 2) The response signal is measured at the input port of the amplifier.
- 3) Its equivalent S-parameter is S_{12} .

8.9.2 Reasons for Measuring Reverse Isolation

An ideal amplifier should have infinite reverse isolation and have no signal coming back from the output to the input. However, in practice the signal may pass through the amplifier in the opposite direction. This undesired reverse transmission may cause the reflected signal at the output port to interfere with the desired signal for forward transmission, so it is important to quantify the reverse isolation.

8.9.3 Factors Related to Measurement Accuracy

Since the amplifier usually exhibits high loss in reverse direction, it is generally unnecessary to use an attenuator or coupler to protect the receiver at port 1 during reverse transmission measurement. Removing the attenuator will increase the dynamic range and thus improve the measurement accuracy. A larger dynamic range and higher accuracy can be provided by increasing the source power.

**Warning****Do not overload power during forward sweep or full dual-port calibration**

With the removal of attenuator and the increase of RF power, the receiver at port 2 of the analyzer may be burned during forward sweep. Therefore, do not perform forward sweep or full dual-port calibration unless the forward power is set low enough to prevent the receiver at port 2 of the analyzer from being burned.

If the isolation of the amplifier under test is very large, the level of the reverse transmitted signal may be close to the noise base or crosstalk level of the receiver. To reduce the noise base, averaging can be used, the average times can be increased, or the IF bandwidth can be reduced to improve the dynamic range and accuracy of the measurement at the expense of the measurement speed.

- 1) When the crosstalk level affects the measurement accuracy, the crosstalk error can be reduced by through response and isolation calibration. The same averaging factor and IF bandwidth must be used during calibration and measurement.
- 2) In the reverse isolation measurement, the frequency response of the test component is the main source of error, and this error can be removed by conducting the through response or through response plus isolation calibration.
- 3) The response of the amplifier may be quite different at different temperatures, and the test shall be carried out with the amplifier at the required operating temperature.

8.9.4 Measurement Steps of Reserve Isolation

- 1) Reset the analyzer.
- 2) Select S12 measurement.
- 3) Connect the amplifier and provide DC bias as shown in Figure 8.22.

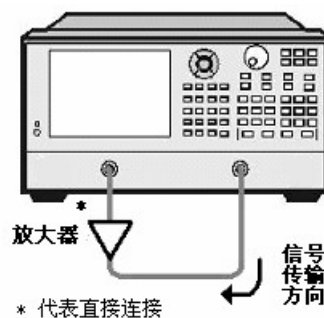


Figure 8.22 Reserve isolation measurement connection

- 4) Set the analyzer according to the amplifier under test.
- 5) Remove the amplifier and perform the through response calibration or through response plus isolation calibration.

8.10 Small Signal Gain and Flatness

- 6) Reconnect the amplifier and scale the measurement display for optimal viewing.
- 7) Use the cursor to read the reverse isolation of the frequency of interest.
- 8) Print or save data.

8.10 Small Signal Gain and Flatness

The small signal gain is the gain of the amplifier in the linear operating region, which is typically measured at a constant input power over the sweep range. Gain flatness is to measure the change of gain within the specified frequency range.

8.10.1 What is Gain

The gain of the amplifier is defined as the power difference (in dBm) between the output signal and the input signal of the amplifier. It is assumed that the input and output impedance of the amplifier are the same, which is the characteristic impedance of the system.

- 1) The gain is referred to as S21 in S-parameter terminology.
- 2) The gain is expressed in dB, that is, the logarithmic ratio of output power to input power.
- 3) When both input and output levels are expressed in dBm (relative to 1 mW power), the gain can be calculated by subtracting the input level from the output level.
- 4) Amplifier gain usually refers to the minimum gain within the operating frequency range. For some amplifiers, both the minimum and maximum gains are provided to ensure that the subsequent stages of the system will not be under-excited or over-excited.

8.10.2 What is Flatness

Flatness refers to the magnitude of change in amplifier gain within the specified frequency range, which may cause the distortion of the signal passing through the amplifier.

8.10.3 Reasons for Measuring Small Signal Gain and Flatness

The deviation of the gain within the bandwidth of interest makes frequency components of the signal not amplified equally, which will cause the transmission signal to be distorted. The small signal gain represents the gain of the amplifier at a specific frequency in the 50 Ω system, and the flatness represents the deviation of the amplifier gain within the specified frequency range in the 50 Ω system.

8.10.4 Factors Related to Measurement Accuracy

- 1) The response of the amplifier may be quite different at different temperatures, and the test shall be carried out with the amplifier at the required operating temperature.
- 2) If necessary, the output power of the amplifier shall be fully attenuated. Too much output power may lead to the following results:

8.10 Small Signal Gain and Flatness

- a) The input compression level of the analyzer receiver is exceeded, resulting in inaccurate measurement results;
 - b) The analyzer receiver is burned down.
- 3) The output power of attenuation amplifier can be completed by an attenuator or coupler. Since the attenuator and coupler are part of the test components, the influence of their mismatch and frequency response on the measurement accuracy must be considered during calibration. Correct error correction can reduce the influence of these accessories.
 - 4) The frequency response of the test component is the most important error in small signal gain and flatness measurement. This error can be significantly reduced by performing through response calibration, and higher measurement accuracy can be obtained by full dual-port calibration.
 - 5) The dynamic range and accuracy can be improved by reducing the IF bandwidth or utilizing the measurement average, but at the expense of the measurement speed.

8.10.5 Measurement Steps for Small Signal Gain and Flatness

- 1) Reset the analyzer.
- 2) Select S21 measurement parameter.
- 3) Set the source power of the analyzer so that the amplifier operates in the linear region. Usually the power is at least 10 dB lower than the input 1 dB compression point.
- 4) If necessary, use an external attenuator or coupler to fully attenuate the output power of the amplifier to avoid compression or burnout of the receiver at port 2 of the analyzer.
- 5) As shown in Figure 8.23, connect the amplifier and provide the amplifier with DC bias.

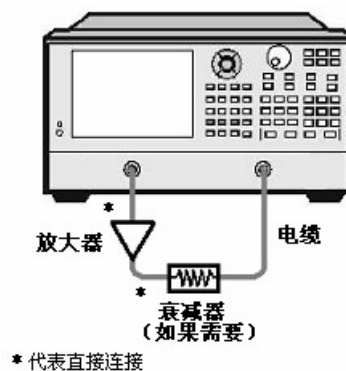


Figure 8.23 Small signal gain and flatness measurement connection

- 6) Set the analyzer according to the amplifier under test.
- 7) Remove the amplifier and perform a calibration which must include the attenuator, coupler and cable used in the measurement.
- 8) Reconnect the amplifier.
- 9) Scale the measurement display for optimal viewing, and use the cursor to read the small

8.10 Small Signal Gain and Flatness

signal gain at the desired frequency point.

- 10) Use cursors to observe the peak-to-peak ripple over the entire frequency range to measure gain flatness.
- 11) Print or save data.

9 Remote Control

This section introduces briefly the programmed control foundation, programmed control interface and configuration method and basic VISA interface programming method of the 3657 series vector network analyzer, as well as the concept and classification of the I/O instrument driver library, so as to facilitate users to start to achieve remote control. Specific contents include:

- [Remote Control Basics.....323](#)
- [Instrument Programmed Control Port and Configuration.....340](#)
- [Basic VISA Interface Programming Method.....343](#)
- [I/O Library.....351](#)

9.1 Remote Control Basics

9.1.1 Programmed Control Interface

This analyzer supports LAN, and the type of port supported by the specific model of instrument is determined by the function of the analyzer.

The remote control interface and related VISA addressing string are described in the table below:

Table 9.1 Remote control interface type and VISA addressing string

Remote control interface	VISA addressing string	Description
LAN (Local Area Network)	VXI-11 protocol: TC PIP::Addressograph[::LAN_device_name][::INSTR] Raw socket protocol: TC PIP::Addressograph::port::SOCKET	The controller realizes remote control by connecting the instrument with the network port on the rear panel of the instrument. For details of the protocol, please refer to: 9.2.1 LAN Interface

9.1 Remote Control Basics

9.1.1.1 LAN Interface

The vector network analyzer can be controlled remotely by computers in 10Base-T and 100Base-T. Various instruments are combined into a system in LAN and controlled uniformly by computers in it. In order to realize remote control in LAN, the vector network analyzer should be equipped with port connector, network card and relevant network protocol in advance, and provided with relevant network services. Meanwhile, the master computer in the network should also be equipped with instrument control software and VISA library in advance. The three working modes of the network card are:

- 10Mbit/s Ethernet IEEE802.3;
- 100Mbit/s Ethernet IEEE802.3u;
- 1Gbit/s Ethernet IEEE802.3ab.

The master computer and the vector network analyzer should be connected to the common TCP/IP protocol network through the network port. The cable between the computer and the vector network analyzer is a commercial RJ45 cable (Category 5 cable with or without shielding). During data transmission, the transmission speed of LAN is faster when data packet transmission is applied. Generally, the cable length between the computer and the vector network analyzer should not exceed 100 meters (100Base-T and 10Base-T). For more information about LAN communications, please refer to <http://www.ieee.org>. 错误!超链接引用无效。错误!超链接引用无效。

1) IP address

When remotely controlling vector network analyzers over a LAN, the physical connection of the network should be ensured. In the vector network analyzer, click the [Start] -> [Control Panel] -> [Network and Internet] -> [Network and Sharing Center] -> [Change Adapter Settings] -> [Local Connection], then right-click and select [Properties], and set the "Local IP" address to the subnet where the master computer is located. For example, if the IP address of the master computer is 192.168.12.0, the IP of the vector network analyzer shall be set to 192.168.12.XXX, whereas XXX is the figure between 1 ~ 255.

When establishing a network connection, only the IP address is required. The VISA addressing string is as follows:

TCPIP::host address[:LAN device name][:INSTR] or

TCPIP: : host address: port: : SOCKET

Where:

- TCPIP represents the network protocol used;
- host address represents the IP address or host name of the instrument, and is used for identifying and controlling the controlled instrument;
- LAN device name defines the handle number of the protocol and subset (optional);
 - VXI-11 protocol is selected for device 0;
 - More recent high speed LAN instrument protocol is selected for high speed LAN instrument 0;
- INSTR represents the instrument resource type (optional);

- port identifies the socket port number;
- SOCKET represents the raw network socket resource class.

Example:

- The IP address of the instrument is 192.1.2.3, and the effective resource string of the VXI-11 protocol is:
TCPIP::192.1.2.3::INSTR
- To establish a raw socket connection, use:
TCPIP::192.1.2.3::5025::SOCKET

Tips

Method of recognizing multiple instruments in the programmed control system

If multiple instruments are connected in the network, the individual IP address and related resource string are used to distinguish. The host computer applies its own VISA resource string for instrument identification.

2) VXI-11 protocol

The VXI-11 standard is based on the ONC RPC (Open Network Computing Remote Procedure Call) protocol, which is the network/transport layer of the TCP/IP protocol. The TCP/IP network protocol and associated network services are pre-configured for communication. Such connection-oriented communication, which follows the sequential exchange and can identify the interruption of the connection, ensures no loss of information.

3) Socket communication

The TCP/IP protocol connects the network analyzers to the network via LAN sockets. As a basic method used in computer network programming, the socket allows applications using different hardware and operating systems to communicate over a network. This method allows for two-way communication between the vector network analyzers and the computers via ports.

As a software class programmed specially, the socket defines the IP address, device port number and other necessary information for network communication, and integrates some basic operations in network programming. Sockets can be used after installing packaged libraries in the operating system. Two commonly used socket libraries are the Berkeley socket library for UNIX the Winsock library for Windows.

Sockets in vector network analyzers are compatible with Berkeley socket and Winsock through application programming interfaces (APIs). In addition, it is compatible with the API of other standard sockets. When SCPI are used to control the network analyzer, the socket program established in the program issues the command. The socket port numbers of the vector port analyzers must be set before using LAN sockets. The socket port number of the vector network analyzer is 5025 by default, and users can change it as needed.

9.1.2 Message

The messages transmitted on the data cable are divided into the following two categories:

1) Interface message

During communication between the instrument and the host computer, the attention cable should be pulled down first, and then the interface message will be transmitted to the instrument through the data cable.

2) Instrument message

The instrument message can be divided into two types as per the transmission direction, namely, command and instrument response. Unless otherwise stated, all programmed control interfaces apply instrument message in the same way.

a) Commands:

Commands (programming messages) are messages sent by the host computer to the instrument for remote control of instrument functions and query of state information. Commands are divided into the following two categories:

- Based on the impact on the instrument:
 - setting commands: change the set state of the instrument, such as reset or setting frequency.
 - query commands: query and return data, for example: identify the instrument or query the parameter value. Query commands end with the suffix question mark.
- Based on the definition in the standard:
 - common commands: with functions and syntax to be defined by IEEE488.2, they are applicable to all types of instruments (if realized)
 - It is used for management of standard state register, resetting and self-detection, etc.
 - instrument control commands: instrument characteristic commands, used to realize instrument functions, such as setting frequency.

The syntax also follows the specifications of SCPI.

b) Instrument responses:

Instrument responses (response message and service request) are the query result information sent by the instrument to the computer. Such information includes measurement results, instrument status, etc.

9.1.3 SCPI

9.1.3.1 Introduction to SCPI Command

SCPI (Standard Commands for Programmable Instruments) are a command set for all instruments established based on Standard IEEE488.2. The main purpose is to make the instrument and equipment of same function have the same remote control command to achieve

the universality of remote control commands.

SCPI consist of a command header and one or more parameters. The command header is separated from the parameters by spaces and contains one or more key fields. A command with direct suffix question mark is a query command. Commands are divided into common commands and instrument commands that have different syntactic structures. SCPI have the following characteristics:

- 1) Programmed control commands are oriented to test function rather than describing instrument operation;
- 2) Programmed control commands reduce the repetition of similar test function realization process, and ensure the compatibility of programming.
- 3) Program messages are defined in layers that are hardware independent of the communication physical layer.
- 4) Programmed control commands are independent of programming methods and languages. The test program of SCPI is easy to transplant.
- 5) Programmed control commands are scalable and can adapt to different scale of measurement control.
- 6) The extensibility of SCPI makes it a "living" standard.

If you are interested in learning more about SCPI, please refer to:

IEEE Standard 488.1-1987, IEEE Standard Digital Interface for Programmable Instrumentation. New York, NY, 1998.

IEEE Standard 488.2-1987, IEEE Standard Codes, Formats, Protocols and Comment Commands for Use with ANSI/IEEE Std488.1-1987. New York, NY, 1998
Standard Commands for Programmable Instruments(SCPI) VERSION 1999.0.

For remote control command set, classification and description of 3657 series vector network analyzer, please refer to:

- 1) "3 Programmed Control Commands" of the Programmed Control Manual;
- 2) "Annex A Zoom table of SCPI commands classified as per subsystems" of the Programmed Control Manual;

9.1.3.2 Description of SCPI

1) General terms

The following terms apply to this section. To better understand the chapters, you shall understand the exact definitions of the terms.

Controller

A controller is any computer used to communicate with the SCPI device. A controller may be a PC, minicomputer, or a plug-in card on a cage. Some AI devices can also be used as controllers.

9.1 Remote Control Basics

Device

A device is any device that supports SCPI. Most of the devices are electronic measurement or stimulus devices.

Programmed control message

A programmed control message is the combination of one or more SCPI commands that have been correctly formatted. Programmed control messages tell the devices how to measure and output the signals.

Response message

A response message is a set of data of specified SCPI formats. Response messages always come from the devices to controllers or listening devices. Response messages tell the controllers about the internal state or measured values of the devices.

Command

A command is an instruction that satisfies the SCPI standard. The combination of commands controlling the devices forms a message. In general, a command includes keywords, parameters, and punctuation.

Event command

Event-type programmed control commands cannot be queried. An event command generally has no corresponding front panel key setting, and its function is to trigger an event at a specific time.

Query

A query is a special type of command. When a control device is queried, a response message appropriate to the controller syntax requirements is returned. A query statement always ends with a question mark.

2) Command type

There are two types of SCPI: common commands and instrument commands. Figure 9.1 shows the difference between the two commands. Common commands, defined by IEEE 488.2, are used to manage status registers and for synchronization and data storage. Because common commands all start with an asterisk, they are easy to be recognized. For example, *IDN? *OPC, *RST are all common commands. Common commands are not part of any instrument commands, and the instrument interprets them in the same way regardless of the current path setting of the commands.

Instrument commands are easy to be recognized because they contain a colon (:). A colon is used in the beginning of an expression or between two keywords, for example: SENS:FREQuency:CW. According to the internal function module of the network analyzer, instrument commands are divided into sets of corresponding subsystem commands. For example,

9.1 Remote Control Basics

the power subsystem (:POWer) contains power-related commands, while the status subsystem (:STATus) contains commands for the status control register.

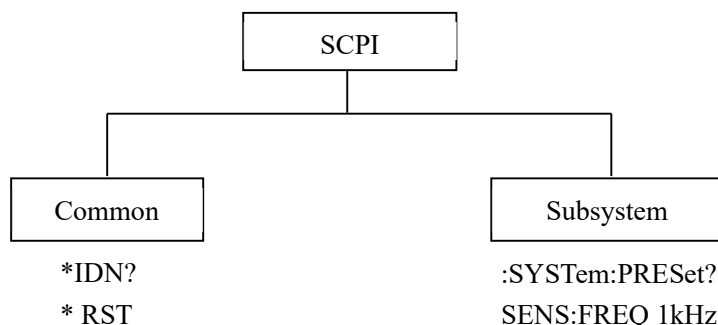


Figure 9.1 Types of SCPI

3) Instrument Command Syntax

A typical command consists of a keyword prefixed with a colon. The keyword is followed by parameters. The following is an example of a syntax declaration:

[[:SOURce]:POWer[:LEVel] MAXimum|MINimum

In the example above, the [:LEVel] in the command follow : POWer closely without any space. MINimum|MAXimum immediately following [:LEVel] is the parameter. There is a space between the command and the parameter. Other parts of the syntax expression are described in Table 9.2 and Table 9.3.

Table 9.2 Special characters in command syntax

Symbol	Meaning	Example
	The vertical bar between the keyword and the parameter represents multiple options.	TRIGger[:SEQuence]:SCOPE ALL CURRent Where ALL and CURRent are options
[]	Square brackets indicate that the keyword or parameter contained is optional when constructing the command. The command will also be executed even if these implied keywords or parameters are ignored.	TRIGger[:SEQuence]:SCOPE ALL CURRent Where SEQuence is optional.
< >	The contents in the angle brackets indicate that the command is not used literally. They indicate the contents that are required.	TRIGger[:SEQuence]:SCOPE <char> In this command, <char> must be replaced with the actual

9.1 Remote Control Basics

		trigger scope string. For example: :TRIG:SCOP ALL
{ }	The part in braces indicates that the parameter is optional.	[[:SOURce]:LIST:POWer <val>{,<val>} For example: LIST:POWer 5

Table 9.3 Command syntax

Characters, keywords and syntax	Example
Uppercase character represents the minimum set of characters required to implement command .	SENSe<num>:FREQuency:CENTer <num> FREQ is the short format part of the command.
The lower case part of the command is optional; such flexible format is loaded called "flexible listening". Please refer to "Command Parameters and Responses" for more information.	:FREQuency :FREQ,:FREQuency or or :FREQUENCY Either of them is correct.
When a colon is between two command mnemonics, it will move the current path down a level in the command tree. Please refer to the command path of "Command Tree" for more information.	TRIGger[:SEQuence]:LEVel <char> TRIGger is the top-level keyword for the command.
If the command contains more than one parameter, adjacent parameters are separated by commas. The parameter is not part of the command path, so it does not affect the path layer.	MMEMory:COpy <file1>,<file2>
The semicolon is used to separate 2 adjacent commands, without affecting current command path.	SENS:FREQ:STAR 2.5GHZ; :STOP 5GHZ
Blank characters, such as <space> or <tab>, are usually ignored as long as they do not appear between or within keywords. However, you must separate the commands and parameters with blank characters,	:FREQ uency or :POWer :LEVel6.2 is not allowed. :LEVel and 6.2 must be separated by a space, namely, :POWer:LEVel 6.2.

which does not affect the current path.	
---	--

4) Command Tree

Most remote control programs apply instrument commands. When parsing such commands, SCPI apply a file system-like structure called command tree, as shown in Figure 9.2:

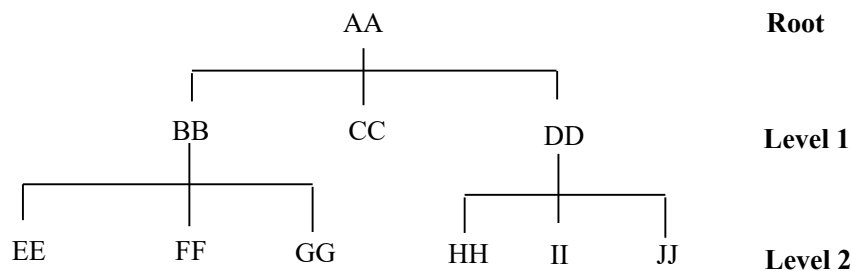


Figure 9.2 Simplified command tree

The top command is the root command, or "root" for short. When a command is parsed, follow a specific path to the next level of command according to the tree structure. For example, in :POWER:ALC:SOURce?, :POWER stands for AA, :ALC stands for BB, :SOURce stands for GG, and the entire command path is (:AA:BB:GG).

A software module in instrument software - **command interpreter**, is responsible for parsing each received SCPI. The command interpreter breaks commands into individual command elements by using a series of rules that distinguish the path of the command tree. After parsing the current command, keep the current command path unchanged. The advantage of this is to parse subsequent commands more quickly and efficiently since that the same command keyword may appear in different paths. After booting or *RST (resetting) the instrument, current command path is reset to root.

5) Command parameters and response

SCPI define different data formats in the use of programmed control and response messages to comply with the principles of "**flexible listening**" and "**precise speaking**". For more information, please refer to IEEE488.2. "**Flexible listening**" means that the formats of the commands and parameters are flexible.

For example, the vector network analyzer sets power level command:

:SOURce<cnum>:POWer<port>[:LEVel][:IMMediate][:AMPLitude] <num>, [src]

The following command formats are all used to set the power level:

:SOURce:POWer:LEVel:IMMediate:AMPLitude -5

:SOURce:POWer -5

:SOUR:POW:LEV:IMM:AMP -5

:SOUR:POW -5.

9.1 Remote Control Basics

Each parameter type has one or more corresponding response data types. During query, a data type will be returned for a numerical parameter, and the response data is precise and strict, known as **"precise speaking."**

For example, when the setting is -5 dBm at the time of querying the power level (:SOURce:POWer?), the returned response data is always -5.000000000000e+000, regardless of whether the previously setting command sent is :SOUR:POW:LEV:IMM:AMP -5 or :SOUR:POW -5.

Table 9.4 Parameter and response types of SCPI

Parameter type	Response data type
Numerical	Real number or integer
Extended numerical	Integer
Discrete	Discrete
Boolean	Digital boolean
String	String
Blocks	Finite-length blocks
	Infinite-length blocks
Non-decimal numeric types	Hexadecimal
	Octal
	Binary

Numerical parameters

Numeric parameters can be used in both instrument-specific commands and common commands. A numeric parameter receives all the usual decimal counting methods, including signs, decimals, and scientific notation. If a device only accepts a specified numeric type, such as an integer, it will automatically round up the received numeric parameters.

Examples of numeric parameters:

- 0 No decimal point
- 100 Optional decimal point
- 1.23 Signed bit
- 4.56e<space>3 Index mark e can be followed by a space
- 7.89E-01 Index marker e can be uppercase or lowercase
- +256 Positive lookahead allowed
- 5 Decimal points can be used first

Extended numerical parameters

9.1 Remote Control Basics

Most measurements related to instrument commands use extended numeric parameters to specify physical quantities. Extended numerical parameters receive all numeric parameters and additional special values. All the extended numeric parameters receive MAXimum and MINimum as parameter values. Whether other special values, such as UP and DOWN, will be received is determined by the ability of the instrument to parse. All effective parameters will be listed in the table of SCPI.

Attention: Extended numeric arguments do not apply to common commands or STATus subsystem commands.

Examples of extended numeric parameters:

101 Numeric parameter	
1.2GHz	GHz can be used as an index (e009)
200Mhz	MHz can be used as an index (e006)
-100mV	-100 millivolt
10DEG	10°
MAXimum	Maximum effective setting
MINimum	Minimum effective setting
UP	Increase by a step
DOWN	Reduce by a step

Discrete parameters

When the number of parameter values to be set are finite, they are identified by discrete parameters. Discrete parameters use mnemonics to represent each valid setting. Like remote control command mnemonics, discrete parameter mnemonics have two formats, long and short, and allow for mixture of upper and lower cases.

In the following examples, discrete parameters and commands are used together.

:TRIGger[:SEQuence]:SOURce MANual|IMMediate|EXTernal

MANual	Manual trigger
IMMediate	Trigger immediately
EXTernal	Trigger externally

Boolean parameters

A Boolean parameter represents a true or false binary condition, which can only have four possible values.

Boolean parameter examples:

ON	Logically true
OFF	Logically false
1	Logically true
0	Logically false

String parameters

9.1 Remote Control Basics

String parameters allow ASCII strings to be sent as parameters. Single quotes and double quotes are used as separators.

The following are example of string parameters:

'This is Valid' "This is also Valid" 'SO IS THIS'

Real response data

Most of the test data are of real number type, and their formats can be basic decimal notation or scientific notation, which are supported by most advanced programming languages.

Examples of real number response data:

1.23E+0
-1.0E+2
+1.0E+2
0.5E+0
0.23
-100.0
+100.0
0.5

Integer response data

An integer response data is a decimal expression of an integer value containing signed bit. When querying the status register, most of the response data returned are of integer type.

Examples of integer response data:

0	Sign bit optional
+100	Positive lookahead allowed
-100	Negative lookahead allowed
256	No decimal point

Discrete response data

Discrete response data are basically the same as discrete parameters, only that the return format of discrete response data is only a short form in uppercase.

Examples of discrete response data:

MLIN	Linear amplitude as display format
MLOG	Logarithm magnitude as display format
PHAS	Display format as phase

Digital bool response data

A binary value 1 or 0 is returned as Boolean response data.

String response data

9.1 Remote Control Basics

String response data and string parameters are alike. The main difference is that the separators of string response data are double quotes instead of single quotes. Double quotes can also be embedded in string response data, and there may be no characters between the double quotes. Here are some examples of string response data:

"This is a string"

"one double quote inside brackets: (\"")"

6) Number System of Commands

The value of the command can be entered in binary, decimal, hexadecimal or octal format. When using binary, hexadecimal or octal format, a proper identifier is required before the value. The decimal format (the default format) does not require an identifier. When a value is entered without an identifier in front of it, the device will ensure it to be in decimal format. The following list shows the required representations for each format:

- #B indicates that the number is a binary number.
- #H indicates that the number is a hexadecimal number.
- #Q indicates that the number is an octal number.

The following are various representations of the decimal number 45 in SCPI:

#B101101

#H2D

#Q55

The following example sets the RF output power to 10dBm (or a value of the equivalent value of the currently selected unit, such as DBUV or DBUVEMF) with a hexadecimal value of 000A.

:POW #H000A

When using a non-decimal format, a measurement unit, such as DBM or mV, is not used with the value.

7) Command line structure

A command line may contain multiple SCPI. To indicate the end of the current command line, the following methods may be used:

- Enter;
- Enter and EOI;
- EOI and the last data byte.

Commands on the command line are separated by semicolons, and commands belonging to different subsystems begin with a colon. For example:

MMEM:COPY "Test1","MeasurementXY";HCOP:ITEM ALL

The command line contains two commands: the first one belongs to the MMEM subsystem, and the second belongs to the HCOP subsystem. If adjacent commands belong to the same subsystem with repeated command path, they can be expressed in abbreviation. For example:

9.1 Remote Control Basics

HCOP:ITEM ALL;:HCOP:IMM

The command line contains two commands: both of them belong to the HCOP subsystem, with the same first level. Therefore, the second command can start from the next level of HCOP, and the colon for starting the command can be omitted. It can be abbreviated as follows:

HCOP:ITEM ALL;IMM

9.1.4 Command Sequence and Synchronization

IEEE488.2 defines the difference between overlapping and sequential commands:

- Sequential commands are sequences of commands that are executed continuously. Each command is usually executed faster.
- An overlapping command is one that is not executed automatically before the next command is executed. It usually takes longer to process overlapping commands, and programs are allowed to process other events synchronously during the period.

Even if there are multiple setting commands on a command line, they may not be executed in the order they were received. To ensure that commands are executed in a certain order, each command must be sent on a separate command line.

Example: the command line contains setting and query commands

If multiple commands on a command line contain query commands, the query results are unpredictable. A fixed value is returned for the following command:

:FREQ:STAR 1GHZ;SPAN 100;:FREQ:STAR?

Returned value: 1000000000 (1GHz)

The following command returns an unfixed value:

:FREQ:STAR 1GHz;STAR?;SPAN 1000000

The returned result can be current start frequency value of the instrument before such command is sent, since the host program will not execute the commands one by one until all command messages are received. If the host program executes the command after being received, the return result can also be 1GHz.

Tips

Setting commands are sent separately from query commands

General rules: The setting command and query command shall be sent in different programmed control messages, so as to ensure the returned result of the query command is correct.

9.1.4.1 Prevent Overlapping Execution of Commands

In order to prevent overlapping execution of commands, multithreading or commands *OPC, *OPC? or *WAI may be applied, which are executed only after the hardware settings are completed. During programming, the computer may force a period of time to synchronize certain events. The descriptions are shown below:

➤ **Controller program using multi-threading**

Multi-threading is used to wait for command completion and synchronization between the UI and programmed control, that is, to wait for *OPC? Completion in separate threading without interfering GUI or programmed control threading execution.

- The application of the three commands in synchronous execution is shown in the table below:

Table 9.5 Command syntax

Method	Action	Programming method
*OPC	After the command is executed, set it in the operation completion bit in ESR.	Set to ESE BIT0; Set to SRE BIT5; Send overlapping commands and *OPC; Wait for service request (SRQ) Service request represents that the overlapping command has been executed.
*OPC?	Stop executing the current command 1. The command is returned only when it is in the operation completion bit of the ESR register, indicating that the previous command has been processed.	Terminate processing of the current command before executing other commands, and send the command directly after the current command.
*WAI	Before the execution of *WAI, wait for all commands to be sent before proceeding with unfinished commands.	Terminate the processing of the current command before executing other commands, and send the command directly after the current command.

9.1.5 Status Reporting System

The status report system stores all operation status information and error information of current instrument. Such information is stored in the status register and error list respectively, which can be queried via the remote control interface.

- [State Register Structure.....311](#)
- [Application of State Reporting System312](#)

9.1.5.1 Structure of Status Register

Status registers are described by classification below:

9.1 Remote Control Basics

1) STB, SRE

The status byte (STB) register and its related mask register - service request enable register (SRE), comprise the top register of the status reporting system. STB saves the general working state of the instrument by collecting the information of lower registers.

2) ESR, SCPI status register

STB receives information from the following registers:

- a) The value of the event status register (ESR) and the event status enable (ESE) mask register.
- b) SCPI status registers include: State:OPERation and State:QUEStionable registers (defined by SCPI), which contain the specific operation information of the analyzer. All SCPI state registers have the same internal structure (For details, see 2.1.5.2 “SCPI state register structure” section).

3) IST,PPE

Similar with SRQ, IST (“Individual Status”) marks a separate bit consisting of all statuses of the instrument. The associated parallel poll enable register (PPE) determines the STB data bits for IST marking.

4) Output buffer zone

It stores the messages returned by the instrument to the controller. It does not belong to the status report system, but determines the MAV position value of the STB.

For details on the registers mentioned above, see [“2.1.5 Status report system” of the programmed control manual](#).

Please refer to Figure 9.3 for the hierarchical structure of the state register:

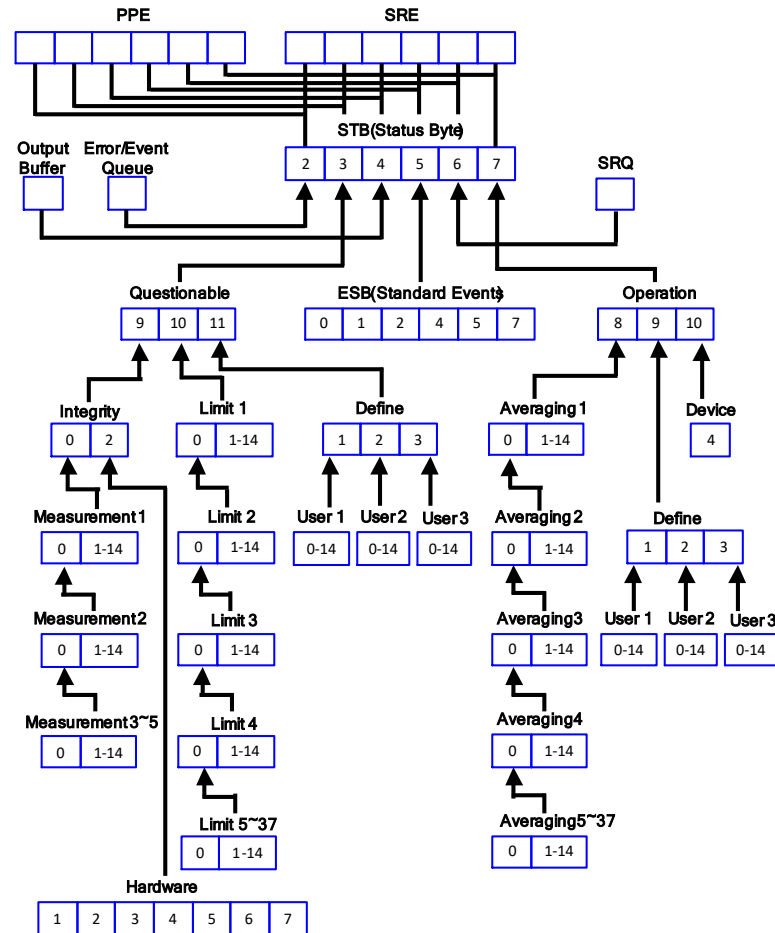


Figure 9.3 Hierarchical structure of the status register

Tips

SRE, ESE

SRE may be used as the enable part of STB. Similarly, ESE may be used as the enable part of ESR.

9.1.5.2 Application of Status Reporting System

The status reporting system is used to monitor the state of one or more instruments in a test system. In order to correctly realize the function of the state reporting system, the controller in the test system must receive and evaluate the information of all instruments. Standard methods used include:

- 1) Service request (SRQ) initiated by the instrument;

9.2 Instrument Programmed Control Port and Configuration

- 2) Serial query of all instruments in the bus system, initiated by the controller in the system, in order to find the initiator of the service request and the reason.
- 3) Parallel query of all instruments;
- 4) Programmed control command to query the status of specific instruments;

For specific operation methods, see “2.1.5.4 Status report system application” of the remote control manual.

9.1.6 Programming Precautions**1) Please initialize the instrument status before changing settings**

During remote instrument setting, initialize the instrument status (for instance, to send “*RST”) first before realizing required status setting.

2) Command sequence

In general, setting commands and query commands should be sent separately.

Otherwise, the returned value of query commands will change with the current order of instrument operation.

3) Fault response

Service requests can only be initiated by the network analyzer. The controller program in the test system should guide the instrument to initiate service request actively when there is an error, and then enter corresponding interrupt service program for processing.

4) Error Queue

Each time the controller program processes a service request, it should query the error queue of the network analyzer instead of the status register for a more precise cause of the error. Especially in the test phase of the controller program, the error queue should be frequently queried to obtain the error command sent by the controller to the network analyzer.

9.2 Instrument Programmed Control Port and Configuration

- [LAN.....340](#)

9.2.1 LAN

SICL-LAN is used by LAN programmed control system to control 3657 series vector network analyzers.

Notice

Application of the connector at the master control port of front-panel USB

Type-A connector on the front panel is the connector at the master control port of USB. In 3657 series vector network analyzers, the port is used to connect the flash disk of USB 1.1 interface to realize the upgrade of resident software of the instrument. It may also be connected to the USB keyboard and mouse to control the vector network analyzer. The port should not be used for programmed control of the instrument.

9.2.1.1 Connection Establishment

Connect the 3657 series vector network analyzer and external controller (computer) to the LAN with network cable, as shown in Figure 9.4:

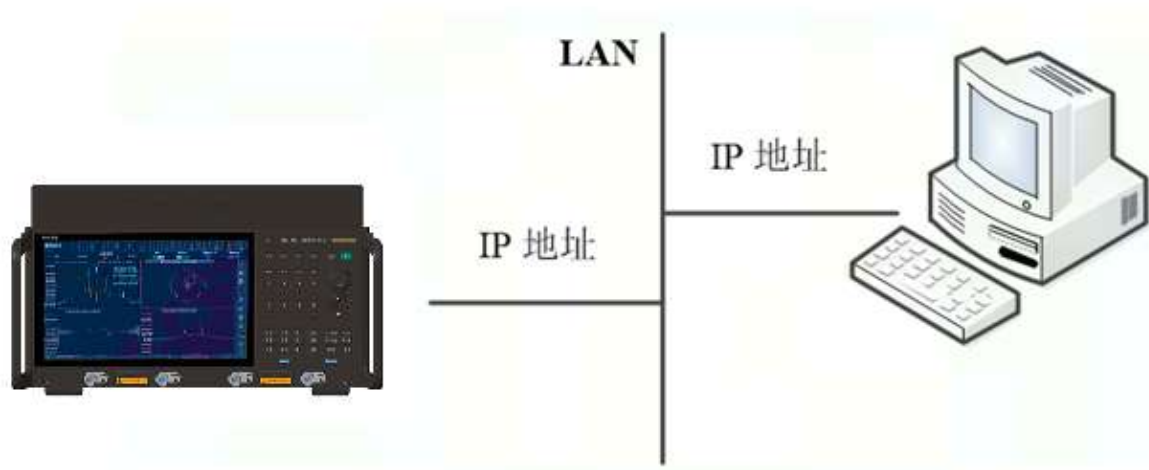


Figure 9.4 LAN interface connection



9.2 Instrument Programmed Control Port and Configuration

Figure 9.5 Instrument rear panel LAN interface

9.2.1.2 Interface Configuration

When remotely controlling vector network analyzers over a LAN, the physical connection of the network should be ensured. Since DHCP, domain name access and WAN connection are not supported, the network remote control setting of the vector network analyzer is relatively simple. Through the menu shown in Figure 9.6, set "IP Address", "Subnet Mask" and "Default Gateway" to the subnet where the master controller resides. The socket port number of the vector network analyzer is 5025 by default, and the value range is 1024~65535. Users can also change it as needed.

Click [System] ->[Remote Control...] to enter the interface shown in Figure 9.6, and then use keyboard, mouse or the number key on the front panel to modify in network terminal input box of the network analyzer.

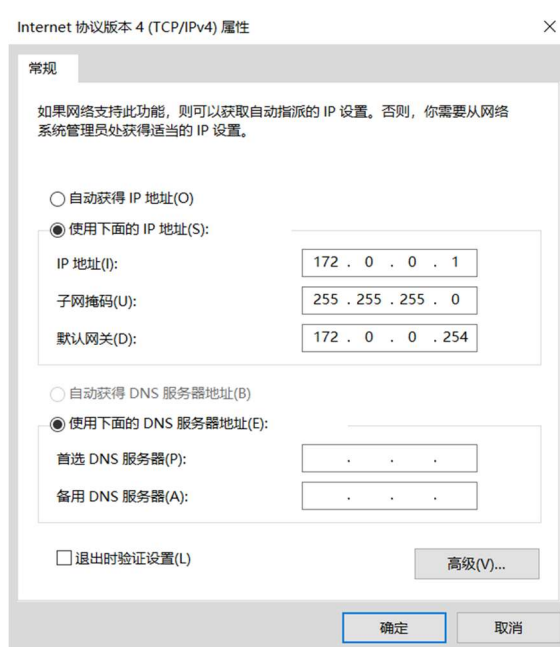




Figure 9.6 LAN interface setting

Notice

Make sure the vector network analyzers are in normal physical connection via 10Base-T LAN or 100Base-T LAN cables.

Since the vector network analyzer only supports the establishment of a single LAN control system and the setting of static IP address instead of DHCP and host access through DNS and domain name server, users are not required to modify the subnet mask that is set to 255.255.255.0 by the instrument.

9.3 Basic VISA Interface Programming

Take the following as an example to describe the basic method for realizing instrument programmed control programming via the VISA library. Take the C++ language as an example.

- [VISA Library.....344](#)
- [Initialize and Set Default State.....344](#)
- [Send Setting Command.....346](#)
- [Read the State of Measuring Instrument.....348](#)
- [Read Frequency Maker.....349](#)
- [Read Trace Data.....349](#)
- [Command Synchronization.....350](#)

9.3.1 VISA Library

VISA is the generic name for the standard I/O function library and its associated specifications. VISA library function is a set of functions that can be easily called. Its core function can control various types of devices without considering the interface type of devices and the use of different I/O interface software. These library functions are used to write the driver program of the instrument and complete the command and data transmission between the computer and the instrument, so as to realize programmed control of the instrument. By initializing the addressing string ("VISA resource string"), a connection to an instrument with a program port (LAN) can be established.

To realize remote control, the VISA library must be installed first. The VISA library is encapsulated with the transmission function at the bottom layer with VXI and LAN interfaces, so as to facilitate users' direct calling. The vector network analyzers support the following programming interfaces: LAN. **These interfaces can be used with VISA library and programming languages for remote control of vector network analyzers.** At present, Agilent I/O Library provided by Agilent is often used as the underlying I/O Library.

Figure 9.13 uses interface as an example to show the relationship between remote control interface, VISA libraries, programming languages and vector network analyzers.

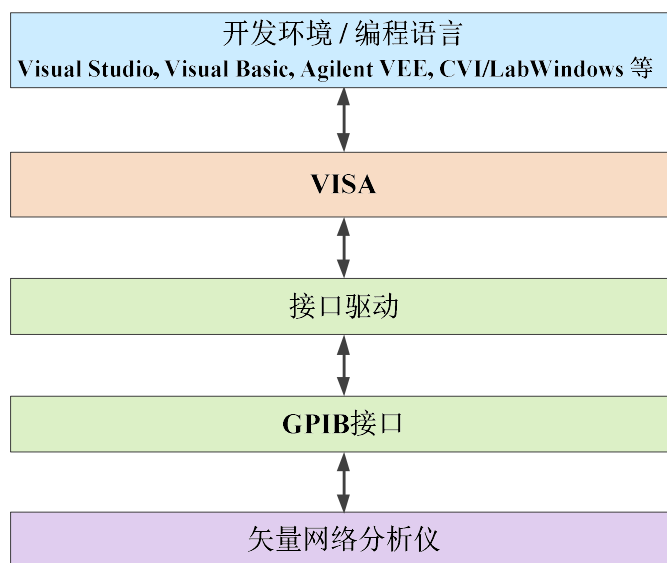


Figure 9.8 Programmed control software/hardware layer

9.3.2 Initialize and Set Default State

To start the program, the VISA resource manager must be initialized, so as to open and establish the communication connection between the VISA library and instrument. The specific steps are as follows:

9.3.2.1 Generate Global Variables

Start by generating global variables that other program modules will call, such as instrument handle variables. The following example programs should contain the following global variables:

```
ViSession analyzer;
ViSession defaultRM;
const char analyzerString [VI_FIND_BUFLen] = "GPIB0::16::INSTR";
const double analyzerTimeout = 10000;
```

Whereas, the constant analyzerString indicates the instrument descriptor, "GPIB0" indicates the master, and "16" indicates the instrument connected with the master. If it is assumed that the network analyzer is connected to the LAN, the IP address is "192.168.1.1", and port number is "5025", then the value of the variable should be:

```
const char analyzerString [VI_FIND_BUFLen] = " TCPIP0::192.168.1.1::5025::SOCKET ";
```

9.3.2.2 Initialize the Controller

The following example shows the way to open and establish the communication connection between the VISA library and instrument (with instrument descriptor specified).

```
//Initialize the master: Open the default resource manager and return the instrument handle
analyzer
```

```
void InitController()
{
    ViState state;
    state = viOpenDefaultRM(&defaultRM);
    status = viOpen(defaultRM, analyzerString, VI_NULL, VI_NULL, &analyzer);
}
```

9.3.2.3 Initialize the Network Analyzer

```
/******
```

The following example shows how to initialize the default state of the instrument and empty the status register. Where "\n" is the terminator, and

a '\n' terminator should be added at the end of the command string for each command, which will not be described below.

```
/******
```

```
void InitDevice()
{
    ViState state;
    long retCnt;
    status = viWrite(analyzer, "*CLS\n", 5, &retCnt); //reset status register
    status = viWrite(analyzer, "*RST\n", 5, &retCnt); //reset instrument
}
```

9.3 Basic VISA Interface Programming

9.3.2.4 Querying Instrument Information

```

/*****

```

The following example lists all currently defined measurements, windows, and traces.

```

*****/

```

```

void QueryMesaurement()

```

```

{

```

```

    ViState state;

```

```

    long retCnt;

```

```

    char rd_Buf_CW[VI_READ_BUFLLEN]; // #define VI_READ_BUFLLEN 20

```

```

    char rd_Buf_LVL[VI_READ_BUFLLEN];

```

```

    // Query all measurements of channel 1

```

```

    status = viWrite(analyzer, ": CALC:PAR:CAT?\n", 15, &retCnt);

```

```

    Sleep(10);

```

```

    status = viRead(analyzer, rd_Buf_CW, 50, &retCnt);

```

```

    // Query all windows

```

```

    status = viWrite(analyzer, ": DISP:CAT?\n", 11, &retCnt);

```

```

    Sleep(10);

```

```

    status = viRead(analyzer, rd_Buf_CW, 20, &retCnt);

```

```

    //Query all traces of Window 1.

```

```

    status = viWrite(analyzer, ": DISP:WIND1:CAT?\n", 18, &retCnt);

```

```

    Sleep(10);

```

```

    status = viRead(analyzer, rd_Buf_CW, 20, &retCnt);

```

```

}

```

9.3.3 Send setting command

9.3.3.1 Set Sweep Parameters

```

/*****

```

The following example shows how to set the sweep parameters of 3657 series vector network analyzer.

```

*****/

```

```

void SweepSettings()

```

```

{

```

```

    ViState state;

```

```

    long retCnt;

```

```

    //Select measurement

```

```

    status = viWrite(analyzer, ":CALC:PAR:SEL 'CH1_WIN1_LINE1' \n", 50,&retCnt);

```

9.3 Basic VISA Interface Programming

```

//Set sweep type as linear sweep
status = viWrite(analyzer, ":SENSe1:SWEep:TYPE LIN\n", 30, &retCnt);

//Set IFBW as 3KHz
status = viWrite(analyzer, ":SENSe1:BANDwidth 3000\n", 30, &retCnt);

//Set the start frequency to 1 GHz and the stop frequency to 10 GHz
status = viWrite(analyzer, ":SENS:FREQ:STAR 1e9\n", 30, &retCnt);
status = viWrite(analyzer, ":SENS:FREQ:STOP 1e10\n", 30, &retCnt);

//Set sweep points as 401
status = viWrite(analyzer, ":SENSe1:SWEep:POINTs 401\n", 30, &retCnt);
//Automatically set sweep time
status = viWrite(analyzer, ":SENSe1:SWEep:TIME:AUTO ON\n", 40, &retCnt);

//Set the output power as -10dBm
status = viWrite(analyzer, ":SOUR:POW -10dBm\n", 22, &retCnt);
}

```

9.3.3.2 Set Display Parameters

```

/*****

```

The following examples mainly include:

Set data format

Display trace, title and frequency annotation

Perform auto scale to trace

Inquire proportional division, reference level and reference position

Open and set the average

Open and set the smoothing

```

*****/

```

```

void DisplaySettings()

```

```

{

```

```

    ViState state;

```

```

    long retCnt;

```

```

    char rd_Buf_Data[VI_READ_BUFLen]; // #define VI_READ_BUFLen 20

```

```

    //Select measurement

```

```

    status = viWrite(analyzer, ":CALC:PAR:SEL 'CH1_WIN1_LINE1' \n", 50,&retCnt);

```

```

    //Set data format as log amplitude

```

```

    status = viWrite(analyzer, ":CALCulate1:FORMat MLOG\n", 30, &retCnt);

```

9.3 Basic VISA Interface Programming

```

//Display trace, title and frequency annotation
status = viWrite(analyzer, ":Display:WINDow1:TRACe1:STATe ON\n", 40, &retCnt);
status = viWrite(analyzer, ":DISPlay:WINDow1:TITLe:STATe ON\n", 40, &retCnt);
status = viWrite(analyzer, ":DISPlay:ANNotation:FREQuency  ON\n", 40, &retCnt);

//Perform auto scale to trace
status = viWrite(analyzer, ":Display:WINDow1:TRACe1:Y:Scale:AUTO\n", 50, &retCnt);

//Inquire proportional division, reference level and reference position
status = viWrite(analyzer, ": DISPlay:WINDow1:TRACe1:Y:SCALe:PDIVision?\n", 60,
&retCnt);
Sleep(10);
status = viRead(analyzer, rd_Buf_Data L, 20, &retCnt);
status = viWrite(analyzer, ":DISPlay:WINDow1:TRACe1:Y:SCALe:RLEVel?\n", 60,
&retCnt);
Sleep(10);
status = viRead(analyzer, rd_Buf_Data, 20, &retCnt);
status = viWrite(analyzer, ": DISPlay:WINDow1:TRACe1:Y:SCALe:RPOSition?\n", 60,
&retCnt);
Sleep(10);
status = viRead(analyzer, rd_Buf_Data, 20, &retCnt);

//Open average and set average factor as 5
status = viWrite(analyzer, ": SENSE1:AVERAge:STATe ON\n", 30, &retCnt);
status = viWrite(analyzer, ": SENSE1:AVERAge:Count 5\n", 30, &retCnt);

//Open smoothing and set smoothing aperture as 20%
status = viWrite(analyzer, ": CALCulate1:SMOothing:STATe ON\n", 40, &retCnt);

status = viWrite(analyzer, " CALCulate1:SMOothing:APERture 20\n", 40, &retCnt);
}

```

9.3.4 Read the Status of Measuring Instrument

The following examples show how to read the set state of the instrument.

void ReadSettings()

```

{
    ViState state;
    long retCnt;
    char rd_Buf_CW[VI_READ_BUFLen]; // #define VI_READ_BUFLen 20

```

```

char rd_Buf_LVL[VI_READ_BUFLLEN];

//Query the point frequency
status = viWrite(analyzer, "FREQ:CENT?", 10, &retCnt);
Sleep(10);
status = viRead(analyzer, rd_Buf_CW, 20, &retCnt);
//Query amplitude
status = viWrite(analyzer, "POW:ALC:LEV?", 12, &retCnt);
Sleep(10);
status = viRead(analyzer, rd_Buf_LVL, 20, &retCnt);
//Print debugging information
sprintf("Cw is %s", rd_Buf_CW);
sprintf("LEVel is %s", rd_Buf_LVL);
}

```

9.3.5 Read Frequency Marker

```

/*****/
The following example shows how to read the frequency marker measurement value.
/*****/

void ReadMarker ()
{
    ViState state;
    long retCnt;
    char rd_Buf_Marker[VI_READ_BUFLLEN]; // #define VI_READ_BUFLLEN 20

    // Turned on the frequency marker 1 and query the peak value of frequency marker
    (frequency and amplitude)
    status = viWrite(analyzer, ":CALC:MARKER ON;MARKER: FUNCTION:EXECute
    MAXimum\n", 25, &retCnt);
    // Read X-axis information
    status = viWrite(analyzer, ":CALC:MARK:X?; ", 15, &retCnt);
    state = viRead(analyzer, rd_Buf_Marker, 30, &retCnt);
    // Read Y-axis coordinate information
    status = viWrite(analyzer, ":CALC:MARK:Y?; ", 15, &retCnt);
    state = viRead(analyzer, rd_Buf_Marker, 30, &retCnt);
}

```

9.3.6 Reading Trace Data

```

/*****/

```

9.3 Basic VISA Interface Programming

The following example illustrates how to read the trace data of the vector network analyzer.

```

/*****
void QueryData ()
{
    ViState state;
    long retCnt = 0;
    int points= 0
    // #define VI_READ_BUFLen 1000000
    char rd_Buf_BigData[VI_READ_DATABUFLen];
    char rd_Buf_Data[VI_READ_BUFLen]; // #define VI_READ_BUFLen 20

    //Select measurement
    status = viWrite(analyzer, ":CALCulate:PARAmeter:SElect ' CH1_WIN1_LINE1'\n", 25,
    &retCnt);
    //Obtain sweep points
    status = viWrite(analyzer, ":SENSe1:SWEep:POIN?\n", 15, &retCnt);
    status = viRead(analyzer, rd_Buf_Data, 30, &retCnt);
    points = atoi(rd_Buf_Data);

    //Sweep once
    status = viWrite(analyzer, ":SENSe1:SWEep:GROups:COUNT 1\n",30, &retCnt);
    status = viWrite(analyzer, ":SENS:SWE:MODE GROups\n",30, &retCnt);

    //Send the reading data command, where the data format is raw data. The data is divided
    into real part and imaginary part, and the returned character//string format is: "The first
    point is real part, the first point is imaginary part, the second point is real part, the second
    point is imaginary part..."
    status = viWrite(analyzer, ": CALCulate:DATA? SDATA\n",30, &retCnt);
    status = viRead(analyzer, rd_Buf_BigData, VI_READ_BUFLen, &retCnt);
}

```

9.3.7 Command Synchronization

The following examples illustrate the methods for command synchronization with sweep process.

```

void SweepSync()
{
    ViState state;
    long retCnt;
    ViEventType etype;
    ViEvent eevent;

```



```

int stat;
char OpcOk [2];
/*****/
/* The command INITiate[:IMMEDIATE] is used to start single sweep (when continuous
sweep is OFF, INIT:CONT OFF)*/
/* Only at the end of single sweep can the next command in the command buffer be
executed */
/*****/
status = viWrite(analyzer, "INIT:CONT OFF", 13, &retCnt);
//Method 1 for waiting for the sweep to end: use *WAI
status = viWrite(analyzer, "ABOR;INIT:IMM;*WAI", 18, &retCnt);

//Method 2 for waiting for the sweep to end: use *OPC?
status = viWrite(analyzer, "ABOR;INIT:IMM; *OPC?", 20, &retCnt);
status = viRead(analyzer, OpcOk, 2, &retCnt); //wait for *OPC to return "1"

//Method 3 for waiting for the sweep to end: use *OPC
//To use the GPIB service request, set "Disable Auto Serial Poll" to "yes"
status = viWrite(analyzer, "**SRE 32", 7, &retCnt);
status = viWrite(analyzer, "**ESE 1", 6, &retCnt); //enable service request ESR
//Set the event enabling position, and end the operation.
status = viEnableEvent(analyzer, VI_EVENT_SERVICE_REQ, VI_QUEUE, VI_NULL);
//Enable the SRQ event
status = viWrite(analyzer, "ABOR;INIT:IMM;*OPC", 18, &retCnt);
//Start sweeping together with OPC
status = viWaitOnEvent(analyzer, VI_EVENT_SERVICE_REQ, 10000, &etype, &eevent)
//Wait for service request
state = viReadSTB(analyzer, &stat);
state = viClose(eevent); //close event handle
//Disable SRQ event
status = viDisableEvent(analyzer, VI_EVENT_SERVICE_REQ, VI_QUEUE);
//Main program continues.....
}

```

9.4 I/O library

9.4.1 Overview of I/O Library

I/O library is a pre-written software library for instruments, known as instrument driver. As a software between the computer and the instrument hardware, it consists of the function library, utility program, tool kit, etc. It is a combination of a series of software code modules and

9.4 I/O library

corresponds to operation of a plan, such as configuring the instrument, reading from the instrument, writing to the instrument and trigger the instrument, etc. Residing in the computer, it is the bridge and link between the computer and the instrument. By providing a high-level modular library for convenient programming, users no longer need to learn the complex low-level programming protocol for a specific instrument. Application of instrument driver is the key to develop test and measurement applications quickly.

Functionally, a universal instrument driver generally consists of five parts: functor, interactive developer interface, programmer interface, subprogram interface and I/O interface, as shown in Figure 9.14.

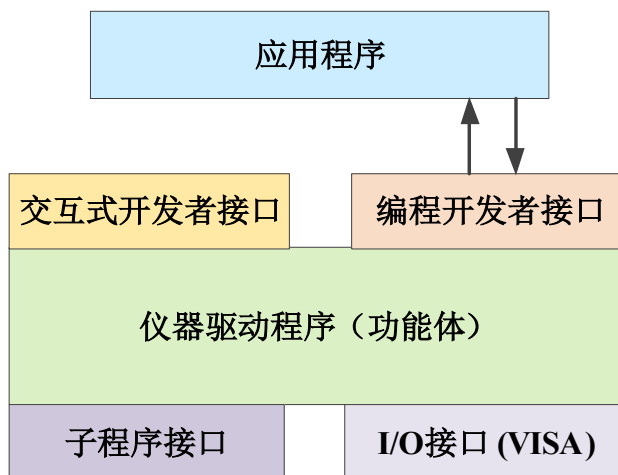


Figure 9.9 Structure model of instrument driver

The details are as follows:

- 1) Functor. As the main function part of the instrument driver, it may be understood as its framework program.
- 2) Interactive developer interface. Application development environment that supports instrument driver development is usually provided with graphical interactive developer interface for user convenience. For example, in Labwindows/CVI, the function panel is an interactive developer interface. In the function panel, each parameter of the instrument driver function is represented as a graphical control.
- 3) Programmer interface. It is a software interface for the application to call instrument driver function, such as dynamic link library file.dll of instrument driver in Windows system.
- 4) I/O interface. It completes the actual communication between the instrument driver and the instrument. The bus-specific I/O software or the generally-standardized I/O software applied cross multiple buses (VISA I/O) can be used.
- 5) Subprogram interface. It is a software interface for the instrument driver to access other support libraries, such as databases, FFT functions, etc. The subprogram interface is used when the instrument driver needs to call other software modules, operating systems, programmed control code libraries and analysis function libraries to complete

its task.

9.4.2 Installation and Configuration of I/O Library

Along with the application in test field, it has gone through different development stages from traditional instrument to virtual instrument. In order to solve the interchangeability of instruments and reusability of test program in automatic test system, instrument driver has gone through different development processes. IVI (Interchangeable Virtual Instruments) driver is relative popular and common at present. Based on IVI specification, it defines a new instrument programming interface, inserts the class driver and VPP architecture onto the VISA to make the test application and instrument hardware completely independent, adds such unique functions as instrument simulation, range sensing and status cache, improves the operation efficiency of the system, and realizes instrument exchange.

There are two types of IVI driver: IVI-C and IVI-COM, where the latter adopts the form of COM API based on the component object model (COM) of Microsoft, and the former adopts the form of C API based on ANSI C. Both types are designed according to the instrument class defined in the IVI specification and have the same application development environment, including Visual Studio, Visual Basic, Agilent VEE, LabVIEW, CVI/LabLinux , etc.

The IVI driver supported by the vector network analyzer includes IVI-C and IVI-COM. For specific installation and configuration, please refer to the attached documentation of the control card and I/O library you selected.

IVI driver after installation are divided into IVI inherent function group and instrument function group (basic function group and extended function group). Specific function classification, functions and attributes are shown in the help document of the driver.

Tips

Port configuration and IO library installation

Before using a computer controlled vector network analyzer, please verify that you have properly installed and configured necessary ports and I/O libraries.

Tips

Use of I/O library

The driver function panel, help document and driver function examples will be installed automatically when installing the attached IVI-COM/C driver installation package, so as to facilitate users to develop integrated programmed control functions.

10 Troubleshooting and After-sales Services

This chapter will show you how to find problems and accept after-sales service, and explain error message of the network analyzer.

If you encounter any problem when operating the 3657 series vector network analyzers or want to buy relevant components or accessories, this institute can provide you with complete after-sales services.

Generally, causes of problems include hardware, software or user maloperation. In case of any problem, please contact us in time. If the spectrum analyzer you purchased is still under warranty, we will repair your spectrum analyzer free of charge as promised in the warranty; Otherwise, we will only charge the cost.

- [Working Principles.....355](#)
- [Troubleshooting and Debugging.....357](#)
- [Errors.....359](#)
- [Method to Obtain After-sales Services.....380](#)

10.1 Working Principles

10.1.1 Response of DUT to the RF signal

The signal source of the vector network analyzer generates the test signal and inputs it to the DUT. As the test signal passes through the DUT, some of the signal is reflected and the rest is transmitted. Figure 10.1 illustrates the response of the test signal after it passes through the DUT.

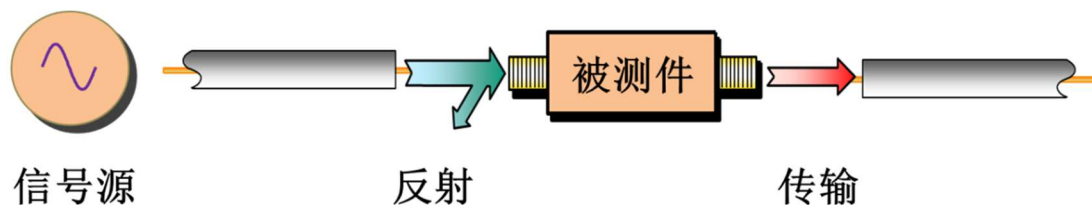


Figure 10.1 Response of DUT to the signal

10.1 Working Principles

10.1.2 Analyzer Principles

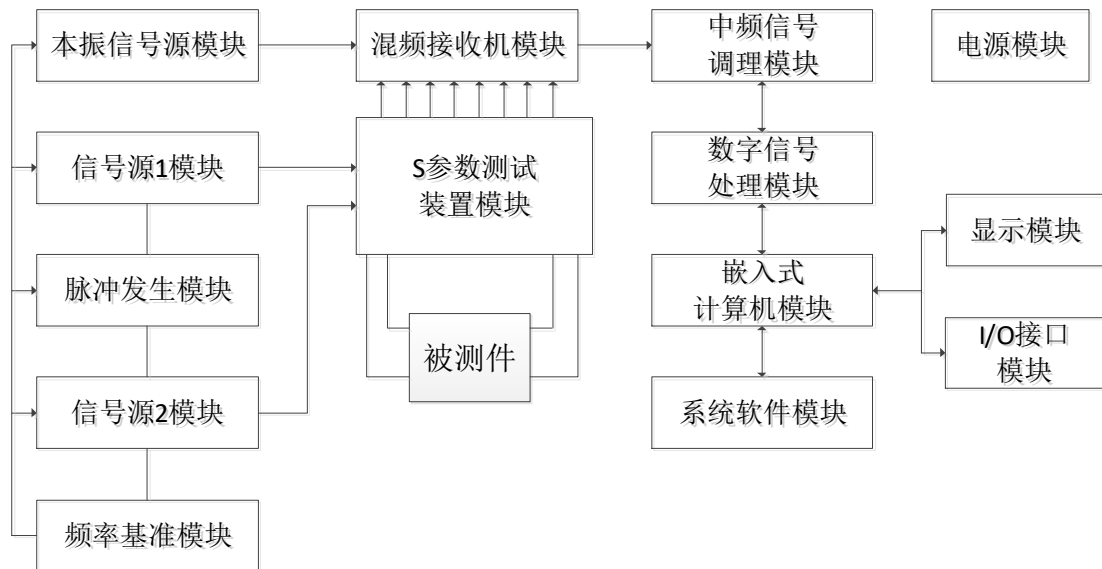


Figure 10.2 Block diagram of the analyzer principle

The 3657 series vector network analyzer is an intelligent test instrument of modular design. With embedded industrial computer module as the control center, this instrument, by applying design concept of "embedded computer hardware platform + test module based on multi-processor", realizes the perfect combination of test instrument and industrial computer, and the adoption of test and control software compatible with Windows OS makes the human-machine interface of the analyzer more user-friendly and more convenient to operate. The modular design block diagram of the network analyzer is shown in Figure 10.2, and the main modules of the system include LO signal source module, signal source 1 module, signal source 2 module, frequency reference module, S-parameter test settings module, mixing receiver module, IF signal processing module, digital signal processing module, embedded computer module, power supply module, display module, I/O interface module, and system software module, etc.

The frequency reference module is phase locked on the internal/external 10MHz reference signal to generate the phase-locked reference signal and VNA sync signal required for LO source and signal source.

The LO signal source module and the stimulus signal source module are both phase locked to the reference signal output by the frequency reference module, generating the required LO signal and RF signal through ON/OFF frequency doubling, subband bandpass filtering and automatic level control circuit (ALC).

The S-parameter test settings module is a signal separator, which separates the incident signal, reflection signal and transmission signal of the DUT.

The mixing receiver module mixes the incident signal, reflection signal and transmission signal containing the magnitude and phase information of the DUT with the LO signal, generates an IF signal and sends it to the IF signal conditioning module for further processing.

10.2 Fault Diagnosis and Troubleshooting

The IF signal conditioning module conditions the IF signal to meet the requirements of the digital signal processing module for the IF signal.

The digital signal processing module converts the analog IF signal into a digital signal, and then further processes the digital signal into magnitude and phase information of the DUT, and sends the measurement results through the high-speed PCI bus to master CPU.

System test and control software running on the embedded PC platform converts the magnitude and phase information of DUT into various formats, and then sends the results to the display module. In addition, the system software is also responsible for the management of various interfaces and the scheduling of various processes of the network analyzer.

10.2 Fault Diagnosis and Troubleshooting

Tips

Troubleshooting and instructions

This section introduces the way on how to judge and handle failures (if any) of the 3657 series network analyzer, and feed them back to the manufacturer as accurately as possible if necessary for quick solution.

If there is an error prompt in the user interface status indication zone of the analyzer, please click "Help→Error→Check Error Log" to understand the specific Error description.

- [System Failure.....357](#)
- [Abnormal Curve Display.....358](#)
- [Sweep Failure.....358](#)
- [Display Failure.....359](#)
- [No Response From Front Panel Keys.....359](#)
- [Menu Operation Failure.....359](#)

10.2.1 System Failure

10.2.1.1 Standby Lamp Not On

Check whether the AC input of the vector network analyzer is normal, which cannot be too high or too low. Otherwise, the analyzer will run abnormally. The maximum allowable deviation is 220V±10% in China, and 110V±5% in some foreign countries. If it is abnormal, check the external lines for any failure. After troubleshooting, power on the instrument again and start it. If the AC power input is normal, check whether the power button on the rear panel of the analyzer is turned on, which shall be in position “ | ” in normal state. If the failure is caused by the power supply of the instrument, send the product back to the manufacturer for repair.

10.2 Fault Diagnosis and Troubleshooting

10.2.1.2 Fail to Access the System After Power-on

If you cannot log in to the system after power-on, check whether hard disks on the rear panel are loose. If the hard disk is loose, tighten the screws or remove the hard disk and reinsert it. If the hard disk is securely installed, restore the system. For details, see "3.1.2.1 System restoration of the analyzer".

10.2.1.3 Fail to Start the Network Analyzer Software Automatically After the System is Started

If the network analyzer software cannot be automatically started after the system is started, double-click the icon  of the vector network analyzer on the desktop or press the [Reset] button on the front panel, or click Start → Program → Vector Network Analyzer from the Windows system, and then click Vector Network Analyzer to start vector network analyzer program. If the system starts successfully, install antivirus software to check for viruses and restart the system. If it does not start automatically, uninstall the vector network analyzer program and double-click Vector Network Analyzer Installation Program in the root directory of drive D or E, and refer to "3.1.2.2 Updating and installation of the network analyzer program" to reinstall the vector network analyzer program.

10.2.2 Abnormal Curve Display

There are many reasons for the abnormal curve display. Firstly, check whether the user reset state has been saved. Click System→ Define User Resetting State on the menu bar to open the Define User State window. Uncheck ☒ in the check boxes before "Save last state as user reset state" and "Enable user reset state" and click OK to close the Define User State window. Press the [Reset] key on the front panel, and the S11 curve displayed should be near 0 under the condition that the port is not connected to any load.

If the curve display is still abnormal after the above method is tried, please contact our technical support personnel to solve the problem.

10.2.3 Sweep Failure

10.2.3.1 Suspected Sweep Stop

During system resetting, the curve displayed by the open port will give an illusion that the sweep stops, which is caused by high stability and high sweep speed of the vector network analyzer. Click **Stimulus**→**Sweep**→**Sweep Time**, enter 20 and press Enter, and then the sweep cursor appears.

10.2.3.2 Sweep Stopped

Check whether the trigger state of the instrument is "Hold". If yes, the trigger mode needs to

10.3 Error Message

be modified. Press Stimulus → Trigger → Continuous Sweep, or open the Trigger dialog box to change the trigger mode to automatic trigger.

10.2.3.3 The White Sweep Cursor is at the Bottom of the Screen

After completing the dual-port calibration, both ports are swept each time for dual-port correction. When the displayed trace is only the trace of the stimulus source in port 1, a white cursor will appear at the bottom of the screen when sweeping port 2. It is the normal instrument sweep, with no need to change the setting.

10.2.4 Display Failure

When the taskbar blocks the display window, you can right-click Taskbar, click Properties, select Auto Hide taskbar, and click OK to close the dialog box. In this case, the taskbar will automatically hide without blocking the network analyzer test window.

10.2.5 No Response From Front Panel Keys

If the front panel keys do not respond, which is caused by the damage of the instrument driver, it is necessary to uninstall and reinstall the vector network analyzer program. For details, refer to "3.1.2.2 Update and installation of network analyzer program". Note: when selecting the installation type, select Full to reinstall the driver. If the fault persists, restore the system. For details, see "3.1.2.1 System restoration of the network analyzer".

10.2.6 Menu Operation Failure

If a menu is found unable to be operated, first check whether the state shown in the status bar at the bottom of the window is LOCAL. If the program is already in the programmed control state, please press the [Macro/Local] button on the front panel or the [Esc] button on the keyboard to set back to LOCAL state.

10.3 Error Message

When an error occurs in a vector network analyzer, both the SCPI (remote interface) error queue and the front panel display error queue will report an error. The two queues are independent of each other in terms of display and handling. For details on the front panel error queue, please refer to the User's Manual.

When using the SCPI (remote interface) error queue to access the error message, the error code and <error description> responded by the error queue are displayed on the host terminal.

The error message of the vector network analyzer can be displayed and recorded in the error file. The user can choose how to display the error message or choose not to display the error message at all.

10.3 Error Message

10.3.1 Local Error

10.3.1.1 Error View

View via interface operation:

If an error or warning prompt is displayed on the testing interface of the analyzer during the use, the software running or hardware of the analyzer is faulty. Vector network analyzer error logs record all prompts, warnings, and error events that occur during the running of vector network Analyzer. The error log allows the user to view the date and time when the selected prompts, warnings, and error details occurred, and to click Help for further information. The vector network analyzer can only display one error prompt at a time. Since multiple errors can occur to the network analyzer, to see all errors, do the following:

- Step 1.** Click [Help]→[Error Message]→[Check Error Log] and then an error log window will pop up.
- Step 2.** The prompt will be displayed in the window.
- Step 3.** Use the mouse to browse the error and close the dialog window.
- Step 4.** Select "Clear error list" to clear historical errors.

10.3.1.2 Error Description

If an error is detected during the measurement of the vector network analyzer, an alarm or error message is displayed on the test interface. By default, the error message is displayed for a period of time. Users can choose to keep them on the screen until they press [OK], or choose not to show them at all. When they appear on the screen, you can select →[Check Fault] in the right menu for further help information, as shown in Figure 10.3 below:



Figure 10.3 Error message display mode dialog box

Table 10.1 Font color distinction error level

10.3 Error Message

Color	Error level	Error Description
Red	Error	Errors occurred during measurement, such as data loss or incorrect settings that prevents the measurement from being completed properly.
Blue	Warning	Irregular working conditions during measurement, such as mismatch between instrument settings and display results, or temporary disconnection from externally connected instruments, making measurement no longer possible.
Black	Prompt	Step setting or measurement information.

10.3.2 Programmed Control Error Information

Error format description

In remote control mode, errors are recorded in the error log, and can be queried with the command "SYSTem:ERRor?". The format is as follows:

"Time:"

"Error code:"

"Error level: "

"Error type:"

Example:

Time: 2014/11/16 20:40:21

Error code: 1400

Error level: Error

Error code: syntax error

The programmed control error includes two types:

- A negative error code defined by the SCPI standard. This type of error is not specified here.
- The positive error code of the network analyzer features, with details shown below:

Table 10.2 List of descriptions on instrument feature error messages

Failure code	Error level	Error Description
512	Prompt	The secondary parameters of the calibration status (power, IF bandwidth, sweep time, step mode) have changed. Type: Changing such secondary parameters after the calibration may make the measurement inaccurate.

10.3 Error Message

513	Prompt	The calibration cannot be completed as the measurement of all standards has not be completed. This message may appear because an attempt was made to enable the error correction function before completing measurement of all calibration standards.
515	Prompt	Correction cannot be turned on (channel <x>, measurement of <x> does not have corresponding error correction data), please perform calibration first. No corresponding error correction data for specified channel and measurement.”
516	Prompt	The key parameters of the current instrument state do not match the calibration set, and the correction is turned off. Key parameters include: sweep type, start frequency, frequency span, and number of points.
517	Prompt	Because the interpolation function is disabled and the stimulus setting has changed, the correction is turned off. The most accurate calibration is to use the original stimulus settings.
518	Prompt	The interpolation function is turned off and the correction function is selected to restore the previous stimulus settings.
519	Prompt	The stimulus range is beyond the original calibration setting and the correction is turned off. Correction data that exceeds the stimulus settings does not exist.
520	Prompt	Channel <x> Measurement <x> has a calibration type of 'None'; please select the calibration type via the calibration menu or by pressing the button. The calibration cannot be performed because the calibration type is not selected or the calibration does not exist. This error is generated because the calibration was not found or the calibration was performed without selecting the calibration type.
521	Prompt	The calibration type does not match the selected measurement, the correction is turned off or no correction is allowed. Some calibrations apply only to certain S-parameters. For example, 1-port calibration for S11 cannot be applied to 1-port calibration of S22.
522	Prompt	Need to measure more standard parts.
524	Prompt	Hint: non-full dual two-port calibration.

525	Prompt	Hint: Two sets of calibration kits are used.
526	Prompt	No user calibration data suitable for this channel was found. Calibration does not exist and cannot be corrected.
527	Prompt	This calibration type does not require this standard This error is generated when requesting an unspecified calibration standard during a calibration procedure.
528	Prompt	The electrical calibration system cannot be configured. Please check if the module is properly connected. During the electrical calibration operation, the network analyzer must first establish normal communication with the electrical calibration module and perform module verification, otherwise the electrical calibration cannot be performed.
529	Prompt	Data out of bound: Exceeding the designed value range. Key parameters include: sweep type, start frequency, frequency span, and number of points.
530	Prompt	Execution error: Number of array dimensions not match.
531	Prompt	Execution error: Array being divided by zero.
532	Prompt	Execution error: Invalid array function.
533	Prompt	Execution error: Array subscribe error.
534	Prompt	Execution error: Array subscribe out of bound.
535	Prompt	Execution error: Wrong matrix rank.
536	Prompt	Execution error: CPU.
537	Prompt	Execution error: Module cannot be deleted.
538	Prompt	Execution error: Module cannot be written.
539	Prompt	Execution error: No input.
540	Prompt	Execution error: System is busy, invalid command.
541	Prompt	Electric calibration: The module is not in the RF path. The module is not in the RF path, please connect it to the designated port. The RF is not connected to the electrical calibration module and no signal was found during the corresponding calibration measurement.
542	Prompt	Execution error: No space for storing new calibrations, please delete one. Key parameters include: sweep type, start frequency, frequency span,

10.3 Error Message

		and number of points.
543	Prompt	Execution error: Insufficient space.
544	Prompt	Execution error: Other matrix errors.
545	Prompt	Execution error: Unequal rank.
546	Prompt	Execution error: Too few constant ranks.
547	Prompt	Execution error: Too few constant ranks.
548	Prompt	Perform a pre-calibration.
549	Prompt	Execution error: Electrical calibration drive dynamic library error or invalid module number.
550	Prompt	Execution error: Unexpected error from the electrical calibration drive.
551	Prompt	Execution error: Internal drive error.
552	Prompt	Hardware error: Cannot access the electrical calibration module.
553	Prompt	Hardware error: Cannot release LPT port, restart.
554	Prompt	Hardware error: Vector network analyzer error.
555	Prompt	Hardware error: Failed to read enough data from the electrical calibration module.
556	Prompt	Operation aborted by host computer.
557	Prompt	Operation aborted by user.
558	Prompt	Memory out of bounds.
559	Prompt	The terminated query: message abandoned.
560	Prompt	Unfinished query: Incomplete program message.
561	Prompt	Unfinished query: No content.
562	Prompt	Queue overflow
563	Prompt	Setting conflict: Additional standards required.
564	Prompt	Setting conflict: Adapter calibration not possible.
565	Prompt	Setting conflict: Command queue overflow.
566	Prompt	Setting conflict: Calibration stopped - Analyzer settings changed.
567	Prompt	Setting conflict: Calibration not done.
568	Prompt	Setting conflict: Unable to find the specified GPIB board.

569	Prompt	Setting conflict: unable to find/load gpib32.dll.
570	Prompt	Setting conflict: unable to find/load sicil32.dll.
571	Prompt	Setting conflict: Unable to initialize network analyzer (wrong address?).
572	Prompt	Setting conflict: Unable to load print port driver or USB driver dynamic library.
573	Prompt	Setting conflict: Invalid calibration sweep mode.
574	Prompt	Setting conflict: Invalid calibration type.
575	Prompt	Setting conflict: Invalid calibration.
576	Prompt	Setting conflict: invalid GPIB board number specified.
577	Prompt	Setting conflict: invalid GPIB board type specified.
578	Prompt	Setting conflict: invalid module status.
579	Prompt	Setting conflict: invalid status.
580	Prompt	Setting conflict: LPT port must be 1~4.
581	Prompt	Unable to configure the electrical calibration system, check that the module is properly connected.
582	Prompt	Setting conflict: specified LPT port does not exist.
583	Prompt	Setting conflicts: please use the frequency domain for calibration.
584	Prompt	Setting conflicts: Please use the frequency domain for calibration.
585	Prompt	Setting conflict: analyzer address must be 0~30.
586	Prompt	Setting conflicts: wrong print port driver or USB driver dynamic library.
587	Prompt	Syntax error: ECAL:DELAY command must have two numbers.
588	Prompt	Syntax error: incorrect syntax.
589	Prompt	Syntax error: unknown command.
590	Prompt	The module port on the RF path is incorrect.
591	Prompt	Not the module described by the user.
592	Prompt	The source power calibration data corresponding to current measurement channel and source port is not found. An attempt was made to turn on source power calibration but no calibration data.

10.3 Error Message

593	Prompt	The source power calibration sweep cannot be performed. The channel and source port corresponding to the current measurement cannot be corrected. An attempt was made to turn on source power calibration but no complete calibration data.
594	Prompt	Reserve.
595	Prompt	Reserve.
596	Prompt	With calibration turned on, the calibration power of the source power calibration in channel %2 of port %1 has changed. The calibration will not be turned off, but the power value may no longer characterize the calibration. Source power calibration accuracy may be in fault.
597	Prompt	This software version does not support the electrical calibration function at present.
598	Prompt	When measuring the sliding load standard, improper sliding will not correctly fit a circle. The standard raw impedance is used to determine the directionality of one or more points. To accurately describe the standard, the sliding load must be properly slid to ensure adequate sampling along the complex circle or Smith chart. Undersampling may cause inaccurate results.
599	Prompt	This feature requires an unused channel but cannot be found. Please release a channel and try again.
600	Prompt	The original calibration cannot be interpolated in the segment sweep mode. Calibration is off.
601	Error	Unable to complete calibration (standard measurement not completed)
602	Error	Correction not turned on.
603	Error	Power correction cannot be turned on. Please perform power meter calibration first.
604	Error	The sweep state changes, the power correction data is invalid, and the power correction is turned off.
605	Error	The power correction is turned on and the sweep restores the original state. Key parameters include: sweep type, start frequency, frequency span, and number of points.

608	Error	Calibration type not set. The calibration can only be performed if the calibration is present or the correct calibration type is selected.
609	Error	Calibration feature cannot be achieved. The specified calibration may be one of several options, for example: Response calibration requires a single standard, 1-port calibration requires 3 standards, and full dual-port calibration requires 12 standards.
610	Error	The calibration class is not valid for the current calibration type. Please choose a different calibration class or a different calibration type.
611	Error	No calibration standard data corresponding to the selected calibration type was found. Raw measurement data for the specified standard was not found in the current measurement buffer.
612	Error	No error term data corresponding to the selected calibration type was found. No specified error term in the error correction buffer (this buffer contains all the error coefficients of the current calibration).
613	Error	Reserve. Access to the calibration set was unsuccessful and the calibration set may have been deleted or corrupted.
614	Error	The specified measurement is not compatible with the current calibration. Please select a different measurement, or load another different calibration type, or recalibrate. Block measurement selection so that measurement calibration is not turned off. Not all measurements are supported for each calibration type, for example: S11 1-PORT calibration cannot be used to calibrate S12 measurements. When a measurement that is not supported by the current calibration is selected, a message is displayed and the calibration is turned off.
615	Error	new calibration set created. The newly created calibration set will be automatically named and time stamped. The calibration set will only be saved if the calibration process is completed; if the calibration is aborted or not completed, the calibration set will be deleted.

10.3 Error Message

617	Error	Pre-tuning calibration failed
620	Error	Measuring status changes, turn off the antenna, RCS calibration.
621	Error	Measuring status changes, unable to turn on the antenna, RCS calibration.
622	Error	Invalid calibration data, unable to turn on the antenna, RCS calibration.
634	Error	calibration set file: <x> loading failed. The calibration kit file is a collection of calibration kits stored on the hard disk.
635	Error	calibration set file: <x> not saved. File operation incorrect, saving operation aborted.
636	Prompt	A calibration kit deleted. A calibration set was successfully deleted, which may be the result of a user application or intentional operation.
637	Error	Calibration kit file version: <x> is not compatible with the current measurement. Version mismatch makes the calibration kit unavailable, which may be due to instrument software upgrades.
638	Error	Incompatible calibration set found: <x> of <y> stored calibration set was loaded. Some calibration sets in the calibration set file have errors.
639	Prompt	calibration set file: <x> not found, a new file will be created. The calibration set file is stored on the hard disk. When the network analyzer program is started, it will search for the existence of the file. If it exists, it will be loaded. Otherwise, a new file will be created and the information will be prompted.
640	Error	Specified calibration set is in use. It indicates that multiple calibration set users are attempting to perform calibration and cause conflicts.
641	Error	Unable to open specified calibration set. Multiple users may be attempting to access this calibration set.
642	Error	Achieving the maximum register number of the calibration set. Delete old or useless calibration sets before attempting to create a new calibration set.

10.3 Error Message

		The network analyzer can set up to 100 calibration sets.
644	Error	A valid calibration must be performed before the error correction is turned on. This usually indicates that the calibration process has not been completed or that there is no calibration that matches the selected measurement in the currently selected calibration set.
646	Error	Calibration set not loaded, version too new. An older version of the software attempts to use the new calibration file, the version is not compatible.
772	Error	The DSP board driver can not work properly, please check the hardware. The software will use the simulated data! The network analyzer is unable to communicate with the DSP board. Possible fault in the hardware or driver.
773	Error	The serial bus of the network analyzer cannot work properly. The serial EEPROM of the instrument may all be '1' or all be '0', which may be caused by a problem of the serial bus hardware.
774	Error	IF overload.
848	Error	Losing lock<x>. The source cannot properly lock the phase. This may be caused by hardware failure, poor calibration, or damaged data in the serial EEPROM.
849	Error	Phase locking failed <x>.
850	Error	Unknown hardware error. A hardware failure prevented communication with the DSP.
855	Error	Source unlevelled. The source cannot be properly leveled to the required power. It indicates that the power is not accurate.
856	Error	LO unlevelled.
859	Error	YIG calibration failure. Internal self-calibration tuning of the YIG oscillator failed.
861	Error	Simulated slope calibration failed. Internal simulated slope calibration failed.
865	Error	EEPROM Write Fails An attempt to store calibration data to the EEPROM failed. It could be

10.3 Error Message

		caused by a hardware failure.
867	Error	An attempt was made to write an input or read-only I/O port. An attempt was made to write to an input or read-only I/O port.
868	Error	An attempt was made to read the write-only I/O port. An attempt was made to read an output or write-only I/O port.
1025	Error	The network analyzer does not shut down properly. The program is in an unstable state and needs to be restarted. This message is displayed when the vector network program crashes. If this message persists, please consult the Service Center.
1026	Error	Incorrect limit segment type. There are three types of limit segments: OFF, MAX and MIN.
1027	Warning	Group delay format cannot be used in point frequency or power sweep mode In the single frequency point sweep mode, the group delay format cannot be used, otherwise invalid data will be generated.
1028	Warning	The limit line test failed. The limit line test failed.
1029	Prompt	Limit line test is successful. Limit line test is successful.
1030	Warning	Exceeding the maximum number of tests allowed. You can only create up to 512 measurements.
1031	Error	Internal error in the analyzer. An error occurred while adding a new measurement. If this message persists, please consult the Service Center.
1032	Warning	No measurement was found for the selected operation. The operation could not be performed. conflict:
1033	Warning	You cannot create two measurements with the same name
1034	Warning	The selected trace has not stored a storage trace. Please save a storage trace before tracing. Before performing the tracing calculation, you must have saved a storage trace.
1035	Prompt	Sweep averaging done.

10.3 Error Message

		COM programming information. The sweep averaging is done, achieving the set average factor.
1036	Error	Reset sweep averaging done. COM programming information. The sweep averaging is not completed, the set average factor is not achieved.
1037	Prompt	Time domain conversion and gate operation require at least 3 points, or they will be canceled.
1038	Prompt	Step type can only be used when the time domain measurement type is low pass; bandpass can only be used when the stimulus type is pulsed.
1039	Warning	Too few points to perform smoothing.
1040	Warning	Too many time-domain low-pass points, exceeding the frequency range, automatically switched to band-pass mode by the system
1041	Warning	Current configuration cannot set the low pass frequency.
1103	Prompt	Repeated measurement name.
1104	Error	Exceeding the limit for measurement number. You can only create up to 512 measurements.
1105	Error	Invalid parameter. The measurement parameters entered during programming are invalid.
1106	Error	No measurement found. conflict:
1107	Error	Storage trace invalid, unable to operate The trace needs to be stored before trace operation.
1108	Error	No reference cursor. An attempt was made to create a Δ cursor function (for COM only) before creating a reference cursor
1109	Error	Data and storage trace not match, trace operation turned off. Warning - The channel settings were changed while performing the trace operation.
1110	Error	Data and storage trace not match. For trace operations, memory and data traces should have similar measurement conditions. An attempt was made to perform a trace operation on unmatched data and storage traces.

10.3 Error Message

1111	Error	Cursor bandwidth not found. The portion of the trace that meets the specified bandwidth criteria cannot be found.
1112	Error	Peak value not found. The portion of the trace that meets the specified peak value standard cannot be found.
1113	Error	Target search value not found. No interpolated data points matching the search value were found.
1114	Error	Reflex measurements (such as S11) must provide an auxiliary port to clearly identify 2-port measurements on a multi-port instrument.
1115	Warning	Receiver power calibration disabled and turned off due to measurement type or source port changes.
1116	Warning	Receiver power calibration requires activation of non-ratio power measurements.
1117	Warning	The current active measurement does not have a suitable source power calibration that matches the channel and source port. Before performing receiver power calibration, source power calibration or callback-one source power calibration should be done.
1118	Error	Attempted operation only suitable for standard measurement types.
1119	Error	Custom measurements cannot be loaded because they do not match the network analyzer hardware.
1120	Error	Custom measurements cannot be loaded because they do not match the network analyzer software.
1121	Error	Custom measurement load failed for unknown reasons.
1122	Error	Custom measurement data processing generates an exception and will be terminated. The network analyzer software may be in an unstable state. It is recommended to close the program and re-run.
1123	Error	Attempted operation only suitable for custom measurement types.
1124	Error	Requested custom measurement not available.
1125	Error	Requested custom algorithm does not exist.
1126	Error	Normalization cannot be performed because the measurement does not have a valid divisor memory zone.

10.3 Error Message

1127	Warning	Unable to provide raw data required for the measurement.
1128	Prompt	Selected sweep type does not allow time domain conversion or gate addition, time domain conversion or gate addition prohibited.
1284	Prompt	State word of instrument changed.
1285	Prompt	Error code caused by received SCPI command is <x>: <x>.
1400	Warning	<X> command error.
1401	Warning	Command parameters must be entered for <x> .
1402	Warning	Incorrect command parameters entered for <x> .
1403	Error	<x> command cannot be queried.
1405	Error	Too few parameters following <x> command.
1406	Error	Too many parameters following <x> command.
1407	Error	Incorrect parameter type entered for <x>.
1408	Error	Incorrect parameter unit for <x>.
1409	Error	Parameter unit for <x> not matching.
1411	Error	Incorrect input of <x> string parameter.
1412	Error	Incorrect input of <x> integer parameter.
1413	Error	Incorrect input of <x> floating-point parameter.
1414	Error	Incorrect input of <x> char parameter.
1415	Error	<x> parameter cannot be used with units.
1416	Error	More than 50 characters used in <x> string.
1419	Error	<x> command must be queried.
1420	Error	Actual number of characters entered for <x> not meeting the requirement.
1421	Error	Incorrect ASCII code entered.
1422	Error	Incorrect input of scientific notation for <x>.
1425	Error	Memory allocation failed.
1427	Error	MIN or MAX parameter input error.
1535	Prompt	Command obtained by parsing: <x>
1536	Warning	Each window supports up to 8 traces, and trace <x> cannot be built in window <x>.

10.3 Error Message

1537	Warning	With more than 32 data windows created, you cannot create any new window.
1538	Warning	Data window does not exist, measurement cannot be completed. Programmable SCPI operation attempts to create new measurements with no window displayed.
1539	Warning	No data trace in the selected window, operation cannot be completed.
1540	Warning	Exceeding the limit of up to <x> traces per window, impossible to arrange existing measurements in <x> windows. The upper limit of up to 8 traces per window has been exceeded, you cannot place existing measurements into the specified window. Refer to arranging the measurement help that exists.
1541	Warning	Unable to establish connection with specified printer.
1542	Prompt	Cancel the printout.
1616	Error	No window found. Specified window in the program does not exist.
1617	Error	Copy the ID of the specified window.
1618	Error	Exceeding the limit of 32 windows to be displayed, no new windows can be created. Up to 32 windows per screen.
1619	Error	Exceeding the limit of up to 8 traces to be displayed in one window, trace <x> cannot be created. Up to 8 traces per window. Refer to related help documents for traces channels, and windows.
1620	Error	No trace found.
1621	Error	Operating system cannot recognize the printer.
1622	Error	No active trace.
1623	Error	Trace already exists.
1624	Error	Start or stop value cannot be set in the 'full bandwidth' state.
1625	Error	Specified cursor does not exist.
1626	Prompt	Specified window already exists.
1792	Prompt	Sweep completion

1793	Prompt	All trigger requests completed.
1794	Prompt	Cursor bandwidth not found. The portion of the trace that meets the specified bandwidth criteria cannot be found.
1795	Prompt	To use the segment sweep method, you must ensure that there is at least one active segment with more than 0 sweep points in the segment list. An attempt was made to change the sweep mode to segment sweep mode, but there is no defined segment or there is no sweep point in the defined segment.
1796	Prompt	Setting channel spam. This message appears because the channel settings have changed, but the data is still obtained according to the previous channel settings. The message Clear Channel Spam will be generated when the data is set according to the new channel settings.
1797	Prompt	Clear channel spam The previous setting message occurred when the channel settings have changed but the channel still has data obtained with the original channel settings. This clear message will occur when new channel data is obtained.
1798	Prompt	An error occurred when deleting the segment.
1870	Error	No measurement is selected. No measurement is selected.
1871	Error	Channel number out of range! The specified channel number exceeds the allowable range!
1872	Error	No channel found. A channel that does not exist is specified under programmed control.
1873	Error	Requested sweep segment not found. A sweep segment that does not exist is specified under programmed control.
1874	Error	Sweep segment list is empty. At least one segment should be defined in the segment table sweep mode. This error only occurs during remote control.
1875	Error	The sweep points for an active sweep segment list is 0.

10.3 Error Message

		In the segment list sweep mode, you must ensure that there is at least one segment with data points. This error only occurs during remote control.
1876	Error	Invalid source attenuation specified. An attempt was made to set the channel attenuation characteristic in a vector network without a source attenuator.
1877	Error	Too many sweep points, unable to use logarithmic sweep mode. Please reduce the sweep points. For logarithmic sweep, the maximum sweep points that can be used is 401.
1878	Error	Number of points set greater than the number that can be selected for logarithmic sweep mode. For logarithmic sweep, the maximum sweep points that can be used is 401.
2048	Error	This feature can be used only after adding options on the basis of a standard analyzer.
2049	Error	Current measuring instrument does not have the features you need.
2050	Error	The feature you need conflicts with the current state of the network analyzer.
2051	Prompt	File <x> saved.
2052	Error	File <x> not saved.
2053	Error	An attempt to open the file failed, file <x> not found.
2054	Error	<X> header error.
2055	Error	Invalid state file, cannot be loaded!
2056	Error	Request to enter sleep mode.
2057	Error	Wake up from sleep mode. The program received a PBT_APMRESUMEAUTOMATIC message.
2058	Error	Wake up from standby mode. The program received a PBT_APMRESUMESUSPEND message.
2059	Warning	Wake up from standby mode. The program received a PBT_APMRESUMECRITICAL message.
2060	Warning	Wake up from unknown sleep mode. The program received a NO PBT_Message message upon allocation and attempt to recover.

2061	Error	File <x> already exists. Overwriting file. Only used in remote applications.
2063	Error	File <x> recovered. Only used in remote applications.
2064	Error	<x> version lower than current version. You are trying to recover a file that is no longer valid.
2065	Error	<x> version newer than latest version. An attempt was made to recover the latest version of a file generated by a network analyzer software.
2066	Error	An error occurred while reading file <x>. This file may be damaged.
2067	Error	Window kernel error: <x>
2068	Error	<x>GPIB: unable to set and clear REN.
2069	Prompt	Converting GPIB mode to system controller mode.
2070	Prompt	Converting GPIB mode to listen-speak mode.
2071	Prompt	The network analyzer can only be set to GPIB system controller mode when the GPIB status is local. Stop any remote network analyzer program and press the Macro/Local button to try again. Refer to LCL and RMT operations
2072	Error	Range setting error.
2073	Error	Configuration changed, please sweep again. The portion of the trace that meets the specified bandwidth criteria cannot be found.
2074	Error	GPIB cannot be a system controller.
2075	Error	GPIB cannot clear IFC.
2076	Error	GPIB cannot write.
2077	Error	GPIB cannot call callbacks.
2078	Error	EDVR: System error.
2079	Error	ECIC: function requires GPIB interface to be in CIC mode.
2080	Error	ENOL: GPIB has no listeners.
2081	Error	EADR: incorrect addressing of GPIB interface.

10.3 Error Message

2082	Error	EARG: parameter invalid for this function.
2083	Error	ESAC: not the GPIB interfaced of required system controller.
2084	Error	EABO: I/O operation canceled (timeout).
2085	Error	ENEB: GPIB interface that does not exist. The portion of the trace that meets the specified bandwidth criteria cannot be found.
2086	Error	EDMA: DMA error.
2087	Error	EOIP: Executing asynchronous I/O operation.
2088	Error	ECAP: Unable to operate.
2089	Error	EFSO: File system error.
2090	Error	EBUS: GPIB bus error.
2091	Error	ESTB: Serial polling status byte queue overflow.
2092	Error	ESRQ: query request signal blocked.
2093	Error	ETAB: table error.
2094	Error	GPIB reading error.
2095	Error	Query interrupted.
2120	Error	COM bound later can't call this method.
2123	Error	Bit weight number out of bound.
2124	Error	File extension inconsistent.
2126	Error	File missing an extension.
2127	Error	Folder or file does not exist.
2128	Error	Specified file not found.
2129	Error	WINNT exception appears in automatic control layer.
2130	Error	Invalid port.
2131	Error	Printer not found.
2132	Error	Manual trigger ignored.
2133	Error	Setting trigger failed.
2134	Error	Macro execution error.
2135	Error	Macro definition incomplete.

2136	Error	Trigger too fast, excessive triggers ignored.
2137	Error	Incorrect data block length. Refer to relevant sections on how to get data from a network analyzer.
2139	Error	Requested data not found.
2142	Error	Parameters provided are out of bound and will be limited to the available range before application.
2144	Error	Request failed, no license found.
2145	Error	hresult<x> returned when remote calling with front panel function. This indicates a possible problem with the front panel.
2146	Error	Data out of bound:
2147	Error	Reserve.
2148	Error	Reserve.
2149	Error	Reserve.
2150	Error	<x> out of range!
2152	Error	Front panel <x>.
2153	Error	Front panel information
2154	Error	Power Service <x> There are multiple power services programs running and there should be only one service routine. This problem is generally caused by running an installer, especially the CPU board upgrade.
2155	Error	Power Service <x>
2156	Error	GPIB driver cannot be loaded or unloaded.
2157	Error	GPIB driver cannot be loaded and unloaded.
2158	Error	GPIB driver already loaded but not working properly.
2163	Error	Network analyzer having performed a reset function.
2164	Error	Access to the file denied. This means that the system cannot open the output file as if it were write protected.
2165	Prompt	The file type is structured storage.
2166	Error	Trigger operation failed.
2167	Error	Parameter out of range!

10.4 Method to Obtain After-sales Services

2168	Prompt	File <x> saved.
------	--------	------------------------------

10.4 Method to Obtain After-sales Services

10.4.1 Contact Us

In case of any failure to the 3657 series vector network analyzer, check and save the error message, analyze possible causes, and refer to the methods provided in “[10.2 Troubleshooting and debugging](#)” for preliminary troubleshooting. If the problem cannot be solved, contact the service and consultation center of the Company as per the contact information provided below and provide us with the error collected. We will coordinate with you to solve the problem as soon as possible.

Contact information:

Free customer service number: 800-868-7041

Technical support: **0532-86889847 86897262**

Fax: **0532-86889056 86897258**

Website: www.ceyear.com

Email: **techbb@ceyear.com**

Postal code: **266555**

Address: **No. 98, Xiangjiang Road, Qingdao Economic & Technological Development Zone, Shandong Province**

10.4.2 Package and Mailing

In case of any failure to the network analyzer that is difficult to be eliminated, contact us by phone or fax. If it is confirmed that the network analyzer has to be returned for repairing, pack it with the original packing materials and case by following the steps below:

- 1) Prepare a detailed description of the failure of network analyzer and put it into the package along with it.
- 2) Pack the network analyzer with the original packing materials, so as to minimize possible damage.
- 3) Place cushions at the four corners of the outer packing carton, and place the instrument in the outer packing carton.
- 4) Seal the opening of the packing carton with adhesive tape and reinforce the packing carton with nylon tape.
- 5) Specify text like “Fragile! Do not touch! Handel with care!” and so on.
- 6) Please check by precision instrument.
- 7) Keep a copy of all shipping documents.

Notice

Precautions when packing the network analyzer

Using other materials for packing the network analyzer may damage the instrument. Never use polystyrene beads as packing materials because on the one hand, they cannot provide sufficient protection on the instrument, and on the other hand, they can be sucked in to the instrument fan by the static electricity generated, resulting in instrument damage.

In addition, the network analyzer is heavy. To avoid damage during transportation, it is better to use the original packaging material and package it in two layers! If self-packing is necessary, please ensure that the network analyzer is not damaged during transportation.

Tips

Instrument package and transportation

Please follow carefully the precautions described in “3.1.1.1 Unpacking” when transporting or handling the instrument.

Annex

- [Appendix 1 Introduction to Typical Measurements.....383](#)
- [Appendix 2 Time Domain Measurement.....388](#)

Appendix 1 Introduction to Typical Measurements

- [After 30 Minutes of Preheating, Reset the Analyzer.....383](#)
- [Set Frequency and Power.....383](#)
- [Select a Measurement and Create a Trace.....384](#)
- [Calibration.....385](#)
- [Connect the DUT386](#)
- [Adjust Scale and Data Analysis.....386](#)
- [Record or Save Data.....387](#)

The 3657 series vector network analyzer features various options and complicated measurement, so certain skills and practical experience are required for a proper operation. In order for you to quickly get familiar with the vector network analyzer measurement process, the following takes the attenuator measurement as an example to introduce the measurement process and related settings of common two-port devices. *Refer to Annex5-Amplifier Gain Compression Measurement for amplifier gain compression measurement, and Annex 6-Mixer Measurement for mixer measurement.*

Preparation before measurement: one 3657 series vector network analyzer, a pair of 3.5mm test cables, a set of calibration kits (31121), DUT (attenuator), and adapters (if necessary).

1.1 After 30 Minutes of Preheating, Reset the Analyzer

1.2 Set Frequency and Power

For more information, see "4.3 Set Frequency Range" and "4.4 Set Signal Power Level".

Press [Frequency] button and [Power] button, click the item to be modified in the auxiliary menubar, and then enter the new value through the input keypad on the front panel to complete the setting.

Tips

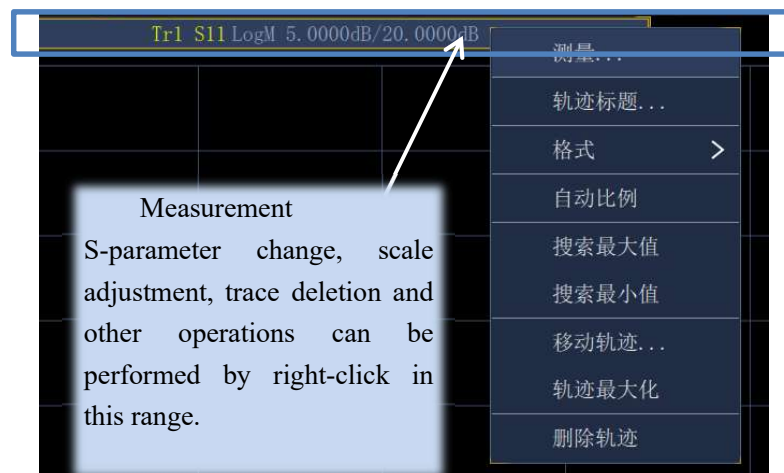
Set Frequency and Power

- 1) The default frequency range of system is 9kHz ~ 9GHz, and the power level is -5dBm.
- 2) The frequency range is to be set according to the measurement needs, and the power level is to be set to the power at the output port of the stimulus source. In principle, as long as the power of the output port of the DUT is not greater than +10dBm, the measurement will not be affected when the power level is changed.

1.3 Select a Measurement and Create a Trace

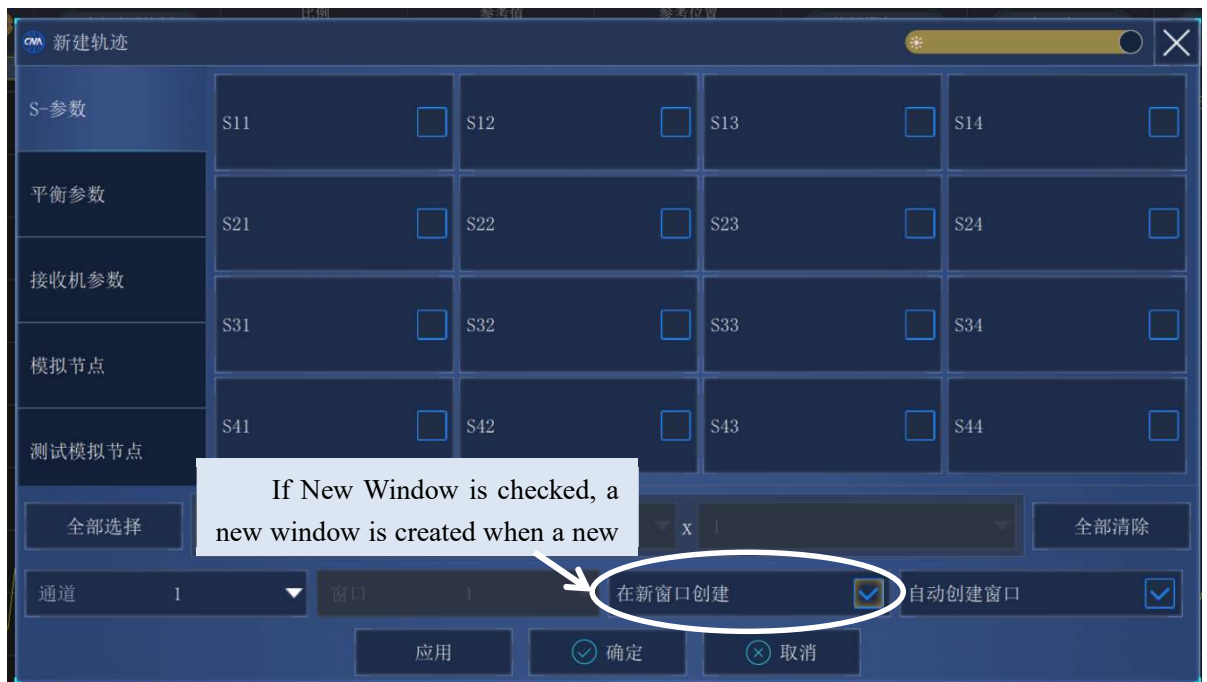
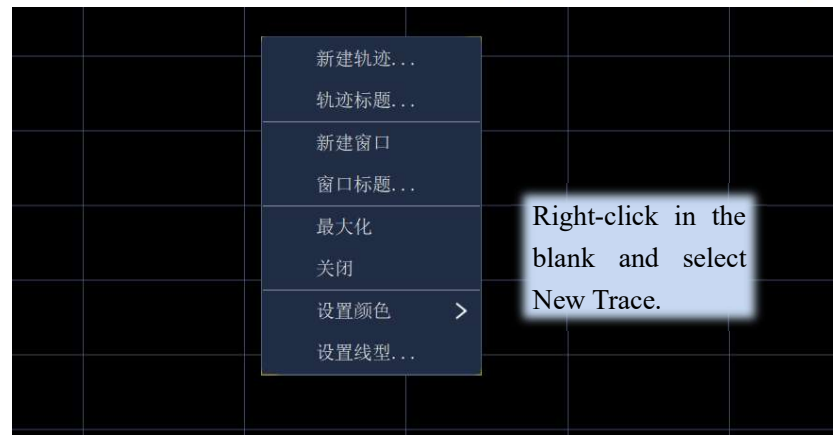
For more information, see ["4.2 Select Measurement Parameters"](#).

To change the S-parameter of the current trace, right click Meas in the titlebar. Check the S-parameter to be measured in the dialog box, and click OK.



To add a measurement trace, right-click in the blank and select New Trace. Select the S-parameter to be added, and click OK.

Appendix 1 Introduction to Typical Measurements



1.4 Calibration

For more information, see "[7 Calibration](#)".

Click **[Cal]**→**[Calibration...]** in the menubar.

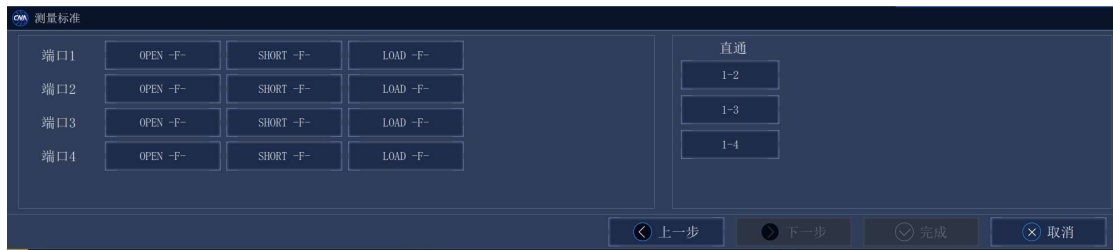
Select between 2-port/4-port calibration, select the DUT connection type, and the calibration kit to be used, and click Next.



Connect the calibration kit, and then click Measure. The system will automatically proceed to

Appendix 1 Introduction to Typical Measurements

the next step after this calibration is completed.



Click [Finish] after all calibrations have been completed.

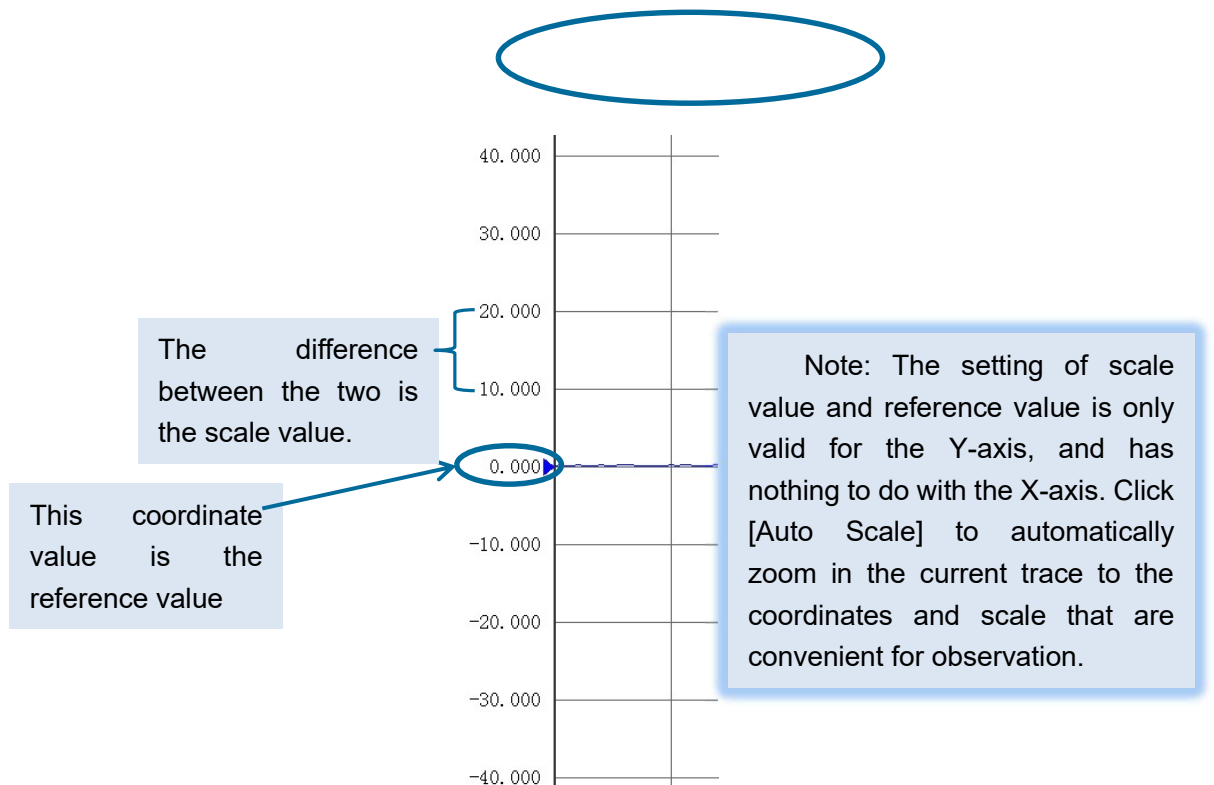
1.5 Connect the DUT

Connect the DUT with a cable.

1.6 Adjust Scale and Data Analysis

For more information, see "4.7 Set Data Format and Scale".

If necessary, click [Response] in the menubar and click [Scale] to carry out corresponding settings.

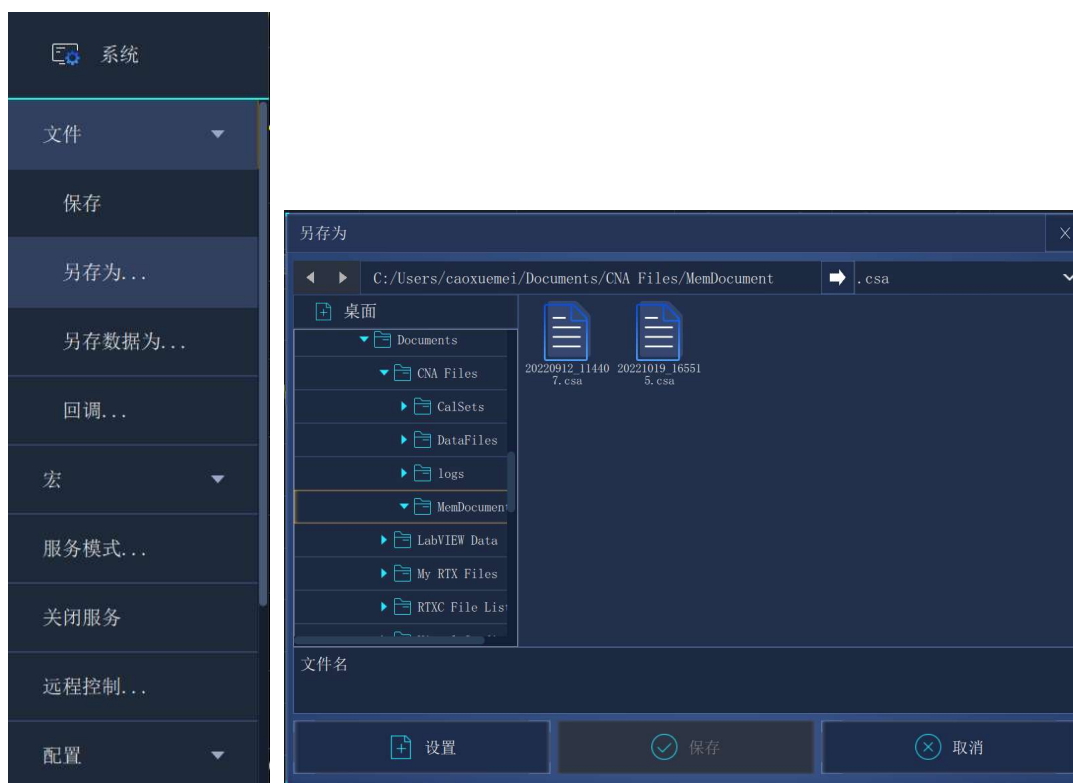


Click [Mrk] in the menubar to turn on the cursor, and turn the adjustment knob to adjust the frequency corresponding to the cursor. The cursor reading is displayed in the upper right corner of the screen.

1.7 Record or Save Data

For more information, see "3.6 Data Output".

Menu path: Click right menubar → [Save As], select the save path and save type, and click Save.



Description:

- 1) The state file and the calibration file can be recalled by the main program, but cannot be edited and processed by other programs.
- 2) The data file can be exported and printed, and edited and processed by other software, but cannot be recalled.
- 3) Graphics files can be exported and printed, but cannot be recalled. The default save path for the file is "C:\Program Files\ceyear\Network Analyzer\MemoryDocuments", and users can set the save path according to their needs.

Appendix 2 Time domain measurement

● Time Domain Measurement Principle.....	388
● Time Domain Measurement Resolution and Range.....	390
● Window Filtering.....	395
● Time Gate Filtering.....	397
● Time Domain Measurement Data.....	402
● Bandpass and Lowpass Time Domain Modes.....	408
● Settings of Time Domain Conversion Measurement.....	411

2.1 Time Domain Measurement Principle

If the analyzer has a time domain option, the response of the device in the time domain can be observed.

In a typical measurement, the analyzer displays the response of the DUT with frequency, called a frequency domain measurement. In contrast, when making time-domain measurements, the analyzer takes the frequency-domain data and performs an inverse Fourier change to obtain time-domain data, and the measurement results are displayed with time as the x-axis. The response values appear at discrete time points, and the characteristics or limitations of the DUT can be analyzed. The following figure shows the frequency and time domain reflection measurements for the same cable, and this cable has two bends, each of which causes a mismatch in the transmission line or a change in impedance.

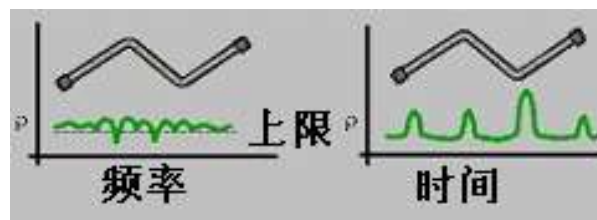


Figure 2.1 Measurement of frequency and time domains

- 1) The frequency domain response of the S_{11} measured at the input port shows a combined reflection response due to cable mismatch wave interactions, but it is difficult to determine the exact physical location at which the cable mismatch occurs.
- 2) The time domain response shows the location and magnitude of each mismatch. From the response we can see that there is a significant mismatch in the second bend of the cable.

In the time domain, the analyzer can use a gate function to filter out unwanted responses and then convert the time data into frequency data. This feature allows you to measure the response of a specific signal in the network, removing the influence of external devices such as connectors or adapters.

Appendix 2 Time domain measurement

Time domain measurement function simulates a traditional time domain reflectometer (TDR): The time domain reflexometer emits an impulse or step signal to the kit under test, and then observe the energy of the reflected signal. By analyzing the amplitude, duration and waveform of the reflected signal, the impedance change of the kit under test can be determined. The network analyzer does not generate incident impulse or step signal, but conducts frequency sweep measurement. Then, it will calculate the time domain information from the frequency domain measurement result by the Fourier algorithm. So far, most common network analyzers use S_{11} ratio measurements for time domain conversion measurements. The S_{11} reflection measurement is not to simply show the reflected signal received by the A or B receiver, but shows the ratio measurement between the measurement receiver and reference receiver. In addition, S_{11} ratio measurement can be used to remove system errors in the calibration. This is especially important for time domain measurements because the measurement reference plane is established by calibration, and the calibration point becomes the zero point of the X time axis, and all time and distance data will use this point as the reference point. In that case, time and amplitude data are very accurate due to calibration. The time domain measurement of the analyzer is done through following steps:

- 1) Collect raw receiver (A and R) data.
- 2) Conduct ratio operation.
- 3) Perform calibration error correction.
- 4) Transform the frequency data into the time domain
- 5) Display measurement results.

Appendix 2 Time domain measurement**2.2 Time Domain Measurement Resolution and Range**

This Section discusses how to observe all effective time domain data of the DUT and how to set it up for the highest resolution and maximum measurement range.

- [Response Resolution.....390](#)
- [Display Resolution.....392](#)
- [Measuring Range.....393](#)

2.2.1 Response Resolution

- [Concept of Response Resolution.....390](#)
- [Factors Affecting the Time Domain Response Resolution.....390](#)
- [Tips for Optimizing Response Resolution.....392](#)

2.2.1.1 Concept of Response Resolution

The analyzer's time domain response resolution refers to its ability to distinguish between two adjacent responses. For equal amplitude response, it is equal to the pulse width of the impulse response defined by the 50% (6dB) amplitude point, or the rise time of step response defined by the 10%~90% amplitude point, as shown in Figure 2.2:

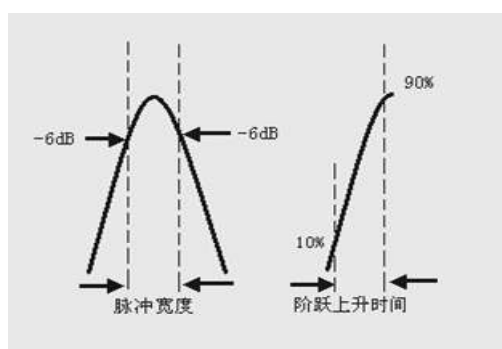
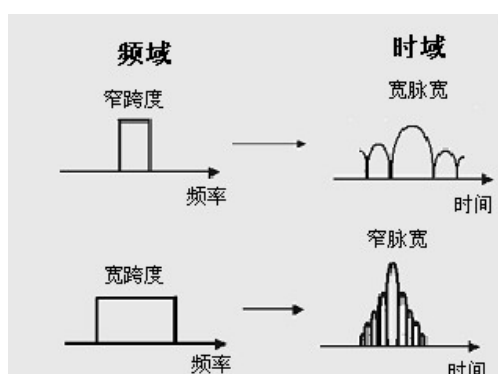


Figure 2.2 Time domain response resolution

2.2.1.2 Factors Affecting the Time Domain Response Resolution**1) Frequency span**

Figure 2.3 shows the effect of the frequency span on the response resolution:



Appendix 2 Time domain measurement

Figure 2.3 Effect of the frequency span on the response resolution

- a) The response measured at a narrow frequency span shows that the impulse response pulses that should be separated independently overlap each other.
- b) The analyzer can distinguish between different response pulses when performing measurements over a wide frequency span.
- c) The frequency span is inversely proportional to the pulse width. The wider the frequency span, the narrower the impulse response pulse, and the shorter the step response rise time.

2) Window width (β parameter)

- a) The response resolution is also a function of the width of the time domain conversion window.
- b) Approximate response resolution can be calculated using the following formulas, which apply only to constant response and calculate 50% pulse width or 10%-90% step rise time.

Lowpass@Step

$$\text{响应分辨率} = \frac{0.45}{\text{频率跨度}} \times \begin{cases} 1.0 & \text{Minimum window} \\ 2.2 & \text{Standard window} \end{cases}$$

Lowpass impulse

$$\text{响应分辨率} = \frac{0.6}{\text{频率跨度}} \times \begin{cases} 1.0 & \text{Minimum window} \\ 1.6 & \text{Standard window} \\ 2.3 & \text{Maximum window} \end{cases}$$

Bandpass impulse

$$\text{响应分辨率} = \frac{1.2}{\text{频率跨度}} \times \begin{cases} 1.0 & \text{Minimum window} \\ 1.6 & \text{Standard windows} \end{cases}$$

The selection of window width depends on the response of the DUT. If the response is a signal with equal level, the minimum window is used to improve the resolution of the measurement; if the response is a signal with unequal level, the maximum window is used to improve the dynamic range of the measurement.

Tips**Pulse width is calculated during time domain conversion measurement**

In the actual time domain conversion measurement, there is no need to manually calculate the pulse width. In the time domain conversion dialog box, the analyzer will automatically calculate the pulse width according to the current setting.

3) Time domain transformation mode

The resolution of the measured response is different in lowpass and bandpass transform modes. For the same frequency span and sweep points, the lowpass mode has a higher resolution. Compared with the bandpass mode, the pulse width can be reduced by half. The following figure shows the comparison of the resolution of the two modes:

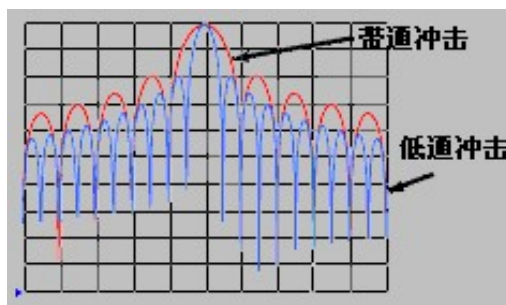


Figure 2.4 Influence of transform mode on resolution

2.2.1.3 Tips for Optimizing Response Resolution

- 1) In most cases, the maximum frequency span is selected for the highest time resolution, but the frequency settings must meet the operating frequency requirements of the DUT.
- 2) Resolution is influenced by frequency span window width and time domain mode.

2.2.2 Display Resolution

Time domain display resolution refers to the ability to accurately locate a signal response in time. A good display resolution can improve the positioning accuracy of the peak point and zero point of the response. The display resolution is affected by time span and sweep points:

$$\text{Display resolution} = \text{Time span} / (\text{sweep points} - 1)$$

The display resolution can be adjusted in the following two ways:

- 1) Reduce the time span
- 2) Increase the number of sweep points

Tips

Changing measurement points may reduce calibration accuracy.

Figure 2.5 shows the measurement results of the same DUT under different time spans:

- 1) A resolution of 50 picoseconds can be achieved at a time span of 10 nanoseconds;

Appendix 2 Time domain measurement

- 2) A resolution of 12.5 picoseconds can be achieved over a time span of 2.5 nanoseconds.

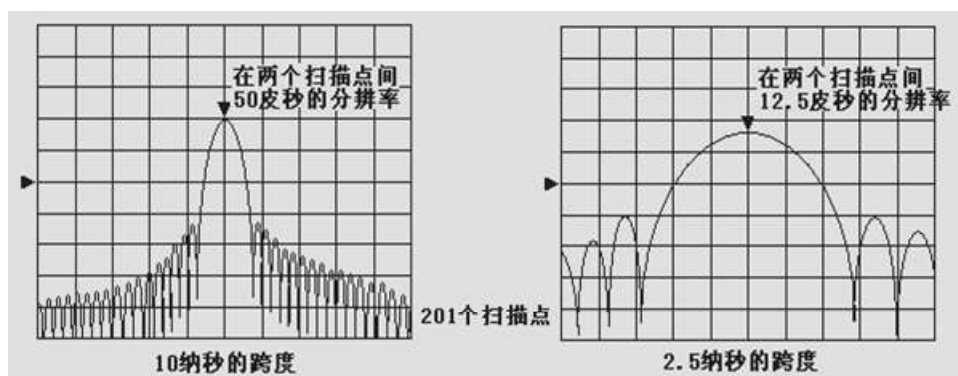


Figure 2.5 Resolution display

Tips

- 1) Changing the time span does not affect the analyzer's ability to distinguish between two signals near each other;
- 2) Changing the sweep points in lowpass mode will change the frequency span and affect the resolution of the response to a certain extent.

2.2.3 Measurement Range

In time domain measurements, the measurement range is defined as the maximum length of time that can be set, and no repeated response occurs when measurements are taken over this length of time. The measurement range is inversely proportional to the response resolution. If one is increased, the other is reduced.

The time domain waveform is a periodic signal that repeats over time, so a repetitive response occurs, as shown in figure 2.6. The repeated response (false response) is not the true response of the kit under test, which only occurs at specific time intervals ($1/\text{sweep point frequency interval}$), so the measurement range is determined also by the frequency interval ΔF of the sweep point:

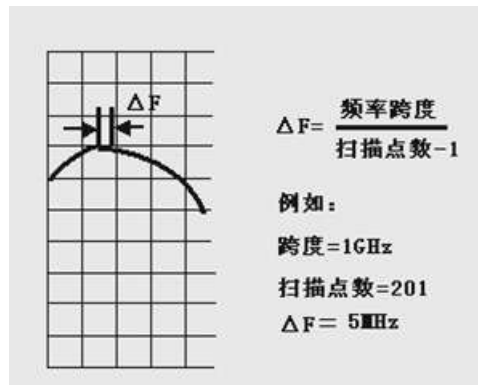


Figure 2.6 Definition of the sweep point frequency space

In the time domain measurements, the maximum measurable stop time is: $1/\Delta F$. The measurement range is proportional to the sweep points -1, which is inversely proportional to the span. To improve the measurement range, the following two settings can be modified:

- 1) Increase the number of sweep points
- 2) Decrease the frequency span.

Notice

The above modifications must be made prior to calibration.

The measurement range in meters is obtained by dividing the speed of light in vacuum by the frequency interval ΔF . To calculate the physical length of the actual measurement range, multiply this by the relative rate of the transmission medium, some of which are as follows:

- 1) $V_{PE} = 0.66$
- 2) $V_{PTFE} = 0.70$

$$\text{Physical length of the actual measurement range} = \frac{V}{\Delta F} \times 3 \times 10^8 \text{ m/s}$$

2.3 Windows Filtering

Due to the influence of sidelobe and pulse width, time domain conversion has certain limitations in time resolution. Window filtering can be used to reduce the influence of these factors. The analyzer is provided with a window function in the time domain conversion, which can more effectively distinguish between various responses. It can change the width of the response pulse, the sidelobe level and the rise time of the step response.

- [Window filtering.....395](#)
- [Set Window Values Correctly.....396](#)
- [Window Properties.....396](#)

2.3.1 Windows Filtering

2.3.1.1 Windows Filtering Concept

There are abrupt changes at the start and stop frequencies in the frequency domain measurement, which will cause overshoot and ringing phenomena in the step response in the time domain. Using window filtering function can reduce the impact of abrupt changes in the frequency domain, but there are also the following shortcomings in the VNA:

- 1) Pulse width of impact response or rise time of step response: This is caused by the band-limited nature of the system itself, which limits the analyzer's ability to distinguish between two response signals adjacent to each other. The pulse width is inversely proportional to the measured frequency span, and the frequency span should be increased to reduce the pulse width.
- 2) Impact response sidelobe: This is caused by the mutation at the termination frequency. The sidelobe adjacent to the high level response will hide the low level response signal, limiting the dynamic range of time domain measurement. The sidelobe can be suppressed by window filtering, as shown in Figure 2.7:

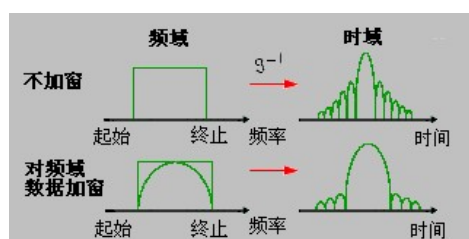


Figure 2.7 Using window filter to suppress side lobes

2.3.1.2 Advantages of Window Filtering

- 1) Narrow window is used to reduce the width of impulse response and improve the response resolution, which can better distinguish the two adjacent responses.
- 2) A wide window is used to reduce the sidelobe level of impact response and improve the dynamic range, which can better measure the low level response.

Appendix 2 Time domain measurement

2.3.1.3 Effects of Window Filtering

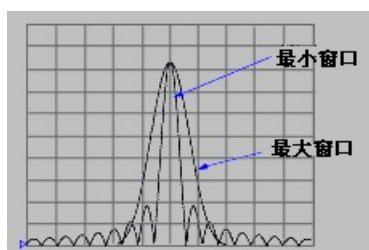


Figure 2.8 Influence of window on response signal

As shown in Figure 2.8, the analyzer performs window filtering on the measured data in the frequency domain, which has the following influences on the time domain measurement response of the DUT:

- 1) In the impact transform mode, the sidelobe level and pulse width are affected, and the measurement resolution is changed.
- 2) In the Lowpass@Step transform mode, overshoot and ringing can be reduced.

2.3.2 Set the Window Values Correctly

As shown in Figure 2.9, the appropriate window value must be selected according to the response type of the DUT:

- 1) The DUT has the same amplitude response: Small window is selected to obtain narrow pulse width and improve the resolution of time domain measurement.
- 2) The DUT has different amplitude response: A large window is selected to obtain a low sidelobe level and improve the dynamic range of time domain measurement.

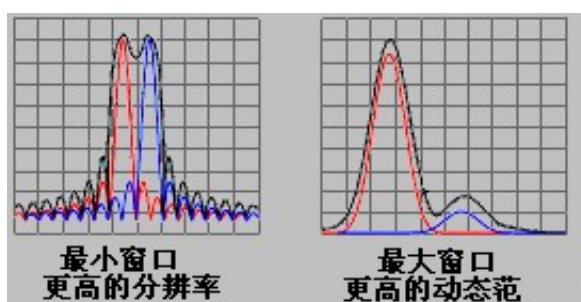


Figure 2.9 Correct Setting Window

2.3.3 Window Properties

In the time domain measurement, the sidelobe level is related to the selected window, and the response resolution is related to the frequency span and the selected window. Figure 2.1 gives the window property index at a frequency span of 2.997GHz:

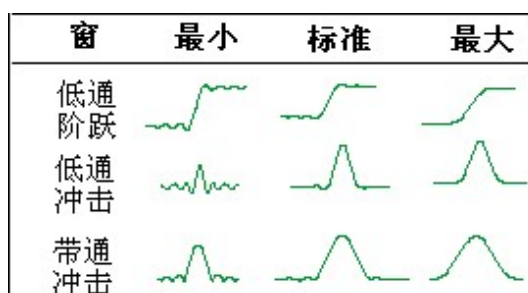
Figure 2.1 Window property index @ 2.997GHz

Conversion mode	Window	Response Resolution	Sidelobe
-----------------	--------	---------------------	----------

Appendix 2 Time domain measurement

		Step rise time	Imp Width	level
Lowpass@Step	Min	150ps		-21dBc
	Standard	330ps		-60dBc
	Max	494ps		<-70dBc
Lowpass impulse	Min		200ps	-13dBc
	Standard		320ps	-44dBc
	Max		481ps	<-75dBc
Bandpass impulse	Min		400ps	-13dBc
	Standard		641ps	-44dBc
	Max		941ps	<-75dBc

Table 2.2 Time-domain waveform of windowing for short reflection measurement



2.4 Time Gate Filtering

Using the time gate function, you can select or remove a response in the time domain, and then the analyzer can transform the processed time domain data to the frequency domain for observation. In other words, after time gate filtering is performed on time domain data, the time domain conversion function can be turned off, and the frequency domain response of the DUT can be observed under the condition that the time gate function is effective. For example, a time gate can be used to remove the influence of multipath transmission during transmission measurement, or only one single time domain response signal can be observed, and the impact of each single response on the measurement result can be analyzed in the frequency domain.

- [How to Use Gate Function.....397](#)
- [Gate Settings.....398](#)
- [Gate Shape.....398](#)

2.4.1 Method to Use the Gate Function

In actual measurement, the steps of time gate measurement are as follows:

- 1) The DUT is measured in the frequency domain.

Appendix 2 Time domain measurement

- 2) Enable the time domain conversion function, and the analyzer calculates the time domain response of the DUT.
- 3) Select the shape of the gate:
 - a) **Bandpass**: Place the gate at the center of the response you want to keep, enable the gate function, and the response outside the gate is mathematically removed from the displayed measurement.
 - b) **Bandstop**: Place the gate at the center of the response you want to remove, enable the gate function, and the response inside the gate is mathematically removed from the displayed measurement.
- 4) Disable the time domain conversion function, observe the frequency domain response of the DUT, and understand the influence of the response of the removal on the measurement result.

Table 2.10 shows a measurement application of a bandpass time gate:

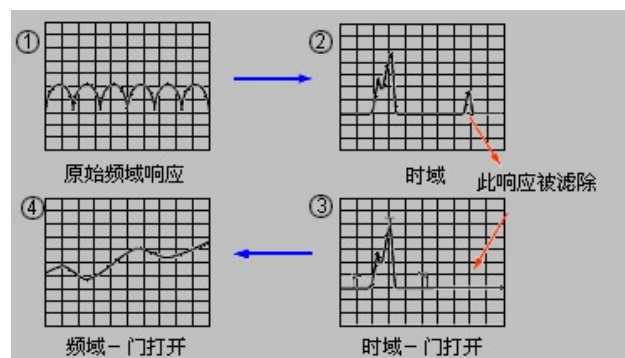


Figure 2.10 Measurement application of bandpass time gate

2.4.2 Door Settings

The boundary and function of a gate can be defined with the following settings:

- 1) **Start and stop**: define the -6dB cut-off time for the gate.
- 2) **Center**: define the center time of the gate.
- 3) **Span**: it's the end time minus the start time.
- 4) **Gate type**:
 - a) **Bandpass**: filter out the response outside the gate.
 - b) **Bandstop**: filter out the response inside the gate.

2.4.3 Gate Shape

2.4.3.1 Gate Shape

Gate and window filters also have filter shapes, and measurements can be optimized by selecting reasonable filter shapes (minimum, standard, width, and maximum):

- 1) **Minimum gate shape**: The sidelobe level of the filter is the highest, the edge drop is the steepest and the passband ripple is the largest.
- 2) **Maximum gate shape**: the sidelobe level of the filter is the lowest, the edge drop

is the slowest, and the passband ripple is the smallest.

2.4.3.2 Passband Ripple

A gate is a bandpass (or bandstop)-shaped filter. Each filter of different shapes has different filter characteristics. The following figure shows the passband ripple of four filters of different shapes with a ratio of 0.5dB/ division. Figure 2.11 shows that the smallest gate has the largest ripple:

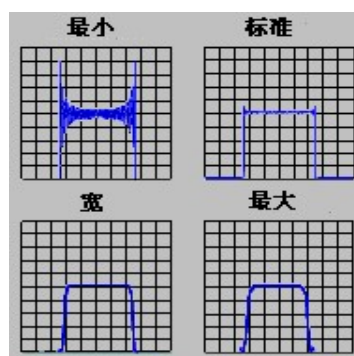


Figure 2.11 Passband ripple of different gate filters

2.4.3.3 Sidelobe Level

As shown in Figure 2.12, four filters of different shapes have different sidelobe levels. In actual measurement, a balance must be achieved between a lower sidelobe level and a faster cutoff rate:

- 1) The smallest gate has the highest sidelobe level and the fastest cutoff speed, which is very suitable for filtering out unwanted responses near the measured response.
- 2) The sidelobe level of the largest gate is the lowest, the cutoff rate is the slowest, the pass band of the gate is the widest, and there is the greatest attenuation outside the gate band.

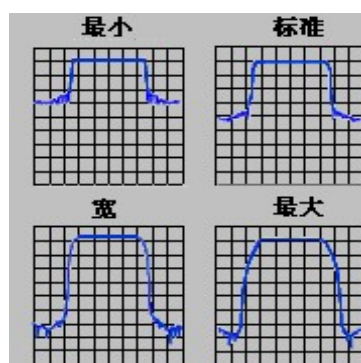


Figure 2.12 Sidelobe levels of different gates

Appendix 2 Time domain measurement

2.4.3.5 Gate Characteristics

Gates of different shapes have different characteristics. The characteristic indicators of different gates are given in Table 2.3. The characteristics of gates are defined by the following indicators:

- 1) Passband ripple and sidelobe level: Describe the gate shape.
- 2) **Cutoff time:** The time between the stop time of the gate (at -6dB of the filter edge) and the peak point of the first sidelobe.
- 3) Minimum gate span: Twice the cutoff time.

Table 2.3 Characteristics of different gates

Gate Shape	Passband ripple	Sidelobe level	Deadline	Minimum gate span
Min	$\pm 0.10\text{dB}$	-48 dB	1.4/frequency span	2.8/frequency span
Standard	$\pm 0.01\text{dB}$	-68 dB	2.8/frequency span	5.6/frequency span
Wide	$\pm 0.01\text{dB}$	-57 dB	4.4/frequency span	8.8/frequency span
Max	$\pm 0.01\text{dB}$	-70 dB	12.7/frequency span	25.4/frequency span

Table 2.13 gives the shape of a complete gate and the meaning of its characteristic index:

- 1) T_1 is the span of the gate, which is equal to the end time minus the start time.
- 2) T_2 is the time between passband edge and -6dB point, representing the cutoff rate of the filter.
- 3) T_3 is the time between the -6dB point and the gate bandstop edge.

For gates of all shapes, T_2 and T_3 are equal, and both sides of the filter center point are completely symmetrical.

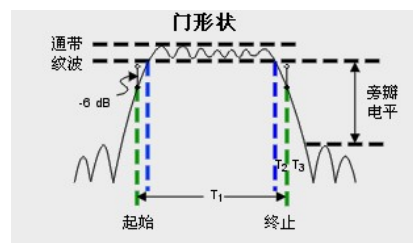


Figure 2.13 Gate shape

2.4.3.6 Minimum Gate Span

Each shape of the gate has a recommended minimum gate span for proper operation, determined by the finite cutoff rate of the gate. This minimum gate span is given by the

Appendix 2 Time domain measurement

following formula, where the passband of the filter is 0.

$$T_{1\text{MIN}}=2\times T_2$$

If the gate span is set to be smaller than this minimum, the analyzer will produce the following effects:

- 1) Gate shape is distorted without passband;
- 2) Gate shape is distorted;
- 3) Indication of start and stop times are not corrected;
- 4) It may increase the sidelobe level.

Appendix 2 Time domain measurement**2.5 Time Domain Measurement Data**

To help the user understand the response of device in the time domain, this section gives the examples of measurements.

● Masking.....	402
● Reflection Measurement in Bandpass Mode.....	403
● Transmission Measurement in Bandpass Mode.....	404
● Error Location in Lowpass Mode.....	405
● Reflection Measurement in Lowpass Mode.....	406
● Transmission Measurement in Lowpass Mode.....	407

2.5.1 Masking**2.5.1.1 Role of Masking**

As masking affects the response observed by the user during time domain measurement, it helps the user to understand the measurement data of time domain.

- 1) Masking occurs when a breakpoint or a loss of a closest reference channel occurs, which will affect all concurrent breakpoint responses.
- 2) The energy reflected or absorbed from the first breakpoint cannot be transmitted to the second or subsequent breakpoint.
- 3) The effect is that each breakpoint response is smaller than the first response which is upcoming but not visible.

2.5.1.2 Masking Due to Energy Reflection

Masking caused by reflection occurs in case of any major error configuration. This phenomenon is illustrated in the linear impedance example below, as shown in Figure 2.14.

- 1) A 50Ω line is connected to a 25Ω air line, and then connected to a 50Ω transmission line.
- 2) The first breakpoint may incur a reflection with a coefficient of 0.333 (exactly equivalent to a 25Ω resistance).
- 3) After the 25Ω end, the response is no longer a reflection with a coefficient of 0.333 (but a reflection equivalent to a 50Ω resistance). Through display of two Δ markers, the difference between them in this order of magnitude is 35.355mU. The reason is that the point when the second pulse occurs shall have a smaller attenuation than the unit of energy reflection at the first detuning point.

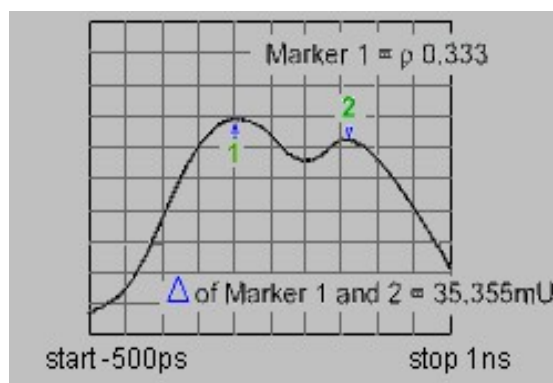


Figure 2.14 Influence of masking caused by energy reflection on measurement

2.5.1.3 Masking Due to Energy Absorption

Masking due to energy absorption refers to masking caused when a circuit with loss is measured. Figure 2.15 shows the masking of a cable in the open loop.

- 1) MARKER 1 displays a return loss of the open-loop response of -2.445dB (in LOG MAG mode).
- 2) This expected response is typical in open loop.
- 3) This marker value represents the sum of losses of the forward path and reverse path.

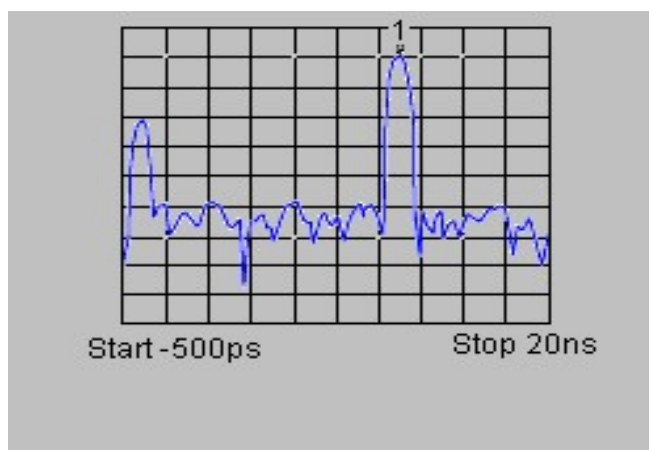


Figure 2.15 Influence of masking caused by energy absorption on measurement

2.5.2 Reflection Measurement in Bandpass Mode

2.5.2.1 Data on the Horizontal Axis

The time taken for a pulse is the time from the test port to the breakpoint and back to the test port.

Appendix 2 Time domain measurement**2.5.2.2 Data on the Vertical Axis**

- 1) If LOG MAG format is used, it refers to return loss (dB).
- 2) In LIN MAG mode, it means the reflection coefficient.

The following example is used to illustrate reflection measurements in bandpass mode, as shown in Figure 2.16.

- 3) There is a pulse at 0 on the timeline that represents the time from the output of the test port to the first discontinuous point.
- 4) A concurrent pulse represents the next breakpoint encountered by the pulse.

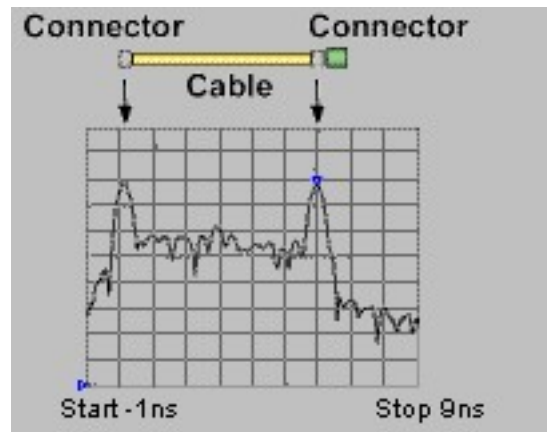


Figure 2.16 Reflection measurement in bandpass mode

2.5.3 Transmission Measurement in Bandpass Mode**2.5.3.1 Data on the Horizontal Axis**

It represents the propagation delay under the test device.

2.5.3.2 Data on the Vertical Axis

- 1) If LOG MAG format is used, it refers to transmission loss or transmission gain.
- 2) In LIN MAG mode, it means the transmission coefficient.

The following example is used to illustrate transmission measurements in bandpass mode, as shown in Figure 2.17.

- 3) This example illustrates the multipath information transmitted in the surface acoustic wave (SAW) filter in this mode.
- 4) The pulse closest to 0 time point describes the transmission time of the network analyzer on the shortest path from the output to the input, which may or may not be the largest pulse or the optimum transmission path.
- 5) Each concurrent pulse may have another transmission path from input to output, which is longer than the shortest path.

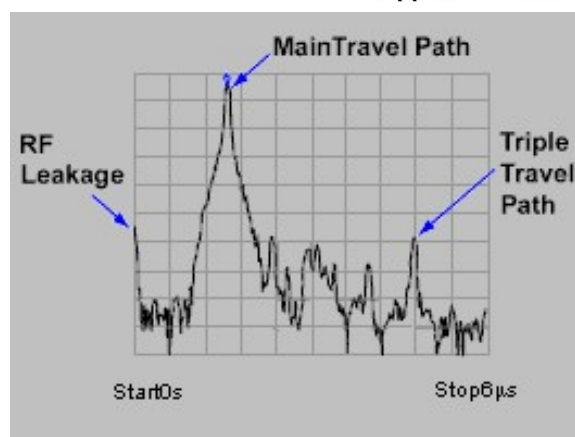


Figure 2.17 Transmission measurement in bandpass mode

Tips

Observe the main path of the SAW filter response

To view the SAW main path of the filter response, it is allowed to set the gates that block all pulses except the main pulse. Then in the frequency domain, only the frequency response on the main path is displayed.

2.5.4 Error Location when Applying Lowpass Mode

The lowpass mode can simulate the time domain reflectometer (TDR) response of the DUT. This response contains information on useful decisions at discontinuities. The figure below the lowpass response for a known breakpoint. All circuit elements pass through the corresponding analog waveform under the lowpass time domain S_{11} response. The lowpass mode allows you to observe either a step response or an impulse response.

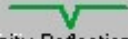
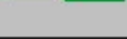
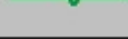
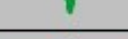
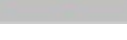
IMPEDANCE	STEP RESPONSE	IMPULSE RESPONSE
OPEN	 Unity Reflection	 Unity Reflection
SHORT	 Unity Reflection = 180	 Unity Reflection = 180
RESISTOR $R > Z_0$		
RESISTOR $R < Z_0$		
INDUCTOR		
CAPACITOR		

Figure 2.18 Lowpass time domain response waveform of discontinuous circuit

Tips

Step response makes it easier to understand the nature of breakpoints. This mode is closer to traditional TDR measurements.

2.5.5 Reflection Measurement in Lowpass Mode

2.5.5.1 Data on the Horizontal Axis

The data on the X-axis refers to the time for the pulse to travel from the test port to the breakpoint and back to the test port.

Note: To confirm the actual physical length, enter the appropriate velocity factor.

2.5.5.2 Data on the Vertical Axis

If the REAL format is used, it refers to the reflection coefficient (ρ). The following example is a reflection measurement of impulse response and step response in LOG MAG format of lowpass mode.

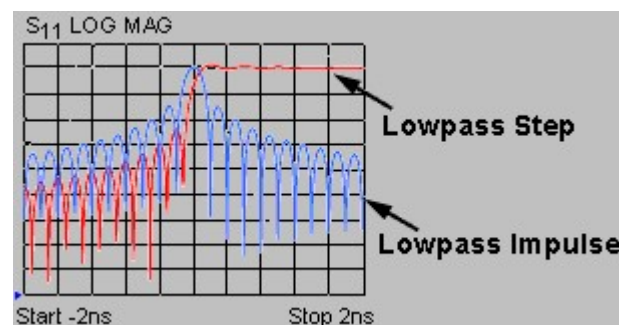


Figure 2.19 Reflection measurement in lowpass mode

The following example illustrates the measurement results of two different cables in REAL format of lowpass mode.

- 1) The lowpass response contains information such as the impedance calibration method where the breakpoint is located.

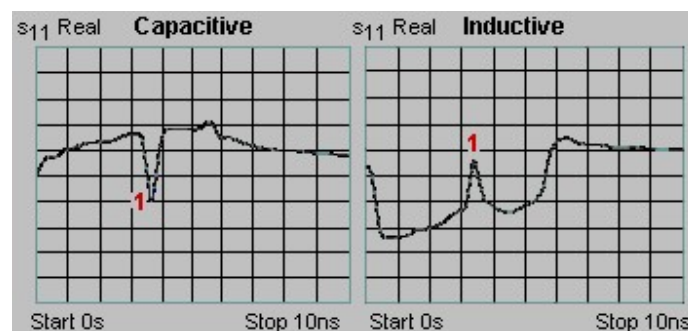


Figure 2.20 Lowpass step measurement results of different cables

Appendix 2 Time domain measurement

- 2) Left measurement-meaning that the response of the crimped cable has a capacitive breakpoint.
- 3) Right measurement-meaning that the response of the worn cable has an inductive breakpoint.

2.5.6 Transmission Measurement in Lowpass Mode

2.5.6.1 Data on the Horizontal Axis

- 1) The average time passing through the DUT in a frequency range.
- 2) The electrical delay in time of a device.

2.5.6.2 Data on the Vertical Axis

- 1) In the case of REAL mode, it is the real unit (such as voltage).
- 2) If LOG MAG (only in pulse mode) format is used, it refers to the transmission loss or gain (dB).

The following example illustrates the lowpass step response of an amplifier.

- 1) The average group delay, which covers the entire measurement frequency range, differs in time between the step response and the amplifier response.
- 2) In the frequency domain sweep, the step time rises to the highest frequency point in a balanced way; the higher the frequency, the faster the rise.
- 3) There is a stop signal in the rise of the amplifier response.

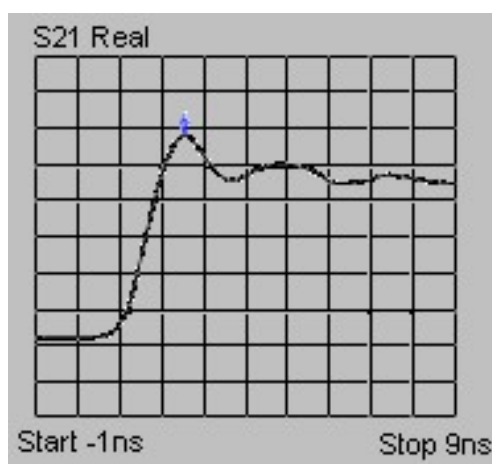


Figure 2.21 Lowpass step measurement results of the amplifier

Tips

Scope of application of Y-axis format

For lowpass step response, the most useful format is REAL (emission coefficient); for pulse responses, the REAL format is also applicable, but the LOG MAG format is recommended to get the best dynamic range for those observed large and small breakpoints.

Appendix 2 Time domain measurement**2.6 Bandpass and lowpass time domain modes**

The measurement must be selected in time domain mode before the user sets the measurement value.

● Mode Comparison.....	408
● Mode Selection Based on Test Device.....	408
● Frequency Range and Data Points.....	409
● Pulse and Step Response.....	409
● Use Data Formats.....	410

2.6.1 Mode Comparison**2.6.1.1 Bandpass Mode**

- 1) Easy applications.
- 2) Only applicable to bandpass devices for which operation to a frequency down to 0Hz is impossible.
- 3) Allow for the measurement of some frequencies with start and stop frequencies already set.
- 4) Only applicable to pulse response measurement as the conversion data is not allowed to include 0Hz.
- 5) Suitable for situations when reflection measurement or transmission measurement is in progress.
- 6) Wrong location allowed.

2.6.1.2 Lowpass Mode

- 1) Applicable to lowpass device for which operation to a frequency down to 0Hz is possible.
- 2) Simulate a conventional time domain reflectometer (TDR).
- 3) Measurement of frequency with a moderating frequency link with the 0Hz to extrapolate the first new data point in the frequency domain.
- 4) Allow for the pulse response measurement and step response measurement as the conversion data includes 0Hz.
- 5) Allow for the wrong location and proof of the type of impedance (capacitive or inductive) present at the breakpoint.
- 6) Suitable for situations when reflection measurement or transmission measurement is in progress.
- 7) Higher response resolution than the bandpass mode at the same frequency span (2 times).

2.6.2 Mode Selection Depends on the Test Device

- 1) Bandpass mode - to be selected when the test device is operating in bandpass,

bandstop, or highpass filter mode.

- 2) Lowpass mode - to be selected when the test device allows the pass-through of all frequency down to 0Hz, especially when the device response has a copy at the low end of the frequency span.

2.6.3 Frequency Range and Data Points

2.6.3.1 Bandpass Mode

- 1) The user can select a number of start and stop frequency values within the measurement range.
- 2) The analyzer will acquire data in the frequency range from start to stop.
- 3) Only pulse response is provided.

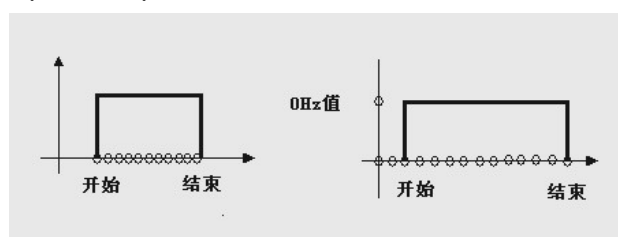


Figure 2.22 Bandpass mode

2.6.3.2 Lowpass Mode

- 1) In the frequency domain, a DC condition that allows for successful inferring from the first few points is required.
- 2) The number of data points is required to be equal to the space.
- 3) The analyzer is set up with the start frequency harmonics for frequency measurement.
- 4) $F_{\text{cutoff}} = N \times F_{\text{start}}$ (N = sweep points) This equation limits the cutoff frequency of N sweep point.

Notice

F-start and F-stop

As the minimum value of F may be 300kHz, F_{cutoff} is $N \times 300\text{kHz}$ at least.

2.6.4 Pulse Response and Step Response

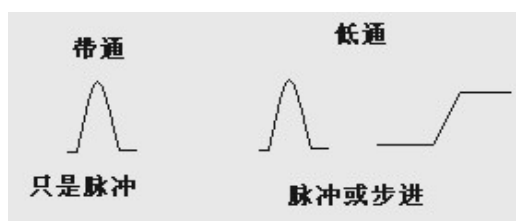


Table 2.23 Lowpass mode

When the DUT involves a stimulus with a pulse function or a step function, the network analyzer will initiate this time domain response.

- 1) **Lowpass impulse** - a voltage waveform from 0 to max voltage and then to 0. The pulse width is determined by the frequency span during frequency domain measurement.
- 2) **Lowpass step** - a voltage waveform from 0 to max. voltage. The step rise time is determined by the highest frequency during frequency domain measurement.

Notice

Scope of application of two lowpass time domain conversion modes

Lowpass step in lowpass mode is the simplest time domain conversion mode to explain breakpoint. The TDR display of general perceptual and capacitive breakpoints can be given in these two modes.

- 3) In the frequency response data, the analyzer calculates the bandpass response for steps, pulses, or settings depending on the selected conversion method .
- 4) The analyzer displays this data in the time domain response.

For more information on the effect of frequency span on pulse width and rise time, please refer to **Response and Scope**.

2.6.5 Use Data Formats

2.6.5.1 Linear MAG

- 1) For reflection measurements, this format shows the average magnitude of the reflection coefficient over the entire measurement frequency range.
- 2) For transmission measurements, this format shows the average transmission coefficient (τ) of the transmission path over the entire measurement frequency range.
- 3) This format is suitable for observing responses of similar magnitude.

2.6.5.2 Log

- 1) For reflection measurements, this format shows the response to return loss (dB) in the unit. The displayed value represents the average return loss of the breakpoint over the measurement frequency range.
- 2) For transmission measurements, this format shows of response to transmission loss or gain (dB) in the unit, and represents the average loss on the transmission path over the measurement frequency range.
- 3) This format is suitable for observing a wide dynamic range of transmission response.

2.6.5.3 VSWR

For reflection measurements, this format shows the average standing wave ratio (SWR) of the breakpoint in the measurement frequency range.

2.6.5.4 Format Available for Lowpass -- REAL

- 1) This format shows the response in Real unit.
- 2) If the DUT involves a response downward to DC, the lowpass mode is to be used.
- 3) If the DUT involves no measurement response on DC, the bandpass mode is to be used.

2.7 Setting of Time Domain Conversion Measurement

Menu path: **[Analy]** → **[Time Domain Conversion]** to display the Time Domain Conversion dialog box.

Click to check **[Time Domain Conversion (ON/OFF)]** to activate the Time Domain Conversion function.

Click [Start], [Stop], or [Center], [Span] to set the time domain measurement range.

In the [Transform Mode] zone, select the transform mode: [Lowpass Impulse], [Lowpass Step] or [Bandpass].

Appendix 2 Time domain measurement

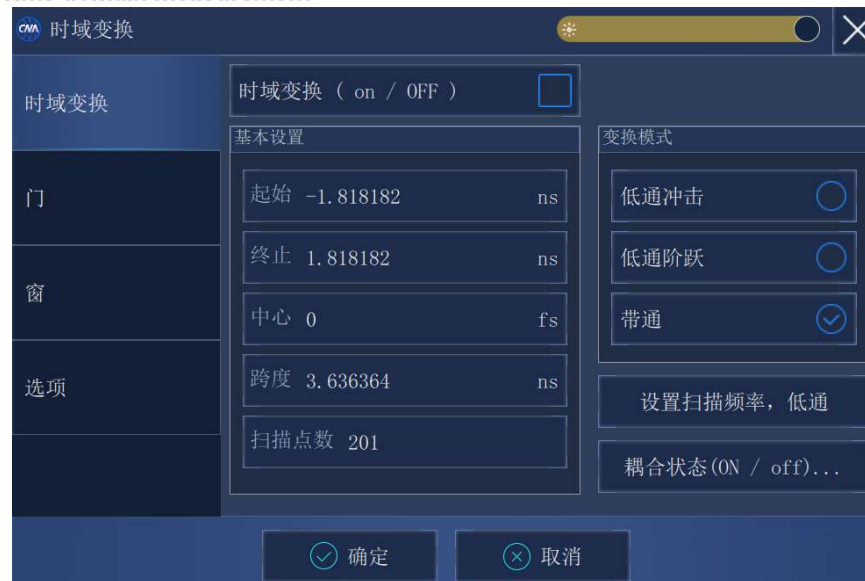


Figure 2.24 Setting the Time Domain Conversion measurement

Set Window function: Click the [Window] option button in the Time Domain Conversion dialog box, and use the following three methods to set window filter:

- Drag the slider with the mouse until the pulse width or sidelobe level meets the requirements;
- Click the [**Kaiser Window β Parameter**] box and adjust the β value setting until the pulse width or sidelobe level meets the requirements;
- Click the [**Pulse Width**] box to set the response pulse width directly.



Figure 2.25. Setting of time domain conversion measurement— Set window

To set the time domain gate filtering function: Click the [Gate] option button in the Transform dialog box, and click to check the [Gate] box to open the time domain gate filtering function.

- Click [Start], [Stop] or [Center], [Span] to set the gate boundary.

Appendix 2 Time domain measurement

- Select the gate type in the [Gate Type] box: bandpass, bandstop;
- In the [Gate Shape] box, select the door shape: Min, Nor, Wide, Max.



Figure 2.26. Setting of time domain conversion measurement - Set door

To observe the frequency domain response of the DUT while the gate function is still in effect, click the [Time Domain Conversion] option button in the Time Domain Conversion dialog box, and click to uncheck the [Time Domain Conversion] check box.

Tips on Time Domain Conversion setting

- To close the Time Domain Conversion dialog box with time domain conversion and gate enabled, you can drag the mouse to change the time gate boundary settings: The start and stop of gates are indicated by symbols “↗” and “↘” respectively, point the marker at the start or stop indicator of the gate, left click and hold down; when the marker pointer turns to horizontal adjustment instruction “↔”, drag the marker to move the gate boundary indicator to the desired position and release the mouse to complete the start or stop setting of the gate.
- After completing the setting of time domain measurement through the Time Domain Conversion dialog box, you can close the Time Domain Conversion dialog box and open the Time Domain toolbar to adjust the setting of time domain measurement, so that you can more clearly observe the impact of setting change on measurement. For details about how to set the Time Domain toolbar, see "Trigger toolbar display" in 4.9 "Setting analyzer display".

2.8 Time Domain Reflectometer (TDR) Impedance Test

With the rapid development of the information industry, increasingly higher demand is posed for the network bandwidth, and the information devices (such as large servers, computers and switches, etc.) are required to carry an increasingly higher data rate.

Appendix 2 Time domain measurement

Information device manufacturers are also paying more attention to the signal integrity in high-speed interconnect channels, and TDR differential impedance is one of the important tests.

Traditionally, TDR differential impedance test is a common method to evaluate transmission lines by using a time domain reflectometer (TDR) oscilloscope. As an alternative time domain analysis method, TDR measurement based on vector network analyzer (VNA) has attracted more and more attention. The test principle of the two methods is shown in the figure below.

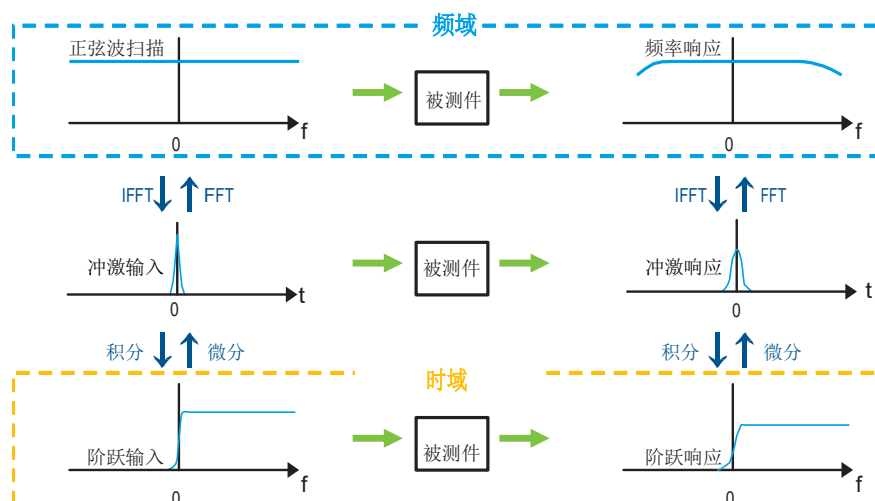


Table 2.27 Comparison of principle of two TDR impedance test methods

2.8.1 Frequency Domain Parameter Setting

According to the setting relationship between time domain measurement resolution and range given in 2.2 of the Annex, set the start frequency and stop frequency following the menu path: **[Stimulus]** → **[Frequency]**, and set the sweep points following the menu path: **[Stimulus]** → **[Sweep]** → **[Sweep Points]**.

2.8.2 Test Trace Settings

Menu path: click **[Trace]** → **[New Trace]** → **[Balanced Parameter...]** to display the **Balanced Parameter Topology** dialog box.

To modify the balanced topology setting, select **[Balanced-Balanced]**, and set the relationship between the balanced port and the network analyzer port according to the connection of the DUT.

Create a trace Sdd11 for differential test.



Figure 2.28 Create a test trace

2.8.3 Time Domain Conversion Settings

Menu path: **[Analy]** → **[Time Domain Conversion...]** to display the **Time Domain Conversion** dialog box.

Click **[Start]**, **[Stop]**, or **[Center]**, **[Span]** to set the time domain measurement range.

Select transform mode in **Transform Mode** zone: **[Lowpass Step]**.

Click **[Set Frequency, Lowpass]** to set the test frequency interval to equally spaced harmonics.



Figure 2.29 Setting of time domain transform mode

Appendix 2 Time domain measurement

2.8.4 Calibration

First, select the calibration type and connection mode according to the calibration kit and the DUT. Then select the calibration kit and the DUT. Then follow the wizard to carry out four-port calibration, and finally click Finish.



Figure 2.30 Calibration process

2.8.5 Selection of Impedance Format

Menu path: click **[Form]** → **[Format...]** to select the impedance format.

Menu path: Click **[Analysis]** → **[Time Domain]** → **[Time Domain Conversion...]** to display the **Time Domain Conversion** dialog box.

Click to check **[Time Domain Conversion]** to activate the Time Domain Conversion function.



Figure 2.31 Time domain reflectometer (TDR) impedance display

Notice

Calibration for time domain measurement

To perform a calibration before a time domain measurement in lowpass mode, it is required to first select the lowpass mode in the Time Domain Measurement dialog box or click **[Set frequency. Lowpass]** button to set the start frequency of the measurement.

Annex 3 Advanced Time Domain Analysis

3.1 Overview

The vector network analyzer advanced time domain analysis option provides the following functions:

- **Wide frequency coverage with a four-port option to enable testing of the latest high-speed data standards**
- **Large dynamic range for testing the true performance of the DUT**
- **Excellent noise base for accurate and repeatable testing**
- **Fast measurement speed for real-time analysis of DUT characteristics**
- **Advanced calibration techniques to reduce measurement errors**
- **Automatic fixture removal technology ensures easy removal of fixture and probe effects on test results**
- **Accurate TDR/TDT and S-parameter data are available simultaneously**
- **Simultaneous time and frequency domain analysis to help find the source of loss, reflection and crosstalk**
- **Single connection for forward and reverse transmission and reflection measurements**
- **Provides test results for all transmission modes (single-ended, differential and mode conversion)**
- **Evaluation of transmission performance of high-speed interconnects through simulated eye diagrams**
- **Provide the simulation code type generator to get PRBS, K28.5, ABS, etc., and support user-defined operation**
- **Built-in predefined high-speed serial standard eye diagram template**
- **No pulse generator is required, and the eye diagram is generated based on frequency domain data**

Annex 3 Advanced Time Domain Analysis

3.1.1 On-screen Display



Figure 3.1 Advanced time domain analysis option display

Graphical area

Display measurement results (such as TDR/TDT and S-parameters) and eye diagrams in the graphical area.

Configuration Wizard

Users can set the test status according to the wizard

Instrument state column**Test mode selection**

You can choose from three advanced time domain analysis modes

- Configuration
- TDR/TDT
- Eye diagram

When one of the tabs is selected, the mode changes. When selected, the mode will be highlighted. The displayed settings area will change based on the selected mode.

Parameter setting

The setting area will change and be displayed according to the selected mode.

3.1.2 Enable and Disable the Advanced Time Domain Analysis Option

Ensure that the advanced time domain analysis option is installed and enabled.

Enable the advanced time domain analysis option

- a) Click [Analysis] > [TDR] to open TDR. Enable the advanced time domain analysis option.
- b) To exit, click the [X] button on the upper right corner of the 3657 Advanced Time Domain Analysis Option application screen to close it.



Figure 3.2 Advanced Time Domain Analysis Menu

3.2 Measurement Settings

3.2.1 Use of the Configuration Wizard

The configuration wizard guides you to perform the advanced time domain analysis measurement settings step by step. The settings will appear automatically when you execute the advanced time domain analysis option for the first time.

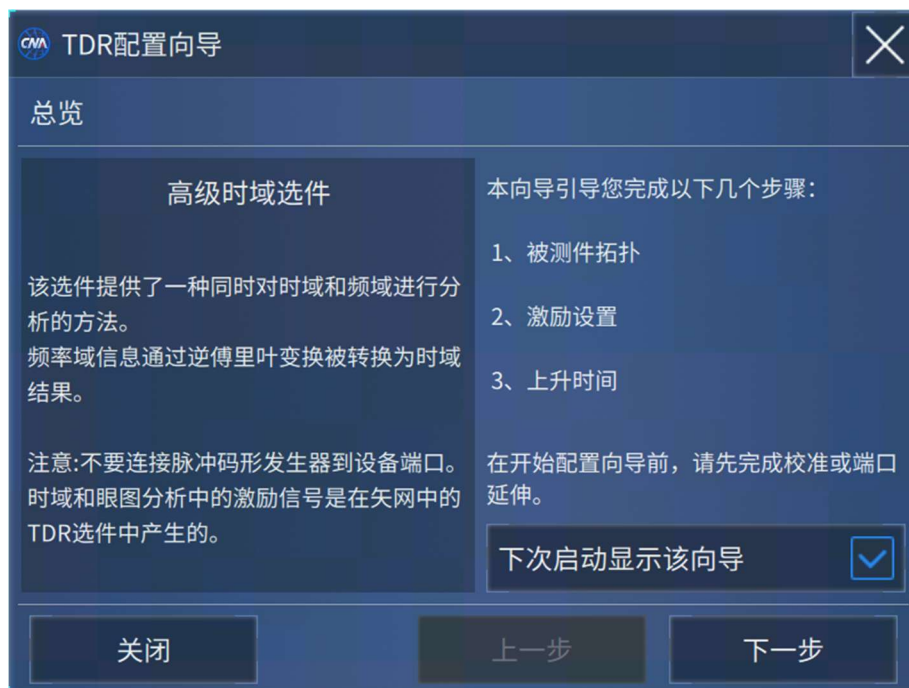


Figure 3.3 TDR configuration wizard

- If you do not want to use the configuration wizard to perform advanced time domain analysis settings, click the **[Close]** button to close the configuration wizard.
- Click **[Next]** → Start the settings process.
- Select the topology of the DUT.



Figure 3.4 Selecting the topology of the DUT

d) Click [Next].

The Stimulus Setting window of the configuration wizard will pop up. In this window, you can set the start/stop frequency, dielectric constant and velocity factor of the channel. You can also check step frequency and length of the DUT.



Figure 3.5 Stimulus setting

Annex 3 Advanced Time Domain Analysis

- e) Click [Next].
- f) After the Rise Time settings dialog box of the configuration wizard is displayed, click [Rise Time] and [Type].

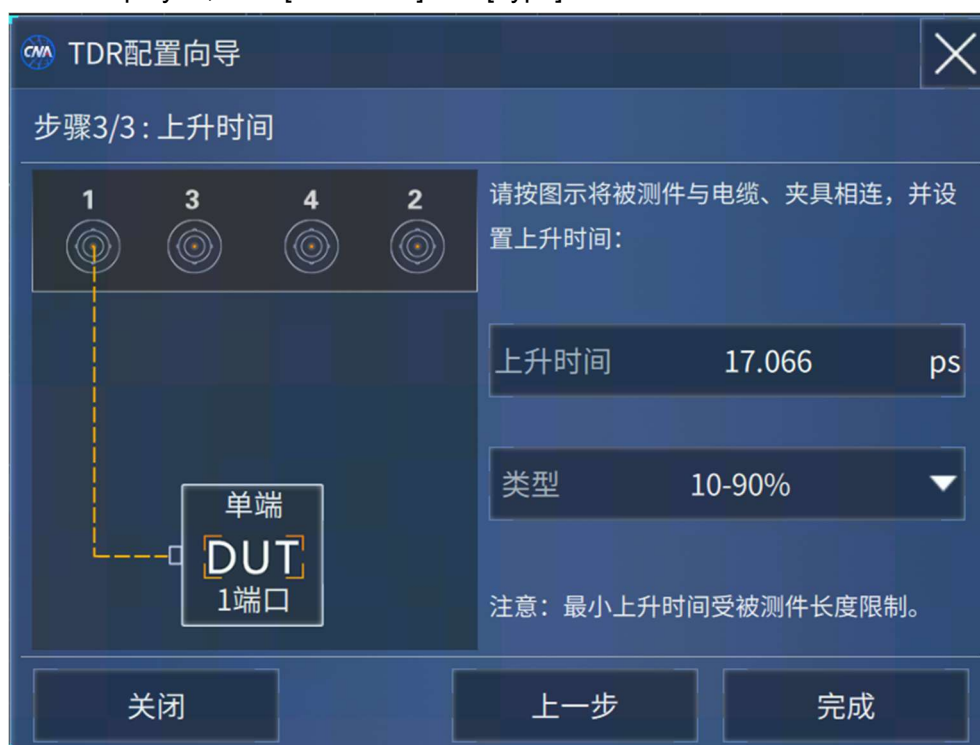


Figure 3.6 Select rise time

- g) Click [Finish] to finish the configuration wizard settings.

3.2.2 Manual Configuration

In addition to using the configuration wizard, users can also perform advanced time domain analysis measurement settings manually. The manual settings are performed on the "Configuration" tab. It mainly includes:

- Configuration Wizard
- Select the DUT topology
- Set the stimulus level
- Configure the stimulus
- Port Extensions
- Calibration
- Balance the port configuration
- Reset

Reset

- a) Click the **[Reset]** button under "Basic Configuration" to reset the advanced time domain analysis option.
- b) All settings shown in the basic area are changed to the default settings.

Notice

When the [Reset] button is clicked, the calibration and port extension data will be cleared. After resetting, all settings are default except DUT type.

DUT topology settings

In the Topology under “Basic Configuration”, select one of the available options from the drop-down list box. This function is the same as the topology of the DUT in Section 8.1 [Configuration Wizard]. A dialog box requesting confirmation will appear when the settings are complete. Click [OK] to continue.

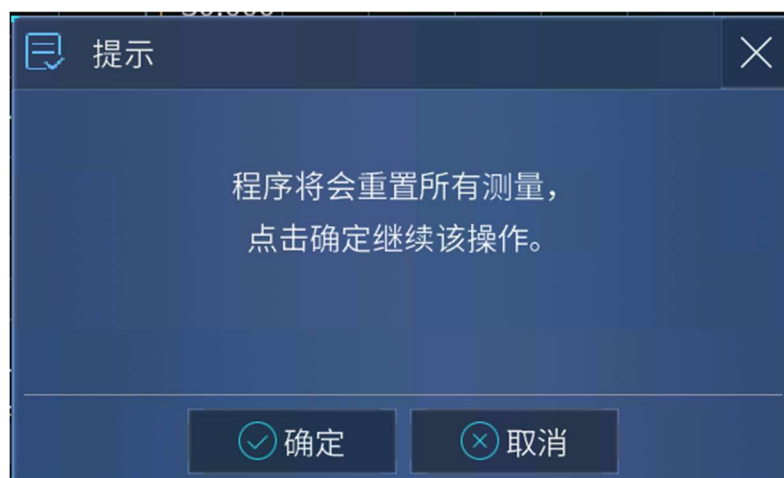


Figure 3.7 Change topology tip

Notice

After selecting a topology, an advanced time domain analysis option resetting will be performed. Therefore, when you change from one topology to another, the calibration and port extension data will be cleared.

Set the stimulus level

Amplitude used to set the stimulus level

Notice

The stimulus level amplitude is independent of the actual applied test level.

After double-clicking the [Stimulus Level] text box in the basic settings, an input dialog box will appear. Enter the stimulus level value and click [OK]. The new value set is displayed in the [Stimulus Level] text box.



Figure 3.8 Soft keyboard

Stimulus setting

The user uses this page to configure the stimulus of the TDR, and the length of the DUT is calculated automatically. By setting the start frequency and stop frequency, the software automatically calculates the number of measurement points that meet the requirements and automatically calculates the length of the DUT.



Figure 3.9 Measuring the length of the DUT

Notice

The automatic settings function can be used as a step in the configuration wizard. TDR measurement requires that the frequency domain signal is equal-stepped, and the first step is from 0 to the start frequency. So the frequency step is equal to the start frequency. In this way, the frequency domain can be converted to a time domain signal (low pass) by FFT. The velocity factor and the dielectric constant (ϵ_r) are set via the velocity factor and electrical delay edit boxes. The relationship between the two is as follows: $\text{velocity factor} = 1/\sqrt{\epsilon_r}$

Perform calibration

The advanced timing analysis option has the following three calibration methods:

- Port Extensions
- Port extension and loss compensation
- General calibration

More functions

- a) Reference impedance settings.

At "Reference Impedance" under "More", double-click it once. An input dialog box will appear. Type in the reference impedance value and click OK. The new value is displayed at [Reference Impedance].

- b) Dielectric constant and velocity factor

$$\text{Velocity factor} = 1 / \sqrt{(\text{dielectric constant})}.$$

When either of them is changed, the other values will be changed automatically.

Change the value of the dielectric constant. At "Dielectric Constant" under "More", double-click it once. An input dialog box will appear. Type in the value of dielectric constant and click **[Enter]**. The new value is displayed at **[Dielectric Constant]**.

At the [Velocity Factor] under "More", you can set the velocity factor value in the same way as the dielectric constant.

- c) Port power

At the "Port Power" under "More", double-click it once. An input dialog box will appear. Enter the port power and click **[Enter]**. The new value is displayed at **[Port Power]**.

- d) Avgs

The averaging function can reduce trace noise. When the Sweep Average function is turned on, it performs the number of sweeps specified by the average factor. To activate the Average option, select Average.

To turn on the average function, select the Pre-Average check box.

Enter the averages times (the number of sweeps to be performed).

- e) IFBW

Double-click once on **[IF Bandwidth]**. An input dialog box will appear. Enter the IF bandwidth value and click **[Enter]**. The new value is displayed on the IF bandwidth. Decreasing the IF bandwidth will improve the dynamic range and reduce the measurement noise level.

Channel configuration

Refer to the section “Advanced Analysis of Eye Diagram Waveform”.

3.2.3 Error Correction

There are many different ways to remove the effects of test fixtures and cables from a measurement. The 3657 advanced time domain analysis option provides you with three available error correction methods, as shown below.

- Port Extensions
- Port extension and loss compensation
- General calibration

Error correction can be performed in [Basic Configuration] under “Configuration” option tab.



Figure 3.10 Basic configuration

After clicking the [Port Extension] button, the "Auto Port Extension" dialog box will pop up, and refer to Section 6.5.2 for details of port extension and loss compensation.



Figure 3.11 Auto port extension

After clicking the [Calibrate] button, the "Calibration" dialog box will pop up, and refer to Chapter 7 for general calibration steps.

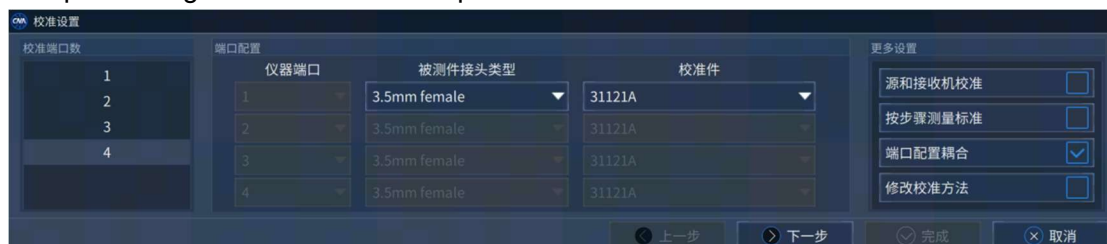


Figure 3.12 Calibration dialog box

3.3 Measurement Process

3.3.1 Trace Parameter Settings

Trace selection

There are two ways to select the displayed trace.

a) Use of the mouse

Double-click on any area of the graph box to exit the full view.

Click on the desired trace to select the trace.

b) Use the list box

Click on the [Trace] list box and select the trace number from the list, as shown below.



Figure 3.13 Trace control options

Notice

The number of traces is variable, and up to 16 traces can be displayed in this option. Select all time domains, frequency domains and mixed mode parametric traces.

Annex 3 Advanced Time Domain Analysis**Select parameters**

Click on the TDR/TDT tab to select the parameters shown below.

All measurements and formats are given in the following table.

Exhibit 3.1 Measurement formats

Measurement	Format
Frequency domain parameter formats	Log Mag
	Linear Mag
	Real part
	Imaginary part
	Group Delay
	SWR
	Phase
	Unwrap Pha
	Pos Phase
	Smith(Lin/Phase)
	Smith(Log/Phase)
	Smith(Re/Im)
	Smith(R + jX)
	Smith(G + jB)
	Polar coordinates (Lin/Phase)
	Polar coordinates (Log/Phase)
	Polar coordinates (Re / Im)
Time domain parameter format	Impedance
	Voltage
	Log Mag
	Linear Mag
	Real part



Figure 3.14 Parameter settings

The list boxes in the "Parameters" area change with the selection of the measurement, as shown below:

Exhibit 3.2 Measurement parameters

Measurement		The beginning of the table content
S parameter	Single-ended	S
	Differential	Sc,Sd
Time domain	Single-ended	T
	Differential	Tc,Td

Choice of stimulus type

There are two choices under the stimulus type.

- Lowpass@Step
- Lowpass impulse

There are two choices of rise time.

- 10 - 90%
- 20 - 80%

The rise time settings are independent for TDR/TDT mode and eye diagram mode.

3.3.2 Cursor and Its Search Function

Refer to section 5.2.7 for regular cursor and its search functions.

Incremental cursor

The incremental cursor can be used for time increment measurement in the time domain. Traces other than those in the time domain cannot be used for time increment measurement.

- Click on the start point of the trace for which you need to

Annex 3 Advanced Time Domain Analysis

measure the time increment.

- b) Click the [Incremental Cursor] button in "Trace Control".
- c) After the "Incremental Cursor" dialog box pops up, select the termination trace, set the [Start Trace Target Value] and [Termination Trace Target Value], and check the [Incremental Cursor] check box to complete the test.
- d) Set the [Limit Value] text box and check the [Limit Test] check box to complete the limit test of incremental time.

During the limit test, if the absolute value of the time increment is less than the limit value, the test is successful.



Figure 3.15 Incremental cursor

The time increment data is displayed in the upper left corner of the graphical area of the application program interface.

3.3.3 Use of Time Domain Gates

The time domain gate function provides the ability to measure the response of a particular circuit element in the frequency domain by algorithmically removing unwanted responses. When you define a time domain gate on a time domain response curve, the time domain gate control is either removed or retained. The effect of the time domain gate function on the S-parameter data can be seen by looking at the original frequency domain response and the frequency domain response after the time domain gate is added.

See Section 2.4 in Annex 2 for the specific procedure of time domain gate.

3.4 Eye Diagram and Eye Diagram Template Measurements

3.4.1 Eye Diagram Measurement

Eye diagrams are commonly used to analyze signal quality in the oscilloscope. You

Annex 3 Advanced Time Domain Analysis

can determine problems such as effect of attenuation, noise, jitter and dispersion on the system, by using an eye diagram.

The 3657's advanced time domain analysis option provides emulated eye diagram analysis function, thus eliminating the need for a hardware pulse pattern generator. The simulation code type generator is used to define an simulation code type and then convolve the defined simulation code type with the impulse response of the DUT to create a very accurate eye-diagram-based measurement.

a) Eye diagram test

1. Select the trace of the eye diagram you want to observe.
2. Click [Draw Eye Diagram] to display the eye diagram.
3. When you change the settings of the stimulus type, you need to click **[Draw Eye Diagram]** to reflect the settings to the waveform.

b) Display results

1. Select **[Rise Time]** under "Test Results".
2. Click **[Draw Eye Diagram]** to display the results. When you change the "Rise Time" settings, you need to click **[Draw Eye Diagram]** to reflect the result.

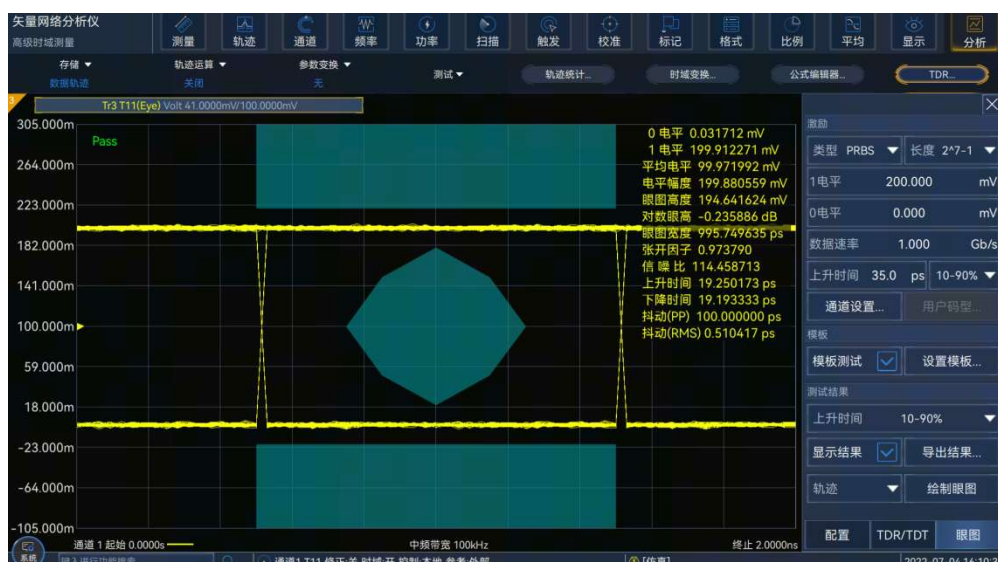


Figure 3.16 Eye diagram display

Figure 3.3 Eye diagram measurement parameters

Name	Unit	Description
Rise time	Picosecond	Rise time definition = 10%-90%: time at 90% level - time at -10% level Rise time definition = 20%-80%: time at 80% level - time at -20% level
Fall time	Picosecond	Down time definition = 10%-90%: time at 90% level - time at 10% level Down time definition = 20%-80%: time at 80% level - time

Annex 3 Advanced Time Domain Analysis

		at 20% level
Jitter RMS	Picosecond	1σ width of the histogram at the eye intersection
Jitter peak-to-peak	Picosecond	Full width of the histogram at the eye intersection
Crossover percentage	%	Crossover height / amplitude $\times 100$
Expansion factor	None	$(1 \text{ level} - \sigma_1) - (0 \text{ level} + \sigma_0)/\text{amplitude}$
Signal/noise ratio	None	$(1 \text{ level} - 0 \text{ level})/(\sigma_1 + \sigma_0)$
Duty ratio distortion	Picosecond	$ T_{\text{rise middle}} - T_{\text{fall middle}} $
Duty ratio distortion (%)	%	Duty ratio distortion (s)/ bit period $\times 100$
0 level	Voltage	Histogram average of 0 level
1 level	Voltage	Histogram average of 1 level
Average level	Voltage	$(0 \text{ level} + 1 \text{ level})/2$
Level amplitude	Voltage	$1 \text{ level} - 0 \text{ level}$
Eye diagram height	Voltage	$(1 \text{ level} - 3\sigma_1) - (0 \text{ level} + 3\sigma_0)$
Eye diagram width	Picosecond	Bit period - $2 \times 3 \times \text{jitter RMS}$
• Bit period = $1/\text{bit rate}$ • Input amplitude = set 1 level - set 0 level • $T_{\text{rising middle}}$ = time for the rising edge to cross the intermediate threshold (50%) • $T_{\text{falling middle}}$ = time for falling edge to cross intermediate threshold (50%)		

c) Save the results to a file

The results can be saved as a text file.

1. After clicking **[Export to csv]**, the **[Save Eye Diagram Results]** dialog box will pop up.
2. Type in the desired file name and then click **[Save]**.

Annex 3 Advanced Time Domain Analysis

```

1 # Ceyear 3661 TDR Eye Results,
2 # 2019-10-16 15:43:02,
3 #,
4 Level Zero,      -0.000,
5 Level One,       0.200,
6 Level Mean,      0.100,
7 Amplitude,       0.200,
8 Eye Height,      0.190,
9 Eye Height,      -0.459,
10 Eye Width,       0.000,
11 Open Factor,     0.948,
12 SNR,            57.336,
13 Rise Time,       0.000,
14 Fall Time,       0.000,
15 Jitter PP,       0.000,
16 Jitter RMS,      0.000,

```

Figure 3.17 Example of saving results

d) Eye diagram scaling

By default, the eye diagram is set to automatic settings, but you can also set the scale manually.

1. In the "Scale" area, the **[Scale]** and **[Reference Value]** options will be activated.
2. Click on the input box **[Scale]** and enter the Y coordinate value.
3. Click on the input box **[Reference Value]** and enter the Y coordinate reference value.



Figure 3.18 Eye diagram scaling settings

3.4.2 Simulation Code Type Selection

The 3657's advanced time domain analysis option can provide the ability to analyze simulated eye diagram, thus eliminating the need for a hardware pulse pattern generator. The simulation code type is selected from the following:

- PRBS
- ABS
- K28.5
- User Defined

a) Simulation code type selection

1. Select your desired simulation code type in the **[Type]** under "Stimulus" area.
2. If you choose **[PRBS]** or **[ABS]**, **[Length]** will be activated. Then, select **[Length]**.

Annex 3 Advanced Time Domain Analysis



激励

类型 PRBS ▼ 长度 2^{7-1} ▼

1电平 200.000 mV

0电平 0.000 mV

数据速率 1.000 Gb/s

上升时间 35.0 ps 10-90% ▼

通道设置... 用户码型...

Figure 3.19 Stimulus setting

3. User-defined simulation code type

Users can easily create custom simulation code type. The bit length should be between 8 and 32768 (2^{15}). Simulation code types containing only 0 or 1, or only 1 change edge are not allowed to be set (e.g. 00,111,0000).

Simulation code type setting

The following parameters can be set for the simulation code type

Table 3.4 Description of simulation code type parameters

Tab	Description
1 level	Regarding the Y-coordinate scale bit "1" of the eye diagram expressed in volts, negative voltages are allowed. For differential eye diagrams, these scale values become doubled.
0 level	Regarding the Y-coordinate scale bit "0" of the eye diagram expressed in volts, negative voltages are allowed. For differential eye diagrams, these scale values become doubled.
Data rate	The rate at which data is transmitted on a circuit or communication line, measured in bits/second.
Rise time	The time it takes for a signal to go from low level to high level, with a maximum value of 40% of the bit bandwidth (bit bandwidth = 1/bit rate). The time can be defined as "10%-90%" or "20%-80%". The rise time is set independently in the [Eye Diagram] and [TDR/TDT].

3.4.3 Eye Diagram Template Test

The eye diagram template test allows you to verify that an eye diagram conforms to the industrial standard eye diagram template definition. In order to comply with industry

Annex 3 Advanced Time Domain Analysis

standards, the input waveform must remain outside the shaded template area. The template test can be found under "Scale".



Figure 3.20 Eye diagram scaling

a) Define templates

Set up the top/bottom template and the polygon template in the center of the eye diagram according to the indicator requirements of the DUT in the Eye Diagram Template Setting window, click OK and check the template test function in the Eye Diagram Setting panel to enable the eye diagram template test.

The available template shapes include "Octagon", "Hexagon" and "Quadrilateral".



Figure 3.21 Customize templates

b) Execute the template test

1. Select the [Template Test] check box under "Scale/Template".
2. Click [Draw Eye Diagram] to redraw the eye diagram pattern and template.
3. The template and success/failure results will be displayed on the screen.

Annex 3 Advanced Time Domain Analysis

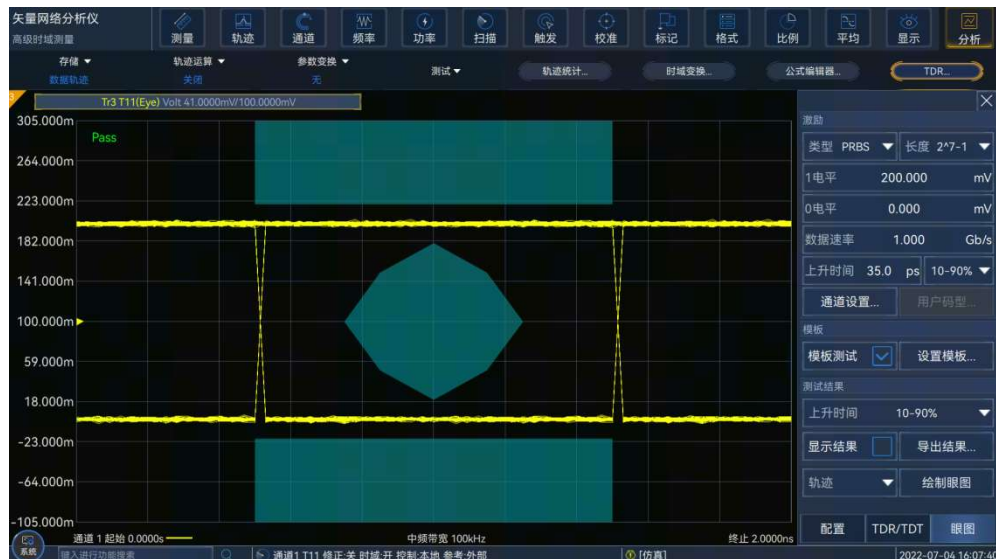


Figure 3.22 Template test

3.5 Advanced Analysis of Eye Diagram Waveforms

In the advanced analysis of the eye diagram waveform, pre-emphasis and equalization help improve the quality of the waveform at the receiving end, and thus the quality of the eye diagram.

Click the [Channel Settings] button under the [Eye Diagram] page to enter the advanced analysis function of the eye diagram waveform.



Figure 3.23 Advanced analysis settings of eye diagram waveform

According to the left and right buttons, the observation point of the eye diagram can be selected.

- The observation before the DUT indicates the eye diagram of the source.
- The observation after the DUT indicates the eye diagram

of the receiver.

3.5.1 Jitter Injection

- Click the [Jitter] button and check [Enable] to perform jitter injection to the source.
- When jitter injection is turned on, a check mark will appear on the [Jitter] button. Three types of jitter injections are available:

- Random jitter - follow a Gaussian distribution and is expressed as the root mean square value of a random jitter distribution.
- Periodic jitter - expressed as a peak-to-peak value.
- Dirac jitter.



Figure 3.24 Jitter injection settings

3.5.2 Noise Injection

- Click the [Noise] button and check [Enable] to apply noise injection to the source.
- When noise injection is turned on, a check mark will appear on the [Noise] button.

Annex 3 Advanced Time Domain Analysis



Figure 3.25 Noise injection settings

3.5.3 Pre-emphasis

- To perform pre-emphasis, click the [Pre-emphasis] button and check [Enable].
 - When pre-emphasis is turned on, a check mark appears on the [Pre-emphasis] button.
- The pre-cursor is the ratio of V_d and V_c :

$$\text{Pre-cursor} = 20 \log_{10} (V_d/V_c)$$
 - Post-1 cursor is the ratio of V_b and V_a :

$$\text{Post 1 cursor} = 20 \log_{10} (V_b/V_a)$$
 - Post 2 cursor is the ratio of V_c and V_b :

$$\text{Post 2 cursor} = 20 \log_{10} (V_c/V_b)$$

Notice

When pre-emphasis is turned on, the sum of the absolute values of the pre-emphasis FIR filters is normalized.



Figure 3.26 Pre-emphasis settings

3.5.3 Equalization

- To perform equalization, click the [Equalize] button and check [Enable].
- When pre-emphasis is turned on, a check mark will appear on the [Equalize] button.



Figure 3.27 Equalization settings

The user can choose to use the equation by specifying the variable or the equation file.

In order to generate the filter, four variables must be specified in this equation:

- DC gain
- Zero frequency
- Pole 1 frequency
- Pole 2 frequency