3674 Series
Vector network analyzer
User's Manual



Ceyear Technologies Co., Ltd.

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

- 3674 series vector network analyzers:

  3674B Vector Network Analyzers

  3674C Vector Network Analyzers

  3674D Vector Network Analyzers

  3674E Vector Network Analyzers

  3674F Vector Network Analyzers

  3674G Vector Network Analyzers

  3674H Vector Network Analyzers

  3674K Vector Network Analyzers

  3674L Vector Network Analyzers

Options other than standard accessories are as follows:

- Two-port programmable step attenuator
- Four-port measurement
- Four-port programmable step attenuator
- Active intermodulation measurement
- Pulse measurement
- Time domain measurement
- Advanced time domain measurement
- Frequency offset measurement
- Mixer scalar measurement
- Mixer vector measurement
- Embedded LO frequency converter
- Gain compression measurement by QR code sweep
- Noise Figure Measurement

Version: A.5 August 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 3674 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.

AV2.733.1220SSCN

Version

A.5 2023.8

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 network analyzer 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

Warning

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

Operation Instructions I

1. Manual navigation 13

1.1. About the manual 13
1.2 Related Documents 14

2 Overview 17

2.1 Product Overview 17
2.2 Safe Operation Guide 22
2.2.1 Safety marks 22
2.2.2 Operation Status and Locations 24
2.2.3 Electrical safety 24
2.2.4 Operation Precautions 26
2.2.5 Maintenance 27
2.2.6 Batteries or power modules 27
2.2.7 Transportation 28
2.2.8 Waste disposal/environmental protection 28

3 Quick Start 31

3.1 Get prepared 31
3.1.1 Preparations before operation 31
3.1.2 System recovery and installation procedures for the analyzer 46
3.1.3 Routine Maintenance 49
3.2 Front and Real Panels 50
3.2.1 Front Panel 50
3.2.2 Rear Panel Description 54
3.3 Analyzer Software Interface 61
3.4 Analyzer Traces, Channels and Windows 62
3.4.1 Trace 62
3.4.2 Channel 63
3.4.3 Window 63
3.5 Analyze data 64

3.5.1 Marker 64
3.5.2 Trace Operation and Statistics 73
3.5.3 Limit test 76
3.5.4 Ripple test 79
3.5.5 BW test 81
3.5.6 Formula Editor 84
3.6 Data output 90
3.6.1 Save and callback files 90
3.6.2 Printer displaying measurement results 96
4 Measurement Settings 101
4.1 Reset analyzer 101
4.1.1 Default reset state 101
4.1.2 User Define State 104
4.1.3 Reset analyzer 105
4.2 Measurement parameter selection 105
4.2.1 S-parameter 105
4.2.2 Differential balance S-parameter 108
4.2.3 Arbitrary ratio 111
4.2.4 Non-ratio power measurement 112
4.2.5 Changing trace measurement type 113
4.3. Setting frequency range 113
4.4 Set the power level of signal 114
4.5 Set sweep 117
4.5.1 Overview of sweep type 117
4.5.2 Sweep type setting 118
4.5.3 Sweep time 121
4.5.4 Sweep setting 122
4.6 Trigger mode 123
4.6.1 Simple trigger setting 123
4.6.2 Detailed trigger setting 123
4.6.3 Trigger dialog box 124
4.7 Set data format and scale 129

4.7.1 Data format 129
4.7.2 Set data format 132
4.7.3 Scale 133
4.8 Observe multiple traces and open multiple channels 134
4.8.1 Automatically Set to Multi-window Display 134
4.8.2 Manually Add Measurement Traces and Windows 136
4.9. Set the analyzer display 136
4.9.1 Status bar 136
4.9.2 Toolbar 136
4.9.3 List 137
4.9.4 Display contents 137
5 Menus 140
5.1 Menu Structure 140
5.1.1 Meas 141
5.1.2 Trace 142
5.1.3 Chan 143
5.1.4 Freq 144
5.1.5 Power 145
5.1.6 Sweep 146
5.1.7 Trigger 147
5.1.8 Cal 148
5.1.9 Mrk 149
5.1.10 Format 150
5.1.11 Scale 151
5.1.12 Average 151
5.1.13 Display 153
5.1.14 Analysis 157
5.1.15 System 160
5.2 Menu Description 163
5.2.1 Meas 163
5.2.2 Trace 165
5.2.3 Channel 166

5.2.4 Frequency 169
5.2.5 Power 171
5.2.6 Sweep 173
5.2.7 Trigger 175
5.2.8 Cal 177
5.2.9 Mrk 182
5.2.10 Format 185
5.2.11 Scale 186
5.2.12 Average 189
5.2.13 Display 191
5.2.14 Analysis 193
5.2.15 System 196

6 Optimization & Measurement 201

6.1 Reducing the impact of accessories 201
6.2 Improving reflection accuracy of low-loss two-port devices 202
6.3 Add dynamic range 204
6.3.1 Increase DUT Input Power 205
6.3.2 Decrease the receiver noise base 206
6.4 Improve Measurement Results of Electrical Length Devices 207
6.5. Improve the accuracy of phase measurement 209
6.5.1 Electrical Delay 209
6.5.2 Port Extension 209
6.5.3 Phase Offset 212
6.5.4 Spacing of frequency points 213
6.5.5 Specific Operations 214
6.6. Reduce trace noise 215
6.6.1 Sweep Averaging 216
6.6.2 Trace smoothing 216
6.6.3 IFBW 217
6.7 Increase Data Points 218
6.8. Improve measurement stability 219
6.9 Increase the sweeping speed 221

6.10 Improving multi-state measurement efficiency 225
6.10.1 Improving measurement efficiency via measurement settings 225
6.10.2 Automatically Change Measurement Setting 226
6.10.3 Quickly Call Measurement 226
6.11 Accelerating Data Transmission 227
6.11.1 Using single trigger mode 227
6.11.2 Transmitting as little data as possible 227
6.11.3 Using real number format 227
6.11.4 Use LAN 227
6.11.5 Using COM programs 228
6.12 Use macro 228
6.12.1 New Macro 228
6.12.2 Macro Settings dialog box 229
7 Calibration 231
7.1 Overview of calibration 231
7.1.1 Definition of calibration 231
7.1.2 Significance of calibration 232
7.1.3 Calibration applications 232
7.1.4 Simple procedures for calibration 232
7.2 Selecting calibration type 233
7.3 Calibration wizard 236
7.3.1 Run Calibration Wizard 236
7.3.2 Perform Calibration Settings 237
7.3.3 Modify Calibration Method 239
7.3.4 Perform Source Calibration Settings 241
7.3.5 Measurement Standard 242
7.3.6 Finish Calibration 244
7.3.6 Save as User Calibration Set 244
7.4 Calibration Set 245
7.4.1 Definition of Calibration Set 246
7.4.2 Display Calibration Set Properties 247
7.4.3 Save Calibration Set As 248

7.4.4 Delete Calibration Set 249
7.4.5 Delete All Calibration Sets 249
7.4.6 Apply/Unapply Calibration Set 249
7.5 Measurement error 250
7.5.1 Drift error 251
7.5.2 Random error 251
7.5.3 System error 252
7.5.4 High-accuracy Measurement Calibration 255
7.6 Edit Calibration Kit 257
7.6.1 Definition of calibration kit 257
7.6.2 Self-define calibration kit 257
7.6.3 Create calibration kit 258
7.6.4 Edit Calibration Kit 259
7.6.5 Edit Calibration Kit Dialog Box 260
7.6.6 Edit Connector Dialog Box 261
7.6.7 Class Information Dialog Box 262
7.6.8 Add Std Dialog Box 263
7.6.9 OPEN Dialog Box 264
7.6.10 SHORT Dialog Box 265
7.6.11 Load Dialog Box 266
7.6.12 THRU/TL Dialog Box 267
7.6.13 DataBased Standard Dialog Box 267
7.7. Calibration standard 268
7.8 TRL calibration 270
7.9 Fixture Compensation Calibration 271
7.9.1 Fixture Simulation Terminologies 272
7.9.2 Fixturing 273
7.9.3 Processing Sequence 273
7.9.4 Port Extension 274
7.9.5 2-port De-embed 279
7.9.6 Port Matching Circuit Embed 281
7.9.7 Port Z Conversion 285
7.9.8 4-port De-embed/Embed 285

7.9.9 AFR 288
7.9.10 Cal Plane Manager 293

8 Network Measurement Fundamentals 309

8.1 Reflection measurement 309
8.1.1 Expression of reflection measurement 310
8.1.2 Summary of reflection measurement expression 312
8.2 Phase measurement 313
8.2.1 What is phase measurement 313
8.2.2 Why phase measurement is made 313
8.2.3 Using the analyzer's phase format 314
8.2.4 Type of phase measurement 314
8.2.5 Linear Phase Offset and Group Delay 315
8.3 Amplifier parameter description 315
8.3.1 Gain 315
8.3.2 Gain flatness 316
8.3.3 Reserve isolation 316
8.3.4 Gain Drift Change (Temperature, Bias) with Time 316
8.3.5 Linear Phase Deviation 316
8.3.6 Group delay 316
8.3.7 Return loss (VSWR, ρ) 317
8.3.8 **Complex impedance** 317
8.3.9 Gain compression 317
8.3.10 AM-PM Conversion Coefficient 317
8.4 Complex impedance 318
8.4.1 What is complex impedance 318
8.4.2 Improve the accuracy of impedance measurement 318
8.4.3 Measurement steps of complex impedance 319
8.5 Group delay 320
8.5.1 What is group delay 320
8.5.2 Reason for Measuring Group Delay 321
8.5.3 What is group delay aperture 322
8.5.4 Improve the measurement accuracy of group delay 323

8.5.5 Measurement steps of group delay 323
8.6. Absolute output power 324
8.6.1 What is absolute output power 325
8.6.2 Why do we measure absolute output power 325
8.6.3 Measurement steps for absolute output power 325
8.7 AM-PM Conversion 326
8.7.1 What is AM-PM Conversion 326
8.7.2 Reason for Measuring AM-PM Conversion 327
8.7.3 Factors related to measurement accuracy 327
8.7.4 Measurement Steps for AM-PM Conversion 328
8.8 Linear Phase Deviation 329
8.8.1 What Is Linear Phase Offset 329
8.8.2 What Is Linear Phase Offset 329
8.8.3 Reason for Measuring Linear Phase Deviation 330
8.8.4 Using electrical delay function 330
8.8.5 Factors related to measurement accuracy 330
8.8.6 Measurement Steps For Linear Phase Deviation 330
8.9 Reserve isolation 331
8.9.1 What is reverse isolation 331
8.9.2 Why measure reverse isolation 331
8.9.3 Factors related to measurement accuracy 332
8.9.4 Measurement steps of reserve isolation 332
8.10 Small signal gain and flatness 333
8.10.1 What is gain 333
8.10.2 What is flatness 333
8.10.3 Why measure small signal gain and flatness 334
8.10.4 Factors related to measurement accuracy 334
8.10.5 Measurement steps for small signal gain and flatness 334
9 Remote Control 337
9.1 Remote Control Basics 337
9.1.1 Remote Control Interface 337
9.1.2 Message 341

9.1.3 SCPI 342
9.1.4 Command Sequence and Synchronization 352
9.1.5 State Reporting System 353
9.1.6 Programming Precautions 356
9.2 Instrument Program Port and Configuration 356
9.2.1 LAN 356
9.2.2 GPIB 359
9.2.3 USB 361
9.3 Basic VISA interface programming 363
9.3.1 VISA Library 363
9.3.2 Initialize and Set Default State 364
9.3.3 Send setting command 366
9.3.4 Read the State of Measuring Instrument 368
9.3.5 Read Frequency Marker 369
9.3.6 Reading Trace Data 369
9.3.7 Command Synchronization 370
9.4 I/O library 371
9.4.1 Overview of I/O Library 371
9.4.2 Installation and configuration of I/O library 372
10 Troubleshooting and after-sales services 375
10.1 Working Principles 375
10.1.1 Response of DUT to the RF signal 375
10.1.2 Analyzer principles 376
10.2 Troubleshooting 377
10.2.1 System problems 377
10.2.2 Abnormal curve display 378
10.2.3 Sweep failure 378
10.2.4 Display problem 379
10.2.5 No response from front panel keys 379
10.2.6 Menu Operation Failure 379
10.3 Error Message 379
10.3.1 Local Error 380

10.3.2 Program control error information 381
10.4 Method to Obtain After-sales Services 400
10.4.1 Contact us 400
10.4.2 Package and mailing 401
Annex 403
Appendix 1 Introduction to typical measurements 403
1.1 After 30 minutes of preheating, reset the analyzer 404
1.2 Set frequency and power 404
1.3 Select a Measurement and Create a Trace 404
1.4 Calibration 406
1.5 Connect the DUT 407
1.6 Adjust scale and data analysis 407
1.7 Record or Save Data 408
Appendix 2 Time domain measurement 409
2.1 Time domain measurement principle 409
2.2 Time domain measurement resolution and range 411
2.3 Windows filtering 417
2.4 Time Gate Filtering 420
2.5 Time domain measurement data 425
2.6 Bandpass and lowpass time domain modes 431
2.7 Setting of time domain conversion measurement 434
2.8 Time Domain Reflectometer (TDR) Impedance Test 437
Annex 3 Frequency Offset Measurement 441
3.1 Frequency Offset Measurement Setting 441
3.2 Coupled and Uncoupled Mode Setting 443
3.3 Frequency Offset Measurement Calibration 445
3.4 Typical Applications of Frequency Offset Measurement 446
Annex 4 Pulse Measurement 453
4.1 Pulse Measurement Principle 453
4.2 Overview of Pulse Measurement 454
4.3 Pulse Measurement Steps 462
Annex 5 Amplifier Gain Compression Measurement 468

5.1 Amplifier Gain Compression Principle 468
5.2 Overview of Gain Compression Measurements 469
5.3 Gain Compression Measurement Steps 475
Annex 6 Mixer Measurement 485
6.1 Overview of Mixer Measurements 485
6.2 Scalar Mixer Measurements 486
6.3 Scalar Mixer Measurement Calibration 496
6.4 Vector Mixer Measurement 501
6.5 Vector Mixer Measurement Calibration 504
6.6 Embedded LO Mixer Measurement 508
Annex 7 Intermodulation Distortion Measurement 514
7.1 Concept of Intermodulation Distortion 514
7.2 Intermodulation Distortion Parameters 515
7.3 Amplifier Sweep Frequency Intermodulation Distortion Measurement 520
7.4 Amplifier Sweep Frequency Intermodulation Spectrum Measurement 530
Annex 8 Noise Figure Measurement 537
8.1 Overview of Noise Figure Measurement 537
8.2 Instructions of Noise Figure Option 541
8.3 Use the Noise Figure Option Application 543
8.4 Converter Noise Figure Measurement 562
Annex 9 Advanced Time Domain Analysis 572
9.1 Overview of Advanced Time Domain Analysis 572
9.2 Measurement Settings 574
9.3 Measurement Process 581
9.4 Eye Diagram and Eye Diagram Template Measurements 583
9.5 Advanced Analysis of Eye Diagram Waveforms 588
Annex 10 Fast Continuous Point Measurement Mode 593
10.1 Overview 593
10.2 Measurement description 593
10.3 Description of Remote Control Commands 594

1. Manual Navigation

This chapter introduces the user's manual functions, chapter structure and main contents of 3674 series vector network analyzer, as well as the instrument-related documents provided to users.

- About the Manual.....13
- Related Documents.....14

1.1. About the Manual

This manual introduces the applications, performance indicators, basic working principles, operation, precautions and other information of the 3674 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 commends 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 3674 series vector network analyzer, including a product overview and a guide for safe use.

- **Quick Start**

  This chapter introduces the preparations before using the 3674 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 introduces 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

reasonable setting 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 3674 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: program control fundamentals, introducing concepts related to program control, software configuration, program control ports, SCPI commands, etc; port configuration methods, introducing the connection methods and software configuration methods for program-controlled ports of 3674 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 3674 series vector network analyzer and the test method instructions recommended for users.

- **Annex**

  This part contains operation instructions of options for 3674 series vector network analyzer.

1.2 Related Documents

Product documentation for 3674 series vector network analyzers includes:

- User's Manual

- Remote 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, remote 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

Remote 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 remote control commands and methods of the instrument. Main chapters include:

- Remote Control
- Remote 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

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 3674 series vector network analyzers. It also gives introductions on correct operation of the instrument and precautions such as electrical safety.

- Product Overview……17
- Safe Operation Guide……22

2.1 Product Overview

The 3674 series vector network analyzer is an upgraded 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 embedded computer with high-performance microprocessor chip and a platform environment based on Windows/Linux OS are adopted, greatly enhancing the interconnection and availability of the network analyzer.

The 3674 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, GPIB and DP. In addition to all measurement functions available for traditional vector network analyzer, the 3674 series vector network analyzer also supports multi-function comprehensive parameter tests for mixer/frequency converter, active intermodulation distortion and harmonic distortion, gain compression two-dimensional sweep and pulse network S-parameter, and is able to accurately measure the magnitude-frequency characteristics, phase-frequency characteristics and group delay characteristics of microwave network.

The 3674 series vector network analyzer can be widely applied in transmitting/receiving (T/R) module measurement, dielectric material characteristics measurement, microwave pulse characteristics measurement, photoelectric characteristics measurement and other measurements, and is an essential test device in development and production of phased array radar, communication devices, microwave RF devices and other systems.

Product features:

- 15.6-inch high-resolution touch screen;
- Windows/Linux 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;
- 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;
- Interfaces including USB, GPIB, LAN and DP available;
- Single-source stimulus dual-port vector network analyzer and dual-source stimulus four-port vector network analyzer optional;
- Functions including pulse measurement, time domain measurement, mixer measurement, active intermodulation distortion measurement, gain compression two-dimensional sweep measurement, as well as MMW spread spectrum, antenna and RCS measurement reception supported.

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 3674 series vector network analyzer is equipped with a 15.6-inch 1920*1080 touch screen (as shown in Figure 2.1), making the operation easier.

Figure 2.1 15.6-inch 1920*1080 touch screen



Figure 2.2 Soft panel interface

2) Multi-window and multi-channel measurement display

The 3674 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.



Figure 2.3 Multi-window measurement display

3) Wide dynamic range

The 3674 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.

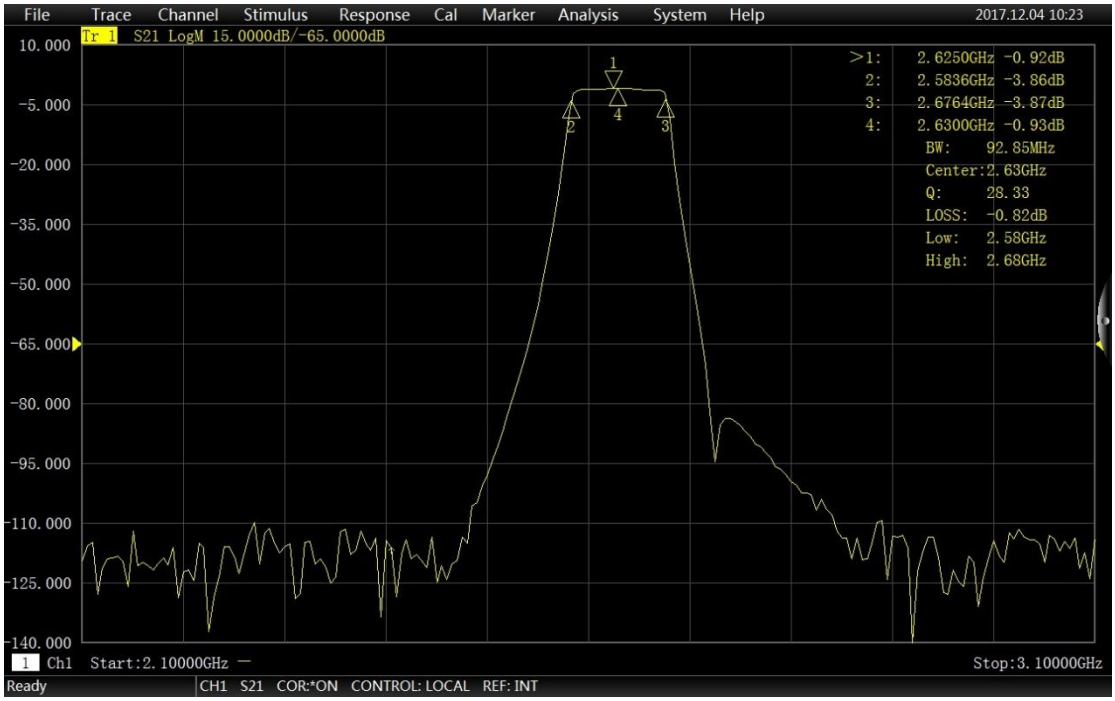


Figure 2.4 Filter measurement results

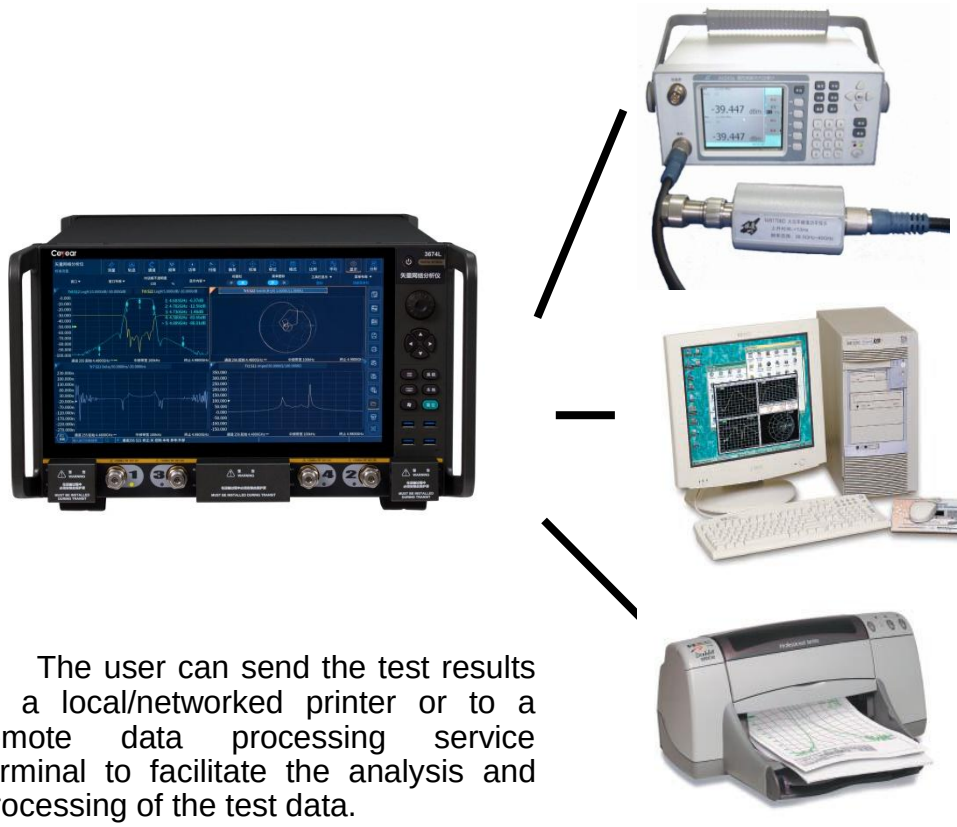
4) Auto test

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 LAN, USB or GPIB interface.

3) The application can be directly run without the need to connect an external PC.



The user can send the test results to a local/networked printer or to a remote data processing service terminal to facilitate the analysis and processing of the test data.

Figure 2.5 Auto test

5) GPIB interface

The 3674 series vector network analyzer provides a 24-pin D-type female GPIB connector, IEEE-488 compliant, for sending and receiving GPIB/SCPI commands.

6) USB port

The 3674 series vector network analyzer provides nine fast USB interfaces (eight in type A configuration and one in type B configuration) for easy connection to keyboards, mice, printers, electronic calibration kits, and other peripheral devices with USB interfaces.

7) Printing

The 3674 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.....22
- Operation Status and Locations.....24
- Electrical Safety.....24
- Operation Precautions.....26
- Maintenance.....27
- Batteries and Power Modules.....27
- Transportation.....28
- Waster Disposal/Environmental Protection.....28

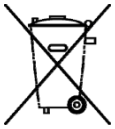
2.2.1 Safety Marks

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

	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.
	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.		

2.2.1.2 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 3674 series vector network analyzer should be operated in an environment that allows for smooth placement, IP protection 2X 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 ±10% of the marked voltage, and the range of supply frequency is ±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 Vrms > 30 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.).

13) The instrument shall comply with IEC60950-1/EN60950-1 or IEC61010-1/EN 61010-1 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. 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 or 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) 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.

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 3674 series vector network analyzers, so that users can have a preliminary understanding of the instrument itself and its measurement processes.

- Get Prepared.....31
- Front and Real Panels.....50
- Analyzer Interface.....61
- Analyzer Traces, Channels and Windows.....62
- Data Analysis.....64
- Data Output.....90

3.1 Get Prepared

- Preparation Before Operation.....31
- System recovery and installation procedures for the analyzer.....46
- Routine Maintenance 49

3.1.1 Preparations before Operation

The chapter introduces precautions for 3674 series vector network analyzers before initial setting 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.....33
- Environmental Requirements.....37
- Turning on/off the Power.....38
- Correct Use of Connectors.....42

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
Mainframe:	
✧ **3674 series vector network analyzers**	—
Standard Configuration:	
✧ **3-core power cord**	—
✧ **USB mouse**	—
✧ **Packing List**	—
✧ **Product Certificate of Conformity**	—
Options	
✧ **006 English option**	For setting language of front and rear panels and operating system to English.
✧ **009 Two-port spread spectrum controller**	For frequency spread of vector network analyzer to achieve function expansion of two-port vector network analyzer.
✧ **010 Four-port spread spectrum controller**	For frequency spread of vector network analyzer to achieve function expansion of four-port vector network analyzer.
✧ **011 110GHz coaxial expansion module**	For frequency spread of vector network analyzer to achieve coaxial S-parameter measurement within 10MHz ~ 110GHz.
✧ **060 External control panel**	For expanding front panel to achieve remote control with more buttons available.
✧ **061 Extended workbench**	For extending the workbench during test with a dimension of 500mm (L) x 350mm (W).
✧ **062 Cabinet mounting kit**	Special kit for installation to the cabinet.
✧ **063 User Manual (hardcopy)**	Providing detailed User Manual of hardcopy version.
✧ **S05 S-parameter signal integrity analysis function**	For analyzing the integrity characteristics of the system signals such as frequency domain signal, TDR signal and crosstalk signal, and automatically converting curves into test reports.
✧ **S07 Automatic fixture removal (AFR) function**	For automatic testing and removal of measurement fixtures for single-ended and balanced devices.
✧ **S10 Time domain measurement function**	For time domain measurements to locate and analyze breakpoints in devices, fixtures or cables.
✧ **S11 Advanced time domain**	For TDR impedance test, eye diagram analysis,

	analysis function	etc.
✧	**S18 Fast continuous wave sweep function**	For real-time data reading with FIFO method applied.
✧	**S16 True differential measurement function**	For balanced parameter measurement of true differential/common-mode stimulus.
✧	**S20 Frequency offset measurement function**	For frequency offset measurement.
✧	**S22 Mixer/converter scalar measurement function**	For mixer scalar parameter measurement.
✧	**S24 Embedded LO frequency converter measurement function**	For embedded LO frequency measurement.
✧	**S26 Gain compression measurement function**	For gain compression measurement of active devices such as amplifiers.
✧	**S28 Phase sweep measurement function**	For phase sweep measurement.
✧	**S31 THD measurement function**	For total harmonic distortion (THD) test of differential amplifier.

Table 3.1 List of articles provided together with 3674

Table 3.1 (Continued)

Name	Function
Options	
✧ **003 noise figure measurement**	For accurate measurement of S-parameters, noise figure and noise parameters.
✧ **008 Pulse measurement**	For S-parameter measurement in pulse state, with ports 1 and 3 outputting pulse modulation signal.
✧ **018 Multiport extension unit**	For extending the vector network analyzer to be a 16-port one.
✧ **023 Mixer/converter vector measurement function**	For mixer vector parameter measurement.
✧ **201 two-port programmable step attenuator**	2 programmable step attenuators are configured in source channel and receiver channel respectively.
✧ **203 Two-port 500Hz LF extension**	For extending the lower limit of the frequency range to 500Hz.
✧ **204 Configurable test device**	For extending the test device of the two-port vector network analyzer with panel jumpers added, with receivers A, B, R1 and R2 able to be used independently.
✧ **205 Two-port T-bias**	2 T-biases are configured inside for the port to output DC bias voltage.
✧ **400 Four-port measurement**	For configuring a dual-source stimulus four-port vector network analyzer with a frequency range of 10MHz ~ 26.5GHz.
✧ **401 four-port programmable step attenuator**	4 programmable step attenuators are configured in source channel and receiver channel respectively.
✧ **402 Active intermodulation distortion measurement**	For active intermodulation distortion signal measurement.
✧ **403 Four-port 500Hz LF extension**	For extending the lower limit of the frequency range to 500Hz.
✧ **404 Configurable test device**	For extending the test device of the four-port vector network analyzer with panel jumpers added, with receivers A, B, C, D, R1, R2, R3 and R4 able to be used independently.
✧ **405 Four-port T-bias**	4 T-biases are configured inside for the port to output DC bias voltage.
✧ **S30 Spectrum analysis function**	For providing multi-channel spectrum test capability.

3.1.1.2 Environmental Requirements

The operation sites of 3674 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 3674

Temperature	0°C ~ 40°C
Temperature range during error adjustment	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 3674

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

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

a) Confirming power supply parameters

3674 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 3674

Power supply parameter	Applications
Voltage, frequency	220V/110V adaptive; 50Hz-60Hz
Rated output current	>3A
Power consumption	500W

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

3674 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;

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.

Warning

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-on

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 3674 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 3674 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;

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 Correct Use 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) 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 defect 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

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.

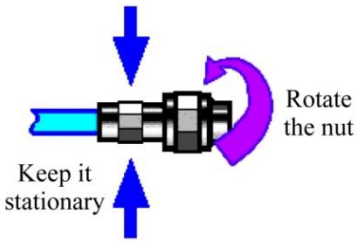


Figure 3.4 Connection method

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.

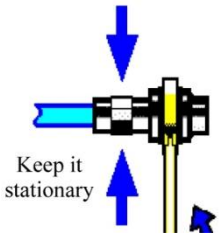


Figure 3.5 Finishing the connection with a torque wrench

4) Disconnection method

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^{o} before applying force.

  .
- 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.

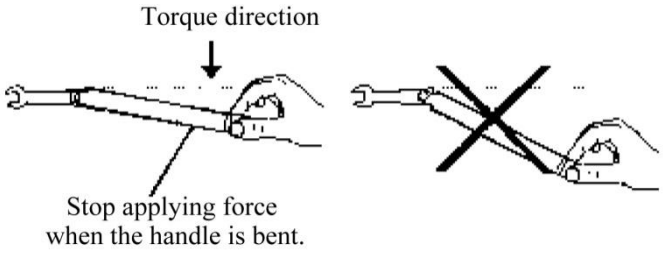


Figure 3.6 Usage of a torque wrench

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

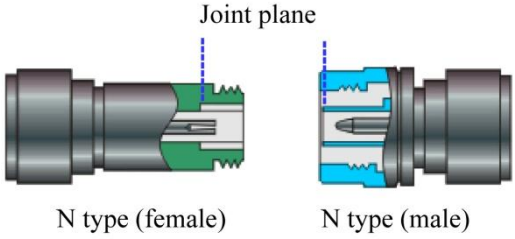


Figure 3.7 Calibration plane

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

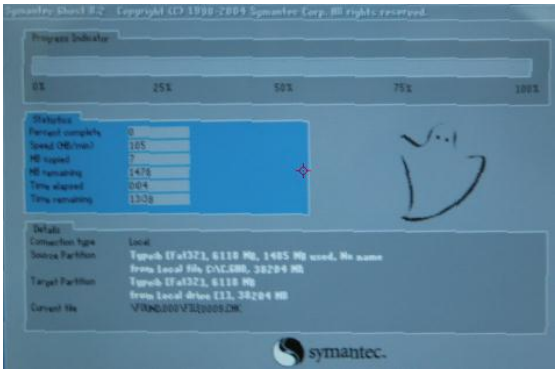


Figure 3.9 System recovery interface

3) 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 Update Installation of Network Analyzer Program

If program update or installation of new program is required for the 3674 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 3.10~3.14.

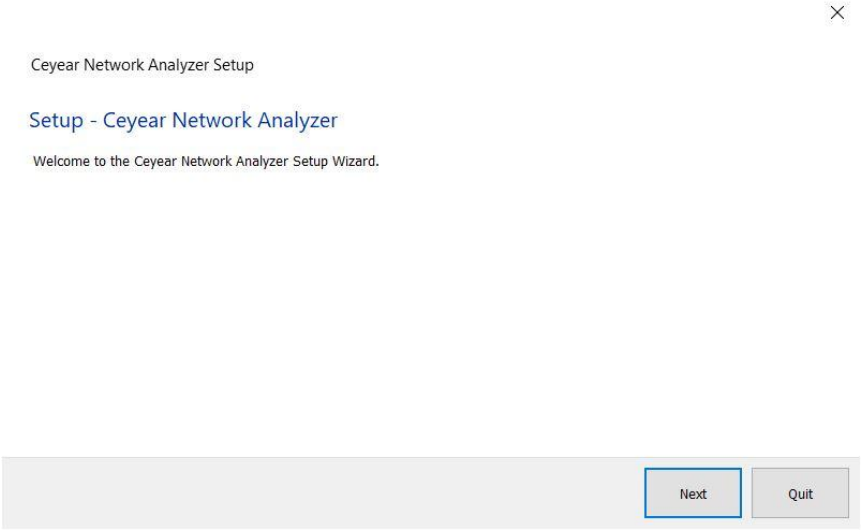


Figure 3.10 Installer boot interface

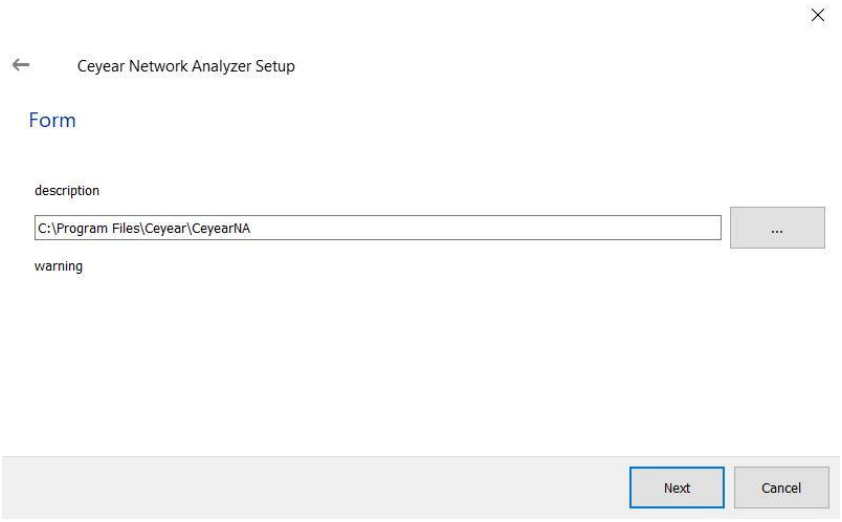


Figure 3.11 Installation directory selection interface

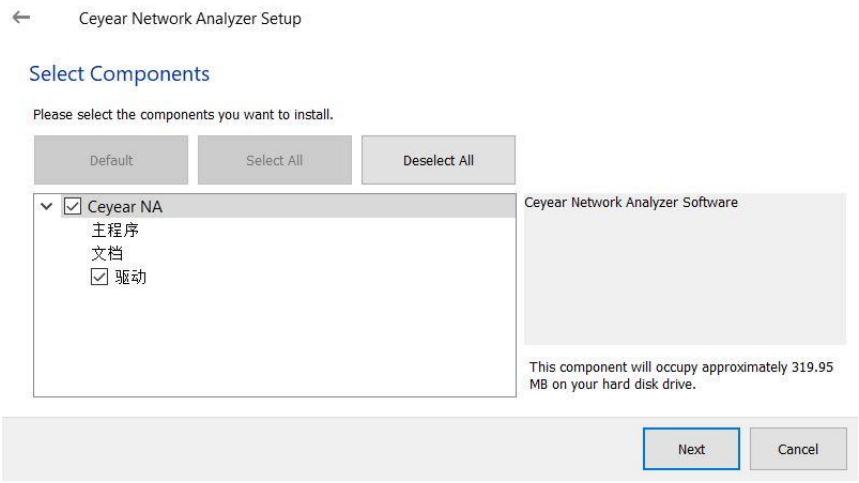
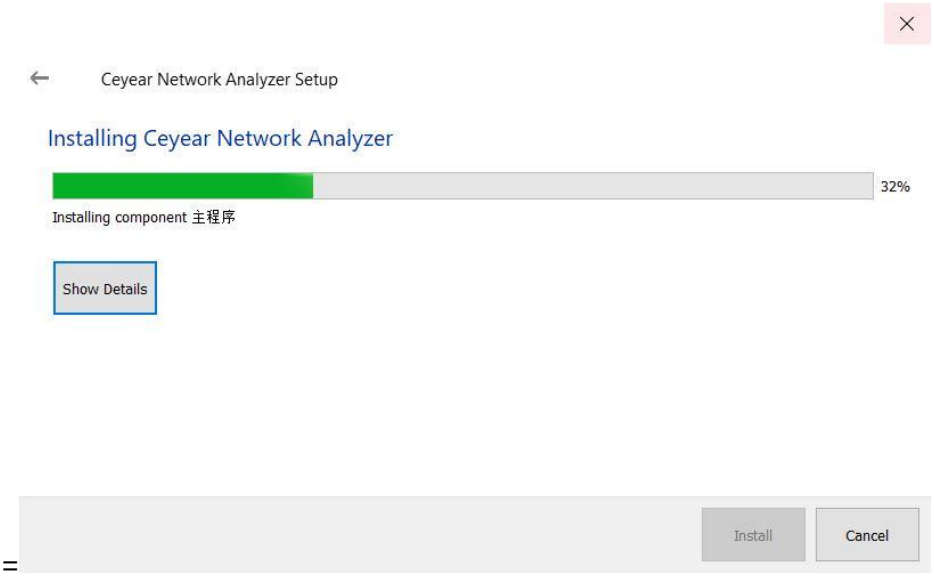


Figure 3.12 Installation type selection interface



=

Figure 3.13 Installation in-process interface

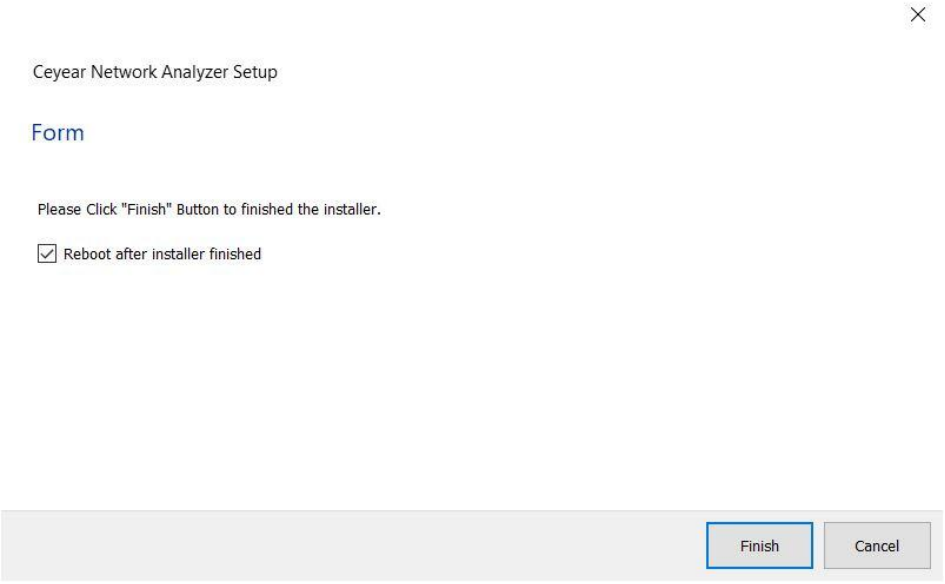


Figure 3.14 Installation completed

3.1.3 Routine Maintenance

This section introduces the routine maintenance of the 3674 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 Front and Real Panels

This section introduces the composition and functions of the front and rear panels as well as operation interface elements of 3674 series vector network analyzers.

- Front Panel..... 50
- Real Panel..... 54

3.2.1 Front Panel

This section introduces the composition and functions of the front panels of 3674 series vector network analyzers. The front panel is shown in Figure 3.17.



Figure 3.17 Front panel of 3674

1) Adjustment keypad

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



Figure 3.18 3674 series 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 six common setting keys, as shown in Figure 3.19.



Figure 3.19 3674 series setting keypad

a)[Menu]

This button is for control the hiding and showing of the button on the software side.

b)[Soft Keyboard]

This button is for calling the soft keyboard of Windows operating system.

c)[Windows]

This button is for calling the start menu of Windows operating system.

d)[System]

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)[Local]

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

f) [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 a keyboard, mouse or other USB devices. Totally 4 USB interfaces are provided on the front panel, which comply with USB3.0 specification.

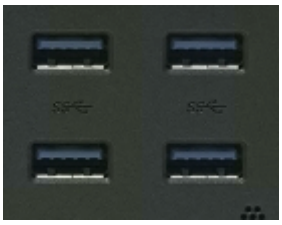


Figure 3.20 USB interface

4) Display screen

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

- 15.6-inch touch screen
- Resolution: 1920 × 1080
- Vertical refresh rate: 60Hz
- Horizontal refresh rate: 48.4kHz

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 on.
- 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.

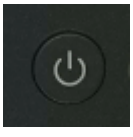


Figure 3.22 [PWR On/SBY] key and indicator

2) Test port

As shown in Figure 3.23, the analyzer is provided with two 3.5mm/2.4mm/1.85mm

(male) ports with a resistance of 50Ω for test, which supports change-over between RF source and receiver so as to make measurements to the DUT in two directions possible. The yellow indicator is used to indicate the source output port.

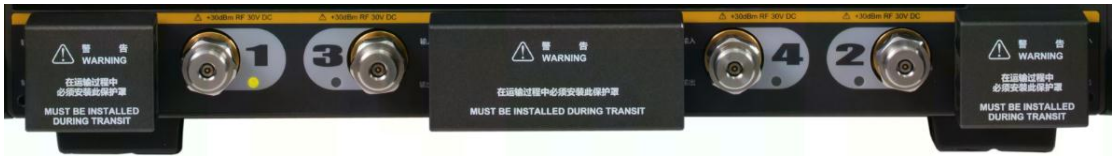


Figure 3.23 Test port of analyzer

3.2.2 Rear Panel Description

This section introduces the composition and functions of the rear panels of 3674 series vector network analyzers. The rear panel is shown in Figure 3.24.



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 the10MHz reference input port are described as follows:

- Input frequency: 10MHz ± 1ppm
- Input level: -15 ~ 20dBm
- Input impedance: 200ohm

b) 10 MHz reference output

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

- Output frequency: 10MHz±1ppm
- Signal type: Sine wave
- Output level: 13dBm±4dB into 50ohm
- Output impedance: 50 ohm

2) GPIB connector

As shown in Figure 3.26, the GPIB connector is a 24-pin D-type female connector that complies with the IEEE-488 standard. It is used to send and receive GPIB/SCPI commands.

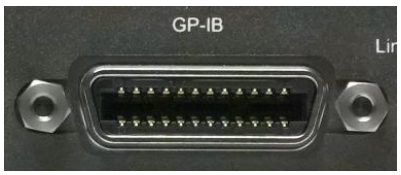


Figure 3.26 GPIB connector

3) LAN connector

As shown in Figure 3.27, this is a 10/100/1000BaseT Ethernet connector with a standard 8-pin structure that allows for automatic selection between three data rates.



Figure 3.27 LAN connector

4) USB connector

As shown in Figure 3.28, 4 USB connectors of the following two types are provided on the rear panel, namely type-A (4 contacts embedded, with contact 1 on the left) and type-B. The type-A USB connector supports connection with USB mouse, keyboard, or other devices with USB interface. The port features are as follows:

- Contact1: Vcc,4.75Vto5.25V, max. 900mA
- Contact 2: Data-
- Contact 3: Data+
- Contact 4: Earthing

The type-B USB connector is a reserved connector mainly for control, and through this connector, the network analyzer can be controlled via SCPI to achieve interconnection with external computers or remote devices.

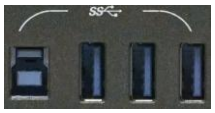


Figure 3.28 USB connectors (type-B on the left, type-A on the right)

5) Display port (DP) output connector

The DP connector, as shown in Figure 3.29, 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 3.29 Display port (DP) output connector

6) LO and RF output connector

As shown in Figure 3.30, the LO output interface and the RO output interface are 3.5mm female interfaces, through which the internal LO signal and the source RF signal are output respectively for fault diagnosis and MMW spread spectrum when necessary. The port features are as follows:

- LO output signal frequency range:
  12.535MHz～13.507606GHz/26.507606GHz
- LO output signal power range: -4dBm ~ 6dBm
- RF output signal frequency range: 3.2GHz ~ 13.5GHz/26.5GHz
- RF output signal power: 2dB-3dBm±



Figure 3.30 LO and RF output connectors

7) IM ON/OFF input/output connector

Port 2/port 4 stimulus output:

Frequency range: 10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz

Power range: according to the power setting of the network analyzer.

Port 2/port 4 test input:

Frequency range: 10MHz~9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz

Maximum input power: 20dBm.



Figure 3.31 IM ON/OFF input/output connector

8) Pulse input/output connector

As shown in Figure 3.32, the pulse input/output connector is a 15-pin D-type female connector, through which the monitoring to the internal pulse generator working state can be synchronized, and the external pulse generator can be used to provide pulse control signal for the internal pulse modulator and IF gate.

The electrical characteristics of this connector are as follows:

- Pulse width range: 33ns~60s
- Pulse transition time: 30ns
- Input impedance: Ohm 1K
- Input: DC <5.5V
- Driving voltage: 0V(off)3.3V(on)

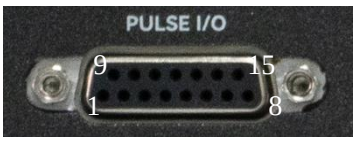


Figure 3.32 Pulse input/output connector

The pins in the connector are defined as follows:

Table 3.5 Definition of pins in pulse input/output connector

Pin	Function description
1 ~ 5	IF pulse gate input A/B/C/D/R(TTL)
6, 9, 15	Reference ground
7	Pulse sync trigger input
8	Source pulse modulation drive input
10 ~ 13	Pulse inputs 1-4 (TTL)
14	Empty

9) External IF input connector

As shown in Figure 3.33, 5 SMA interfaces are provided for external IF input of the vector network analyzer. For two-port model, the interfaces are identified as A, B, R1, R2,

and for four-port model, the interfaces are identified as A, B, C, D, R. The characteristics are as follows:

- Input impedance: 50ohm
- RF input: < 23dBm
- DC input: < 5.5V
- 0.1dB compression point: -9dBm



Figure 3.33 External IF input connector

10) Auto test port connector

As shown in Figure 3.34, the interface is a 36-pin female connector, through which the network analyzer and the material handler interact signals to provide users with a stable and reliable auto test environment.



Figure 3.34 Auto test interface connector

The port consists of TTL arrays, and the electrical characteristics are as follows:

a) Input voltage range: -0.5V ~ 5.5V

TTL high level: 2.0V ~ 5.0V

TTL low level: 0V ~ 0.5V

b) Output current/voltage range: -10mA ~ 10mA

Output current:

TTL high level: -5mA

TTL low level: 3mA

Output voltage:

TTL high level: 2.0V ~ 3.3V

TTL low level: 0V ~ 0.8V

The pins in the connector are defined as follows:

Table 3.6 Definition of pins of auto test interface connector

Pin	Function description
1 (lower left corner)	Ground line
2	Input 1
3, 4	Input 1,2
5 ~ 12	Output port A0-A7
13 ~ 17, 19	Output port B0-B5
22 ~ 25	Input/output port C0-C3
26 ~ 29	Input/output port D0-D3

18	External trigger
20	Output port B6/index signal
21	Output port B7/Trigger ready
30, 31	Port C/D status
32	Output port write confirmation
33	Pass/Fail state information
34	Sweep end
35, 36	5V, Pass/Fail state information confirmation

11) Expansion interface connector

As shown in Figure 3.35, this interface is a 9-pin D-type female connector.

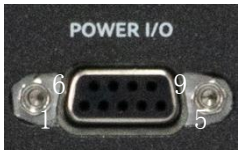


Figure 3.35 Expansion interface connector

The pins in the connector are defined as follows:

Table 3.7 Definition of pins in expansion interface connector

Pin	Name	Function description
1, 2 (lower left)	+15V, -15V	15V@400mA
3, 4	Analog output ports 1 and 2	Control via internal setting dialog box +/-10V@100mA output Resolution 2.44mV
5	ACOM	System ground wire
6	GndSence	Analog input/output test ground
7, 8	Analog input ports 1 and 2	mailto:Analog%20input%20+/-10V@1.22mVAnalog input +/-10V@1.22mV resolution
9	Power switch	Power button input

12) External test device interface connector

As shown in Figure 3.36, the test device interface is a DB-25 female interface with 13 address and data multiplexing lines, 3 control lines and 1 control interruption line, which, through the control logic, achieves control to the external test device (such as external spread spectrum controller).

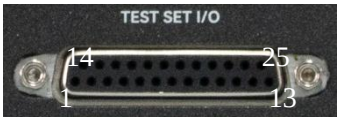


Figure 3.36 External test device interface connector

Table 3.8 Definition of pins in external test device interface connector

Pin	Name	Function description
1,15,16,18 (pin 1 at left lower part)	Test device address selection bit	Connected to ground. The value of bit is initially LOW, and when an external test device is connected, the value increases by 1
2	Sweep waiting	Trigger method: program-control sweep switch, with delay mode introduced
3 ~ 6, 9 ~ 11, 17, 19 ~ 23	Address bit AD0-AD12	For outputting data address, or receiving/sending data information
7, 12	Ground line	Reference ground wire
8	Address switch LAS	LOW indicates that the currently transmitted information is address
24	Data switch LDS	LOW indicates that the currently transmitted information is data
25	Read/write indicator RLW	Low level output, high level data input
13	Input interrupt	Programmable reading of TTL input state
14	NC	Maximum range： +22V 100mA

13) Trigger input/output interface connector

As shown in Figure 3.37, 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 3.37 Trigger input/output connector

14) 28V(BNC) interface connector

As shown in Figure 3.38, the BNC (female) connector can be used to drive a noise

source.



Figure 3.38 28V (BNC) interface connector

3.3 Analyzer Software Interface

As shown in Figure 3.39, 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, setting of color and display of scale.

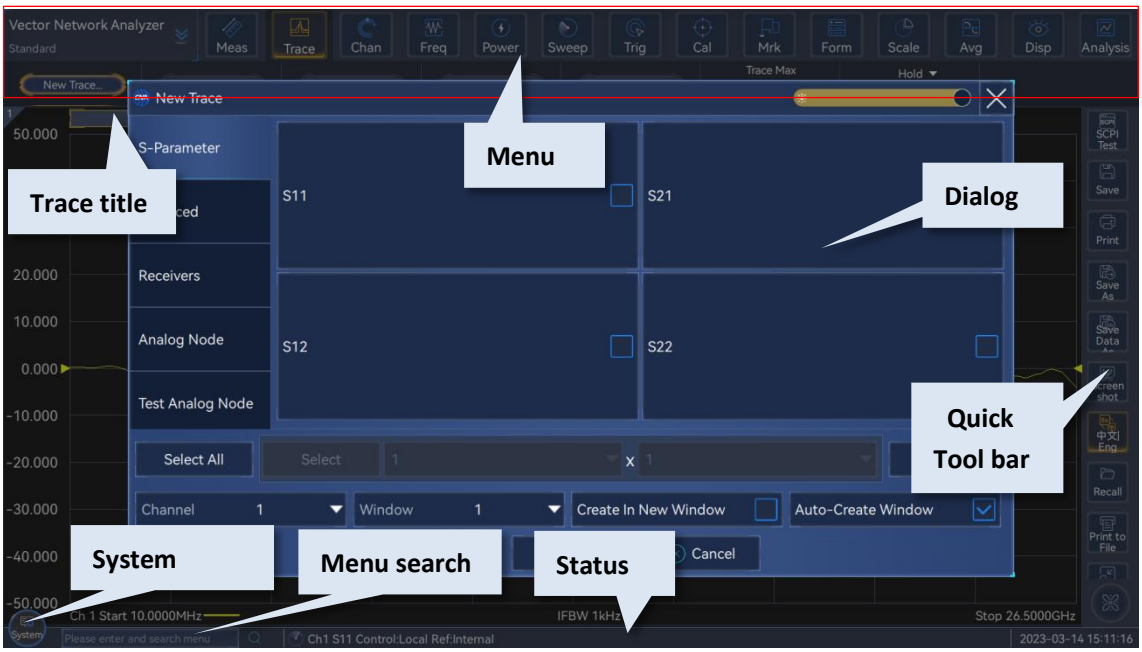


Figure 3.39 Display screen of the analyzer

3.4 Analyzer Traces, Channels and Windows

- Trace.....62
- Channel.....59
- Window.....63

3.4.1 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. 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
- Marker
- Electrical delay
- Phase Offset
- Smoothing
- Time Domain Conversion

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]**.

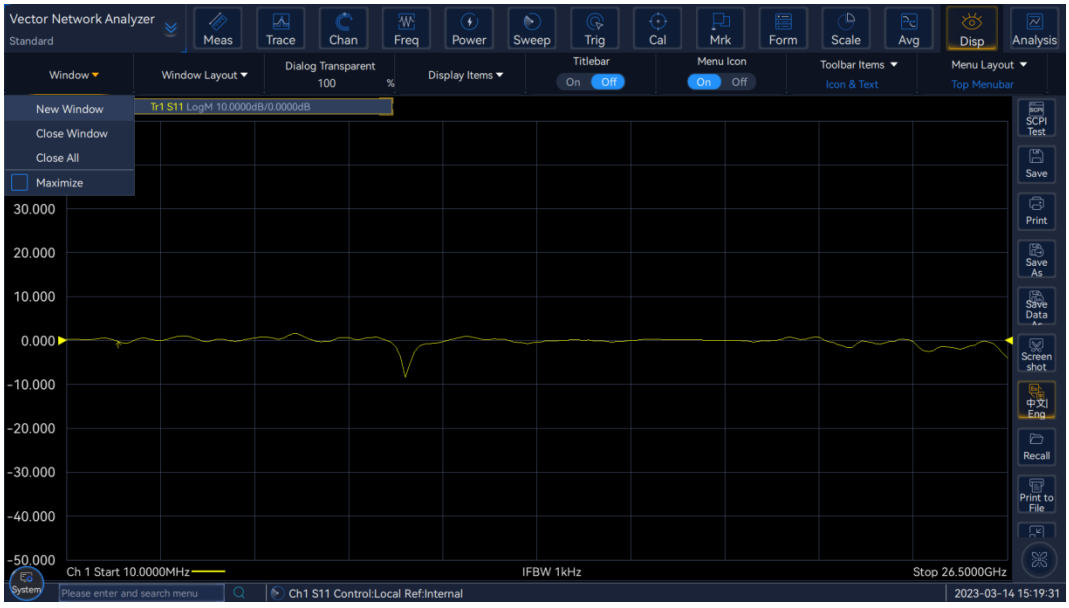


Figure 3.40 Creating a new window

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

- Marker.....64
- Trace Operation and Statistics.....73
- Limit Test.....76
- Ripple Test.....79
- BW Test.....81
- Formula Editor.....84

3.5.1 Marker

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

- Create a Marker.....64
- Move a Marker.....63
- Marker Search.....66
- Marker Functions 70
- Marker Setting.....71
- Marker Table.....72

3.5.1.1 Create a Marker

1) Marker menu

Menu path: click **[Mrk]** to display the **marker** toolbar.

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

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

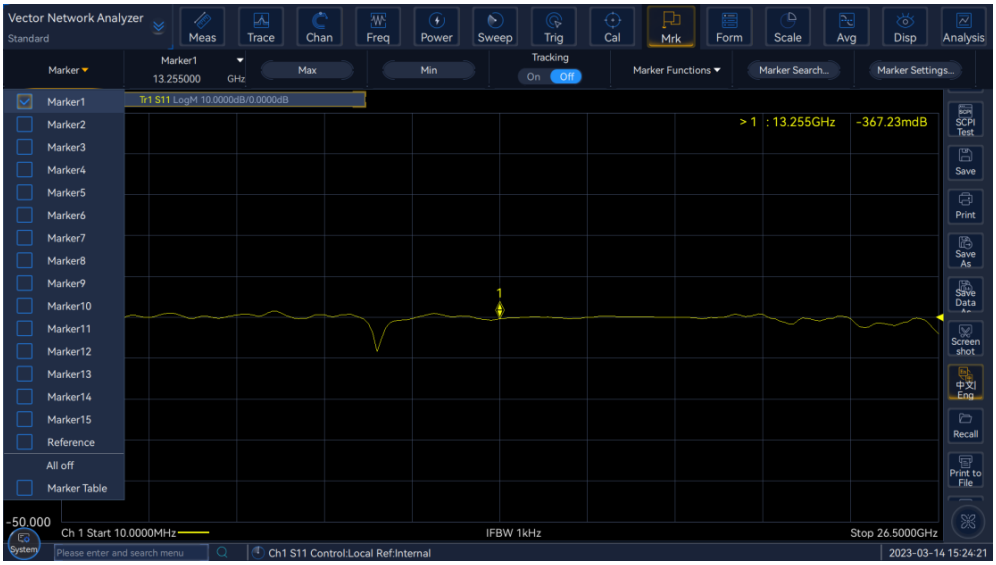


Figure 3.41 Creating a marker using the Marker menu

2) Marker toolbar

a) [Mrk] box

Select the marker you want to define.

b) [Mark n] box

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

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 Functions] 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 Marker

Move the marker using one of the following methods:

a) Click [Marker n] box in the **marker toolbar**, and move a marker as follows:

i. Enter the value to be marked by the marker directly.

ii. Turn the knob to move the marker.

iii. Click the knob button in the **[Mark n]** box to change the marker stimulus value.

iv. Press **[↑]** or**[↓]** in the **[Mark n]** key area to change the marker stimulus value.

3.5.1.3 Marker Search

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

1) Method to search for marker

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

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

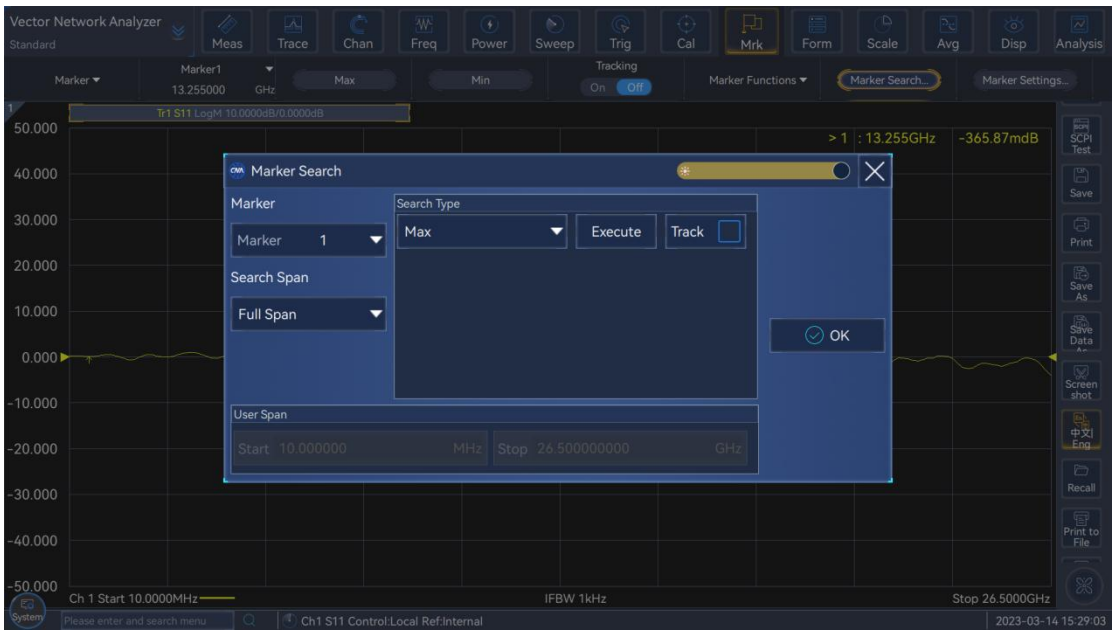


Figure 3.42 Marker search

2) Marker search dialog box

a) Marker zone

Select the marker for search definition.

b) Search domain zone

Define the marker search range. The default search domain is Full Bandwidth. In addition, 16 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 markers can use the same search domain. Figure 3.43 shows a set of user-set search domains:

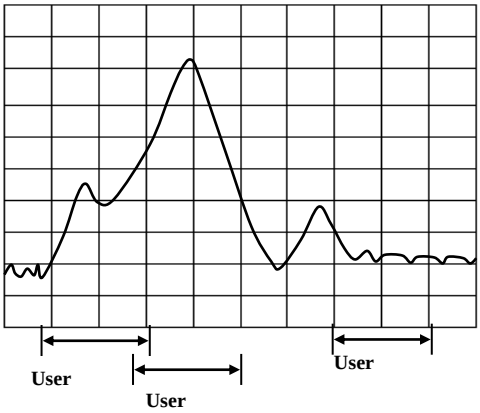


Figure 3.43 Schematic diagram of user-set marker search domain

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 marker 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 marker magnitude, provided the peak has been defined.
- **Right Peak** Search for the next valid peak to the right of the marker 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 3.44, 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;
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;

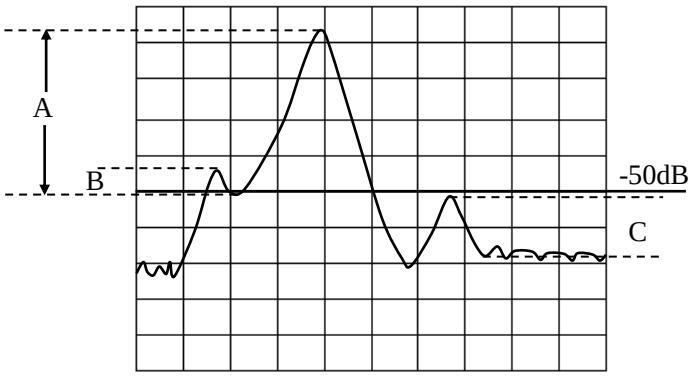


Figure 3.44 Schematic diagram of the peaks in the marker domain

- **Left Peak** Search for the next valid peak to the left of the marker position, provided the peak has been defined.
- **Target**: Enter the search target value in the **[Logarithmic Value]** box, click the **[Execute]** button, the marker moves to the first target value to the right of the current marker position, then click the **[Execute]** button consecutively, the marker 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 marker 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 marker moves to the interpolated data point equal to the target value.

- **Left Target** searches only the target value to the left of the current marker.
- **Right Target** searches only the target value to the right side of the current marker.
- **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 3.45. Measurements are made using Marker 1 to Marker 4. Marker 1 searches for the maximum peak, Marker 2 searches for the point where the left side of the peak drops to the specified level, Marker 3 searches for the point where the right side of the peak drops to the specified level, and Marker 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 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.

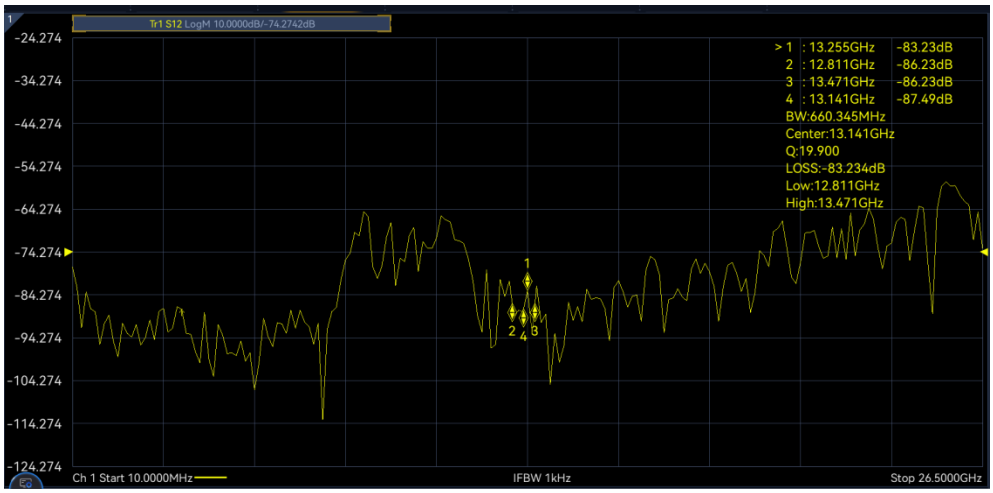


Figure 3.45 Bandwidth search

- **Notch Bandwidth** The analyzer tests the notch bandwidth of the bandpass filter, as shown in Figure 3.46.

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

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



Figure 3.46 Notch bandwidth

e) [OK] button

Click **[OK]** to perform a marker 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 marker is in the desired position after each sweep.

3.5.1.4 Marker Functions

1) Steps to use marker functions

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

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

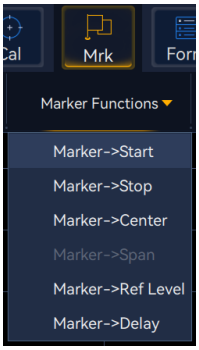


Figure 3.47 Setting a measurement using the Marker Function checkbox

2) Marker functions dialog box

a) [Marker-> Start] button

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

b) [Marker -> Stop] button

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

c) [Marker -> Center] button

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

d) [Marker -> Ref Level] button

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

e) [Marker -> Delay] button

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

f) [Marker -> Span] button

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

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 3.48 Marker Settings dialog box

2) Marker Display box

a) [Marker Readout]

The corresponding marker value is displayed on the screen when it is checked, and the marker 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] checkbox

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

c) [△ Marker] check box

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

d) [Discrete] check box

When it is checked, the marker 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) [Form] box

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

f) [Fixed]

When checking fixed marker, according to the position on the trace when it is set as a fixed type marker, the marker 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 marker stimulus value, but the Y-axis coordinates remain unchanged. This type of marker is mainly used to observe changes in the trace data. For example, a fixed marker 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 marker. If it has a fixed X-axis position, and the Y-axis position changes with the amplitude of the trace data, the marker position on the X-axis can be moved left and right by changing the marker stimulus value.

g) Coupled Marker

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

3.5.1.6 Marker Table

1) Marker table

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

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

2 Opening/closing method

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

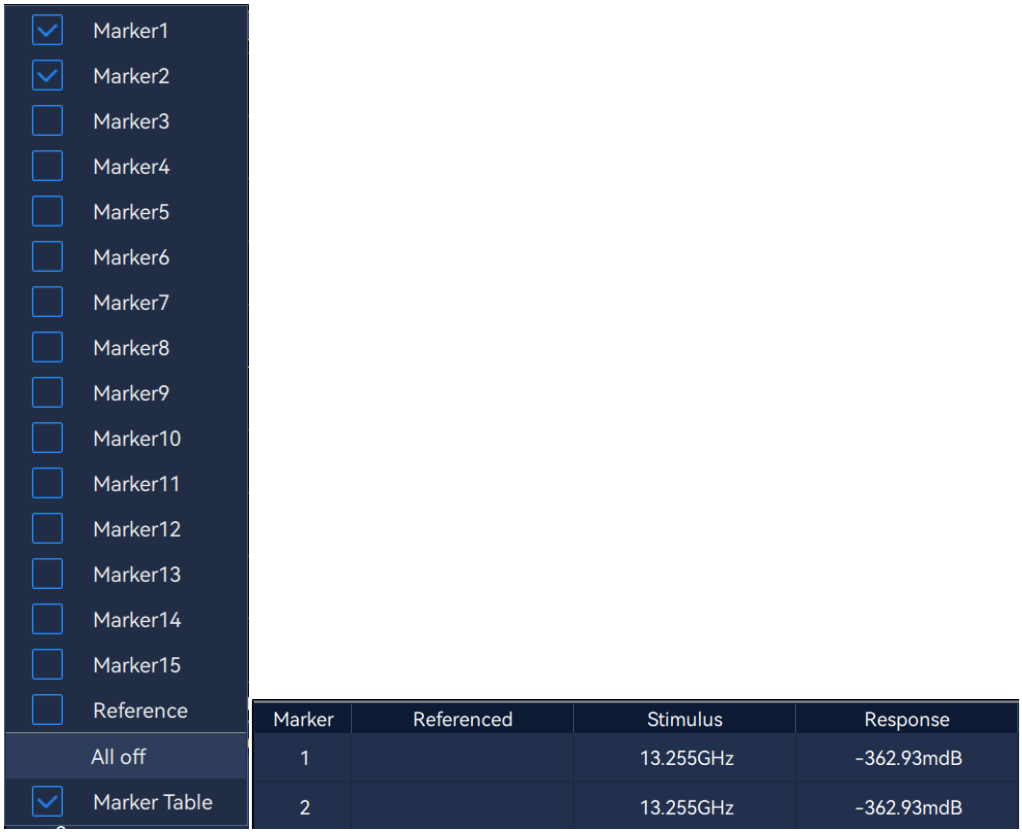


Figure 3.49 Setting Marker Table display via Marker menu

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.....73
- Trace Statistics.....74

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.

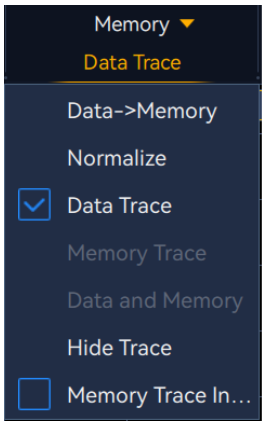


Figure 3.50 Setting trace operation

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 marker 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 marker 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

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.

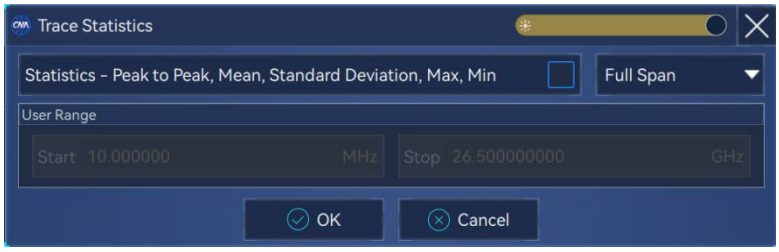


Figure3.51 Trace Statistics dialog box

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 setting of trace statistics. You can select full bandwidth or customize 9 user settings.

c) User Range

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 + Memory

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.....76
- Limit Table.....77
- Set Limit Test.....77

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.

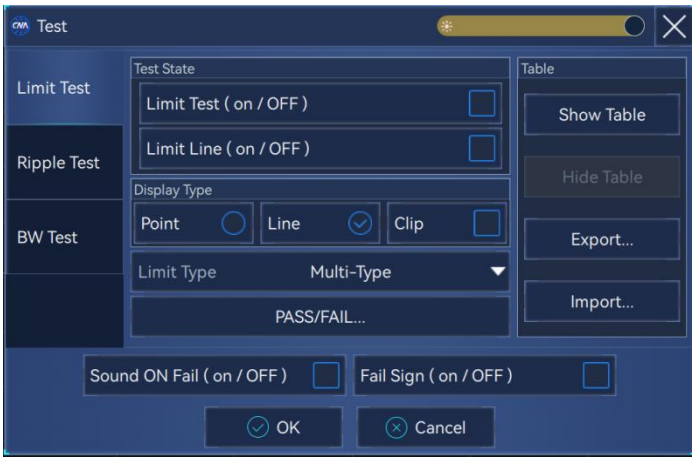


Figure 3.52 Limit test

3.5.3.2 Limit Table

	Type	Start Stimulus	Stop Stimulus	Start Response	Stop Response	Limit Type
1	OFF	10.000 MHz	26.500 GHz	-100.000 dB	100.000 dB	SLOP

Figure 3.53 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 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 are 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)] checkbox

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)] checkbox

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] checkbox

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

[Fail flag (ON/off)] checkbox

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 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.

- Create and Edit Ripple Limit Lines.....76
- Set Ripple Test.....77
- Show Ripple Test Results.....77

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

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

b) click **[Show Table]** button in the dialog box to open the ripple limit table.

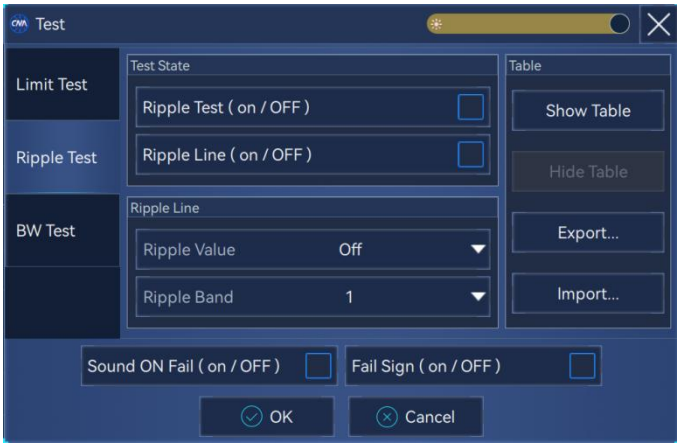


Figure 3.54 Display Ripple Test dialog box

2) Ripple limit table

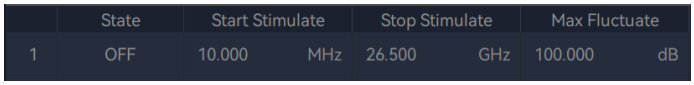


Figure 3.55 Ripple Limit table

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 are 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)] checkbox

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)] checkbox

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)] checkbox

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

[Fail flag (On/OFF)] checkbox

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

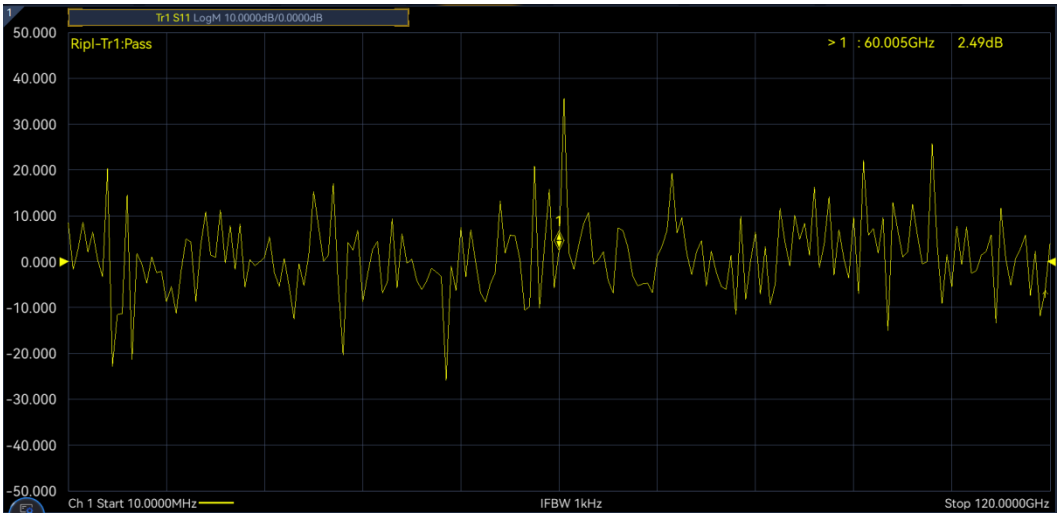


Figure 3.56 Ripple test result

3.5.5 BW Test

The BW test function is mainly used to test the bandwidth of the bandpass filter.

The BW 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 BW 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.

- Turn on and Set BW Test.....76
- Display BW Test Results.....77
- Turn off BW test.....77

3.5.5.1 Turn on and Set BW Test

Before performing the BW 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)] checkbox in the dialog box to turn on the BW test;
3) Check the [BWV Disp (on/OFF)] checkbox in the dialog box to display the measured bandwidth value on the screen;
4) Check the [BWM Disp (on/OFF)] checkbox 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)] checkbox in the dialog box to provide an audible alert in the event of a BW test failure;
9) Check the [Fail flag (on/OFF)] checkbox in the dialog box to display the FALL prompt in case of a BW test failure;
10) Click [OK] to complete the BW test settings.

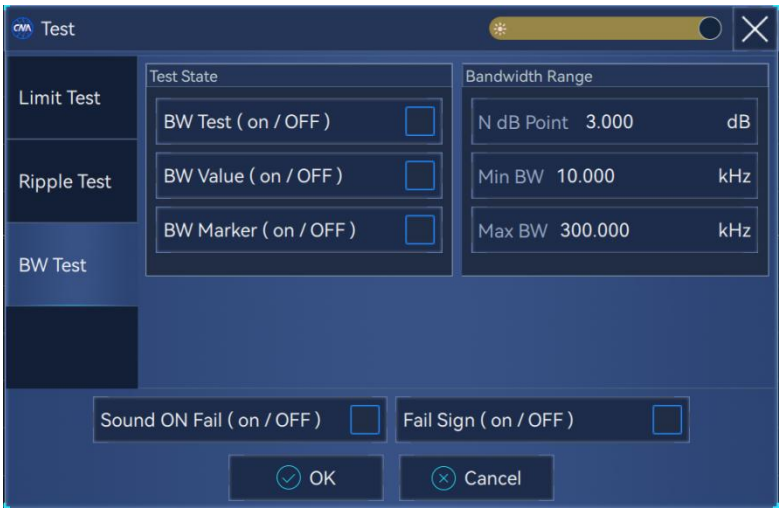


Figure 3.57 BW test dialog box

3.5.5.2 Displaying BW Test Results

BW 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.

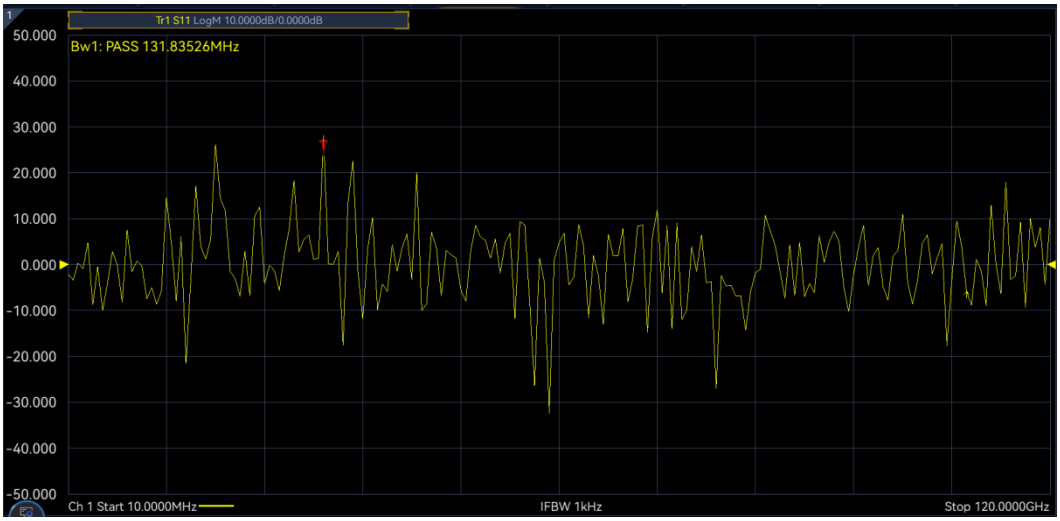


Figure 3.58 BW Test display results

3.5.5.3 Turn off BW Test

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

bandwidth locator mark display;

5) Click [OK] to turn off the BW test.

3.5.6 Formula Editor

- Overview.....84
- How to Use Formula Editor.....85
- Data Used by Formula Editor and Precautions.....89
- Save Formula Editor Data.....90

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 S21/(1-S11), the data for each point on the formula trace calculated by this expression is obtained by dividing the S21 data of the corresponding point by 1 minus the data of S11. 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: S21+S23-S13. 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 S21*S23/S13 in order to achieve the purpose, as shown in Figure 3.59; when the formula is entered, it is displayed as Eq=S12*S23/S13 on the currently active trace of the vector network analyzer, as shown in Figure 3.60.

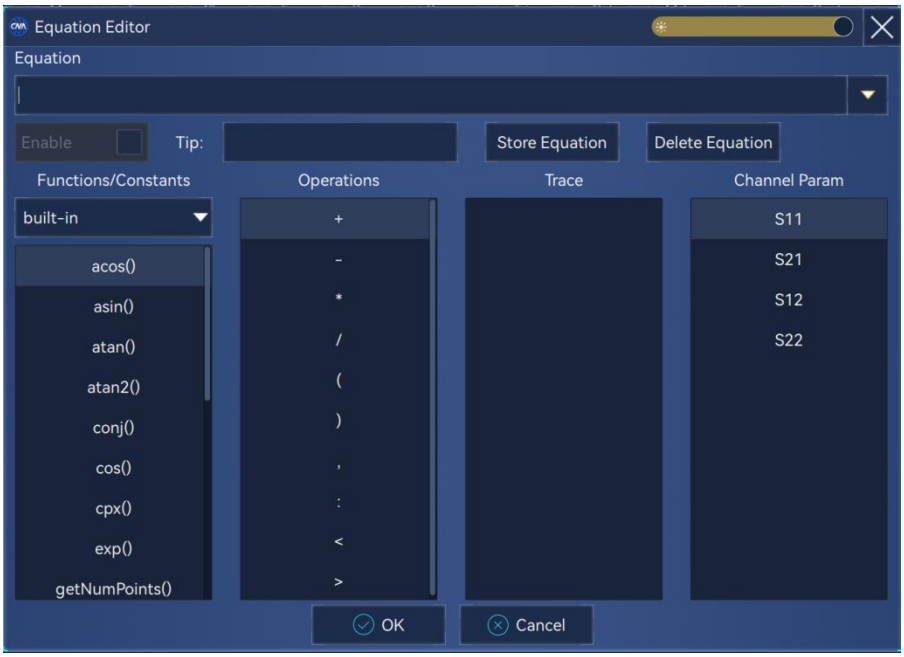


Figure 3.59 Formula Editor dialog box

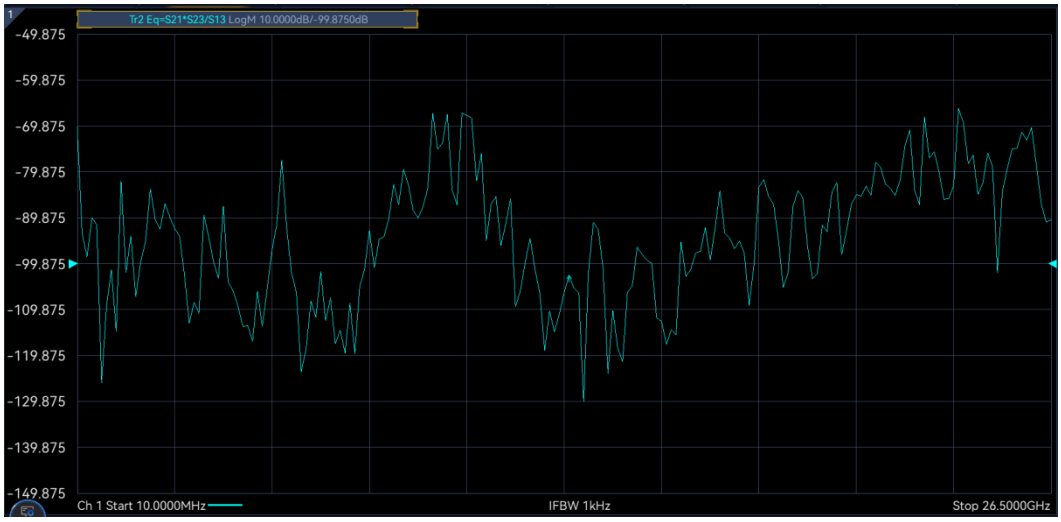


Figure 3.60 Display of formula trace

3.5.6.2 How 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) How to enter a formula in the Formula Editor dialog box

a) Input 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 operators

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

c) Input 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.

i Data input area

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

3) Activate checkbox

The default state of the active checkbox 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 checkbox is unchecked, no calculation will be performed regardless of whether the entered expression is computable or not. In addition, if the Activate checkbox is checked, but the formula entered is not computable (e.g., enter "2+1+"), the checkbox will change to a gray button.

4) Save and delete 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 marker. If there is no character on the left, the marker position remains unchanged.

6) Left-shift marker and Right-shift marker

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

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

character on the right, the marker position will not change.

7) Function/constant selection

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 3.10.. 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 3.10 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 mu1 = (1 - |a|^2) / (|d - 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 mu2 = (1 - |d|^2) / (|a - 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) Operators selection

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 3.11 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
:	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) Digital 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 3.61.

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.

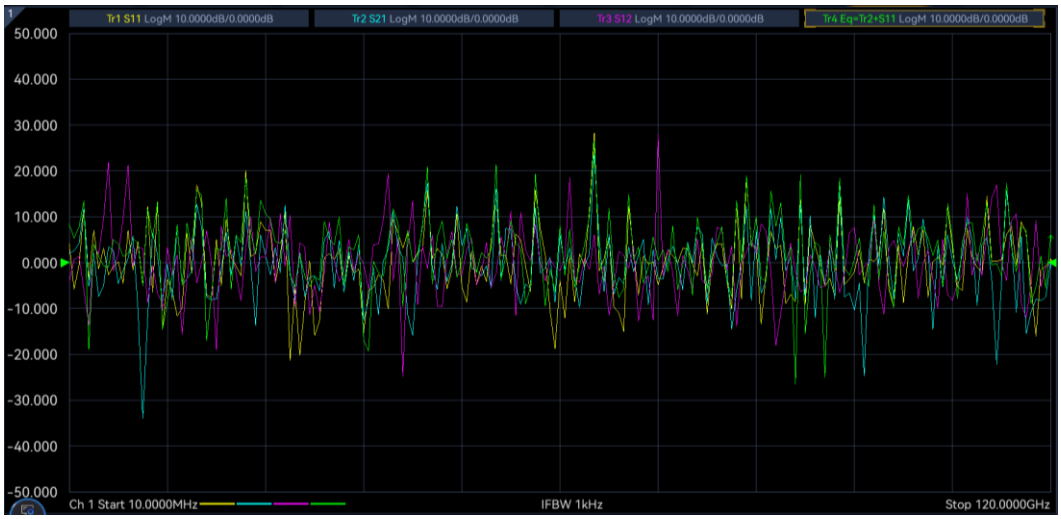


Figure 3.61 Formula Trace display window

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……90
- Printer Displaying Measurement Results……90

3.6.1 Save and CallbackFiles

The 3674 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...]**.

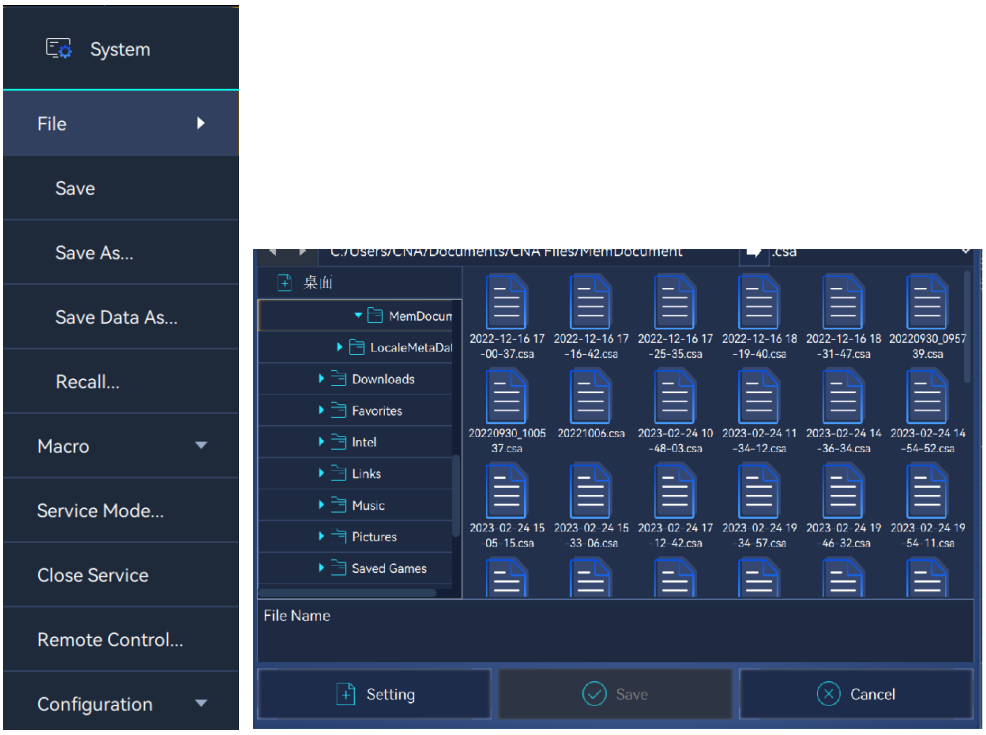


Figure 3.62 Save files

2) [Save] file menu item

When **[Save]** is clicked, the analyzer saves the instrument status and calibration data to the default file (defaut.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...] File 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

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 markers.

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 setting of the instrument. Therefore, it is necessary to ensure that the status setting 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) How to call back files:

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

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.

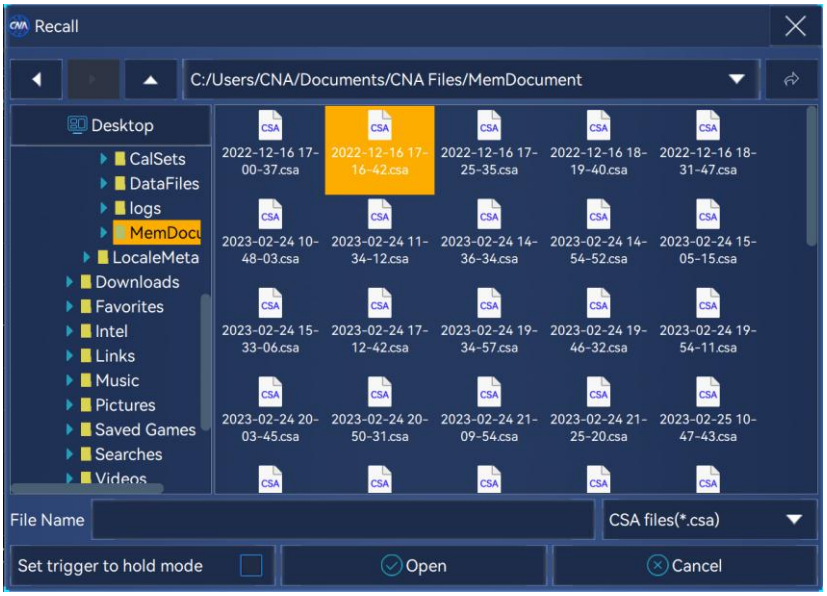


Figure 3.64 Callback file

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.

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.

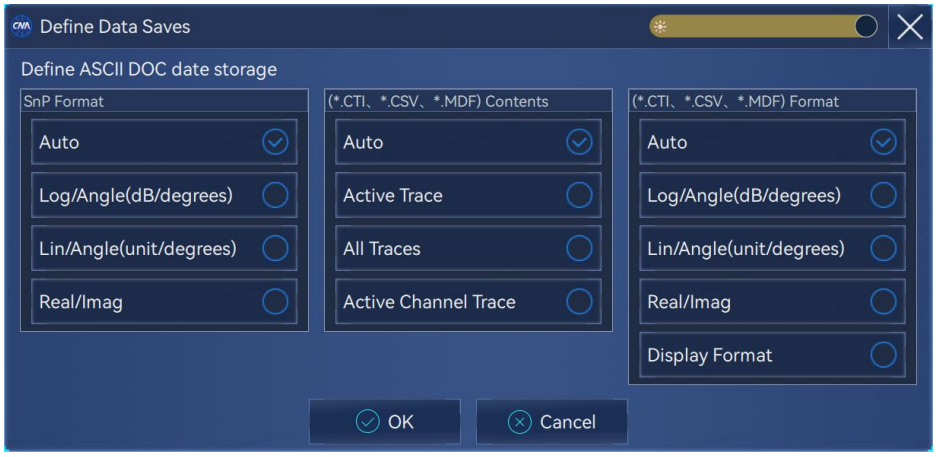


Figure 3.65 Data Save Settings dialog box

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 Format Area

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 (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 **Save** 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
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. It may be a parallel port, serial port or USB interface printer. It can be used as long as the printer is added via the Windows operating system.

3.6.2.1 Print Content Settings

How to set print content

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

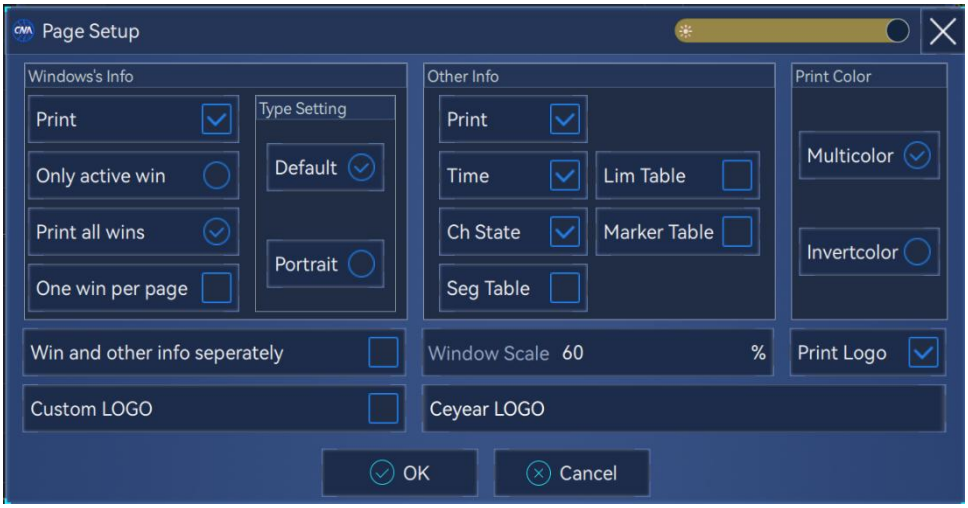


Figure 3.66 Setting print content

Page Settings dialog box

1) Window information zone

a) [PRT] checkbox

Print window trace information when it is checked, and does not print when it is unchecked. The following two checkboxes are only valid if this checkbox is selected.

b) [PRT ONE Pg] checkbox

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) [PRT] checkbox

Print channel setting information when selected. The checkbox 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] checkbox

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.

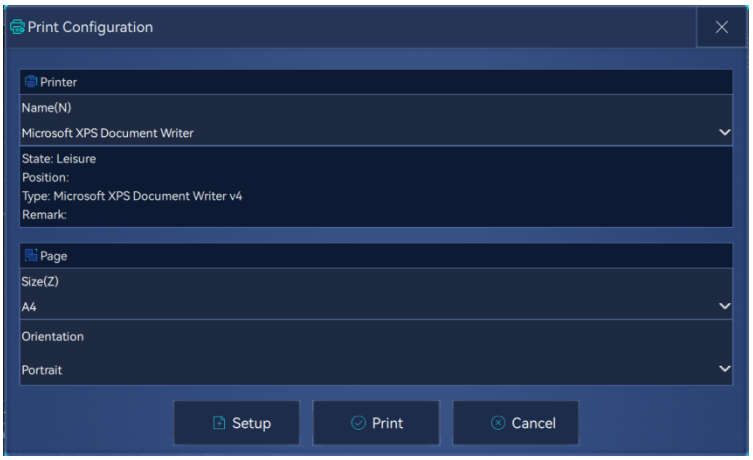


Figure 3.67 Printing measurement information

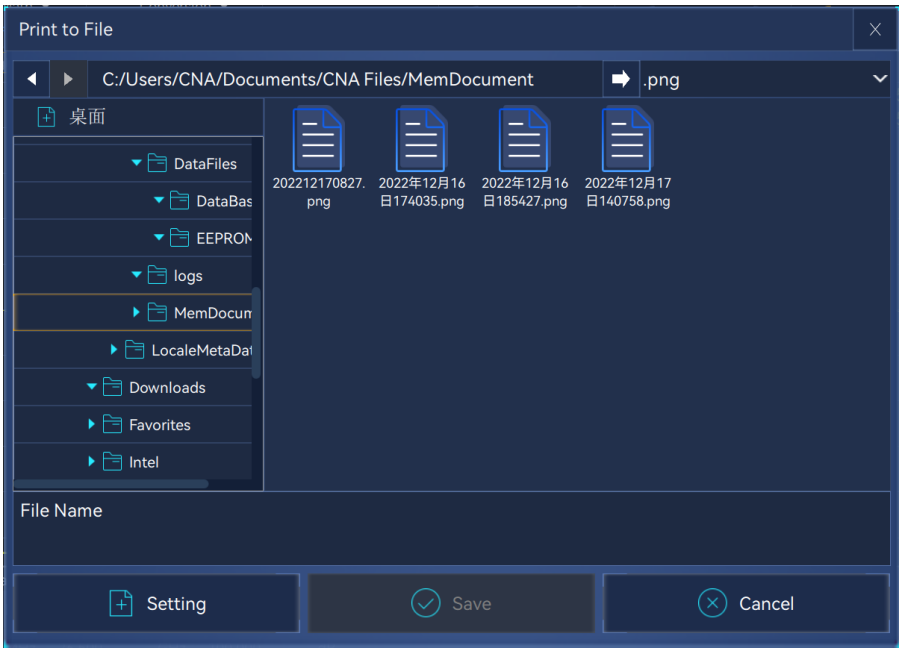
3.6.2.3 Print to File

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.



3.68 Print to File

4 Measurement Settings

This chapter introduces the operation of the 3674 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.....101
- Select measurement parameters.....105
- Set frequency range.....113
- Set the power level of signal.....114
- Set the sweep 117
- Trigger mode.....123
- Set data format and scale.....129
- Observe multiple traces and open multiple channels.....134
- Set analyzer display 136

4.1 Reset analyzer

- Default reset status.....101
- User reset state.....104
- Reset Analyzer.....105

4.1.1 Default reset state

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 parameter: S11

2) Frequency setting:

a) Start frequency: 10MHz
b) Stop frequency: 13.5GHz/26.5GHz/43.5GHz/50GHz/67GHz
c) Point frequency: 2 GHz

3) Power setting:

a) Test port power: -5dBm
b) Port Powers Coupled: ON
c) Attenuation: auto attenuation
d) Attenuation value: 0 dB
e) Power slope: off
f) Slope: 0dB/GHz

4) Sweep setting:

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

5) Sweep segment setting:

a) Number of open segments: 1
b) Start frequency: 10MHz
c) Stop frequency: 13.5GHz/26.5GHz/43.5GHz/50GHz/67GHz
d) Points: 21
e) Power: -5dBm
f) IF bandwidth: 1kHz

6) Trigger setting

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 setting for 3674 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) Cal setting:

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: 0 s

10) Marker setting:

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

11) Limit test setting

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

12) Limit table settings:

a) Type: OFF
b) Start stimulus: 10MHz
c) Stop stimulus: 13.5GHz/26.5GHz/43.5GHz/50GHz/67GHz
d) Start response: -100dB
e) End response: 100dB

13) Time domain conversion settings (option):

a) Time domain conversion: OFF
b) Transform mode: bandpass
c) Start of transformation: -5ns
d) Transform stop: 5ns
e) Kaiser window beta factor: 6.0
f) Gate state: closed

g) Start of gate: -5ns
h) Gate stop: 5ns
i) Gate shape: standard
j) Gate type: bandpass

14) Global display settings:

a) Trace state: ON
b) Frequency/Stimulus: ON
c) Marker 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] checkbox.

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.

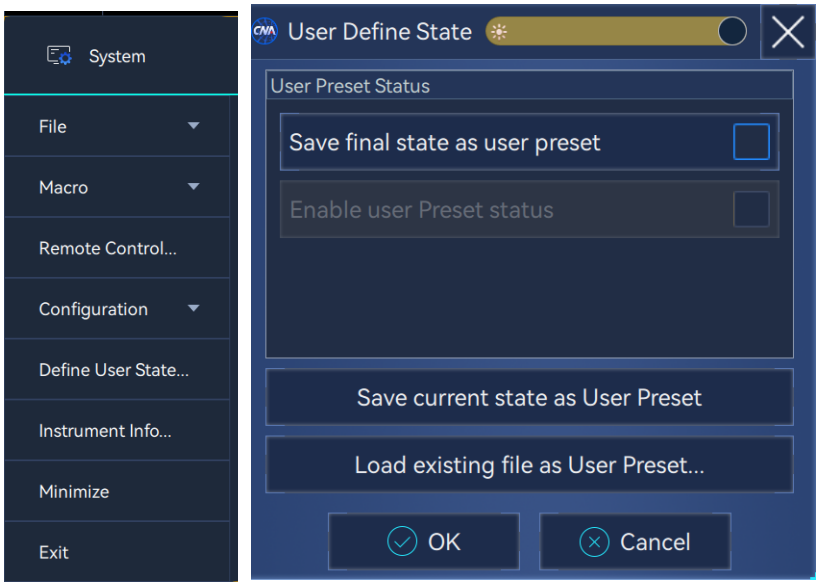


Figure 4.1 Define user resetting state

4.1.3 Reset analyzer

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

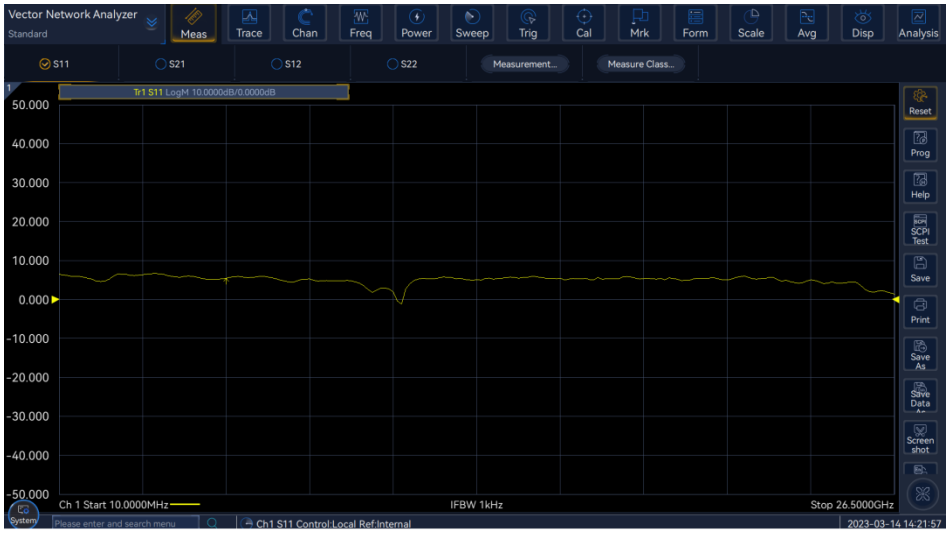


Figure 4.2 Reset analyzer

Tips

【Reset】 shortcut key

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

4.2 Measurement Parameter Selection

The 3674 series vector network analyzer can be set up to measure the electrical 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.....105
- Differential Balance S-parameter.....111
- Arbitrary Ratio.....112
- Non-ratio Power Measurement.....113
- Changing Trace Measurement Type.....113

4.2.1 S-parameter

1) Overview of S parameter

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: $S_{output\ input}$. 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-parameter

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

a) Reflection measurement: SXX (X=1,2,3,4)

i. Echo 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
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:

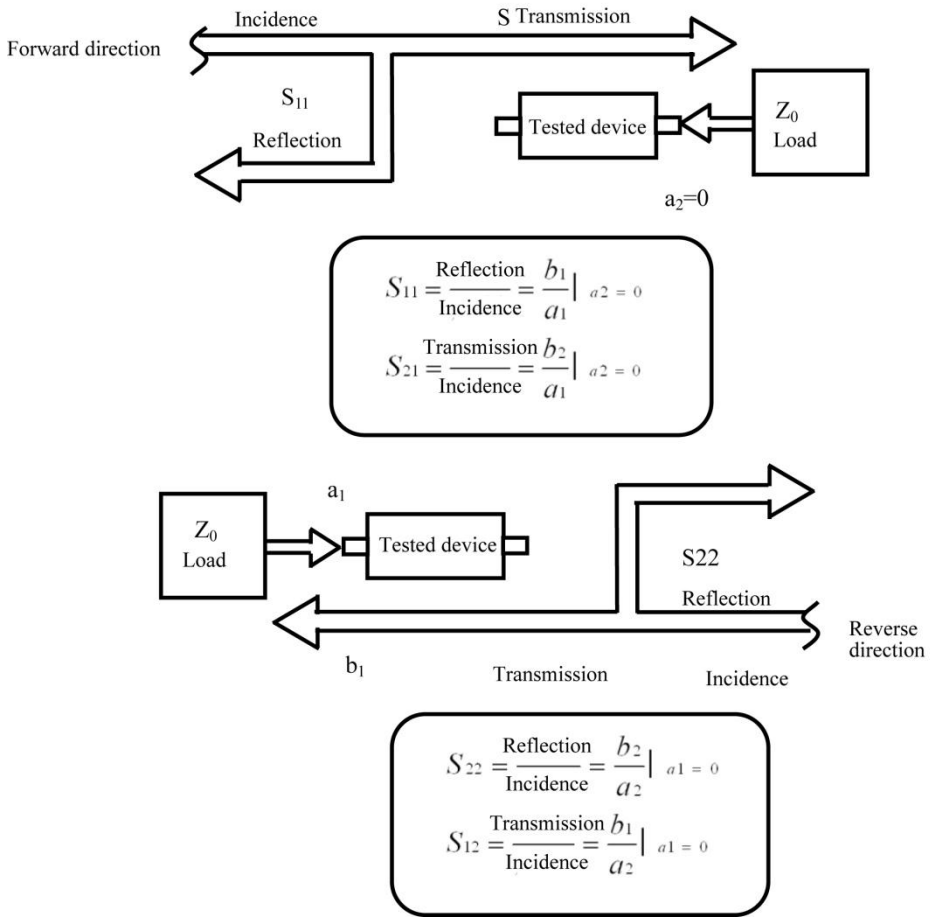


Figure 4.3 S-parameter definition

3) New S-parameter measurement trace

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

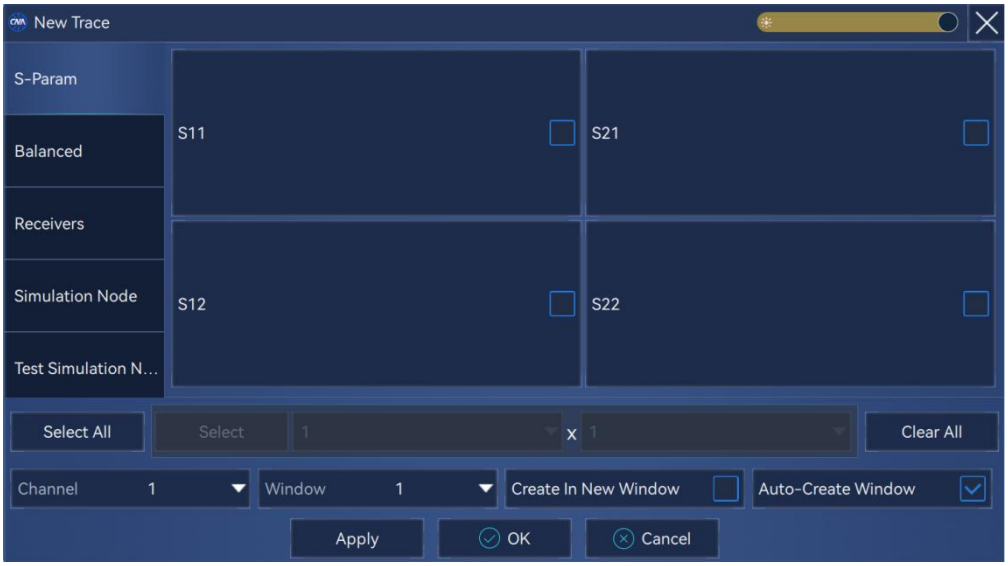


Figure 4.4 S-parameter definition

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) [Bal PARAM] 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] checkbox

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.

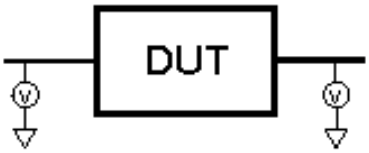


Figure 4.5Single-port device

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 4.6Balanced 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 stimuluss. Taking the 4-port balance parameters as an example, the mixed mode S-parameter matrix can be used to completely characterize its characteristic index.

$$S_{mm} = \left[\begin{array}{cc|cc} S_{dd11} & S_{dd12} & S_{dc11} & S_{dc12} \\ S_{dd21} & S_{dd22} & S_{dc21} & S_{dc22} \\ \hline S_{cd11} & S_{cd12} & S_{cc11} & S_{cc12} \\ S_{cd21} & S_{cd22} & S_{cc21} & S_{cc22} \end{array}\right]$$

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.

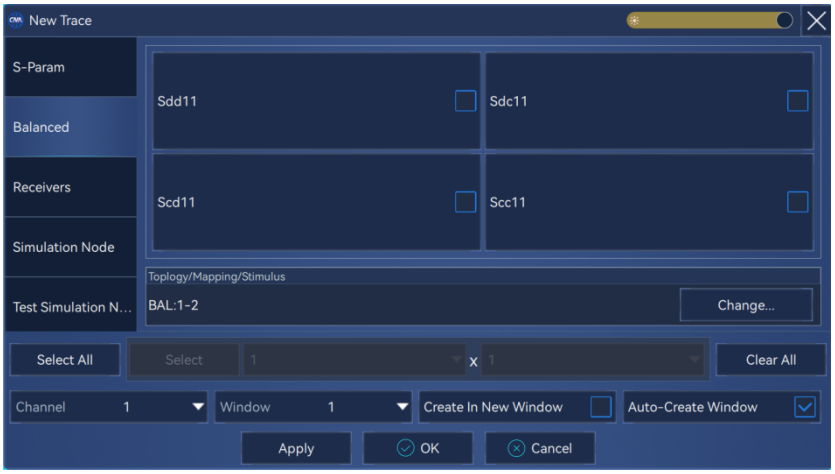


Figure 4.7 Trace creation for differential balance S-parameter

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.

a) Any two physical test ports can be mapped to a balanced logical port;

b) Any physical test port can be mapped to a single-ended logical port

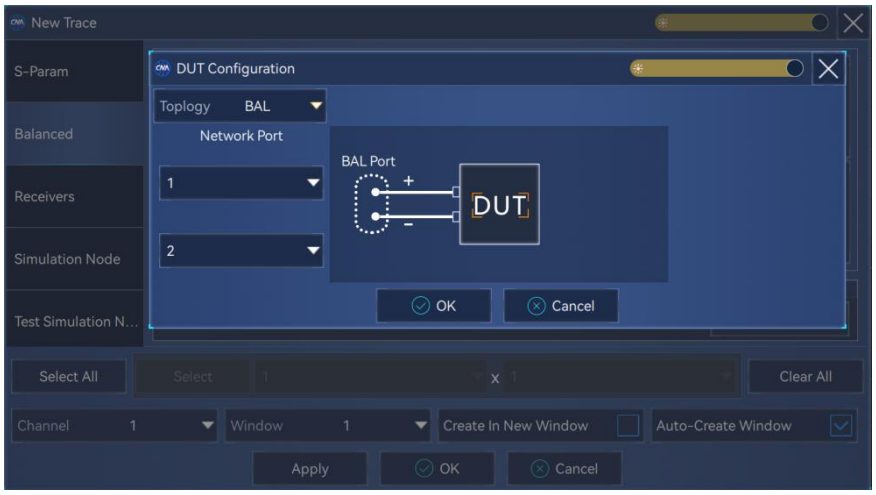
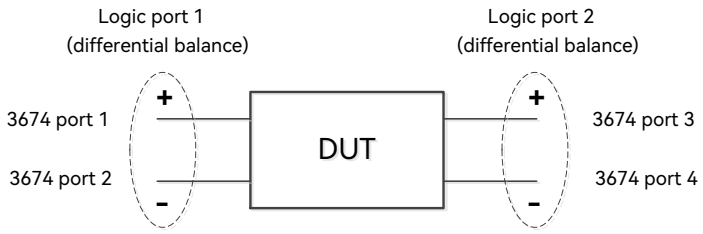


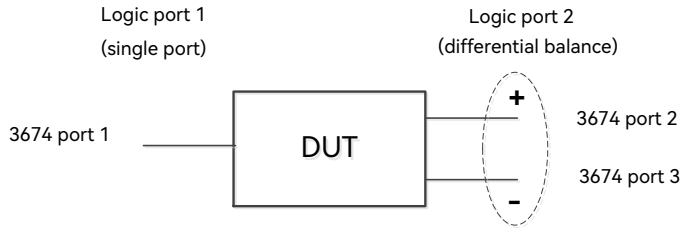
Figure 4.8 Differential Balanced S-parameter Topology Settings dialog box

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:

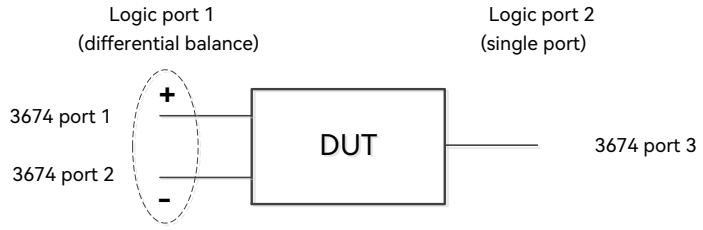
a) Balanced/balanced (two differential balanced logical ports and four 3674 series physical ports)



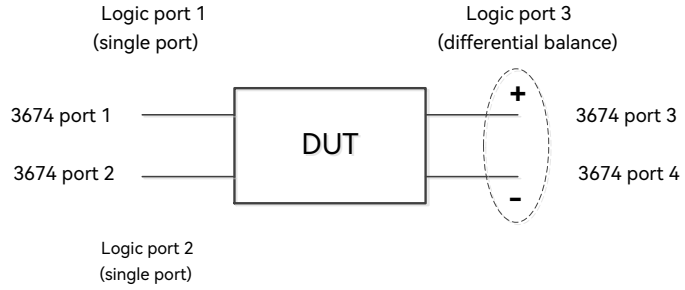
b) Single-ended/balanced (two logical ports and three 3674 series physical ports)



c) Balanced/single-ended (two logical ports and three 3674 series physical ports)



d) Single-ended - single-ended/balanced (three logical ports and four 3674 series physical ports)



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

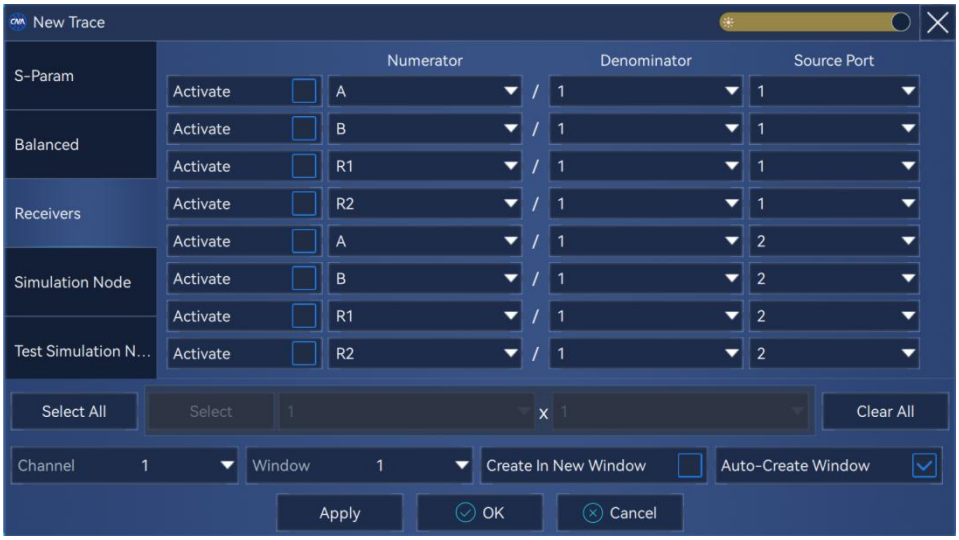


Figure 4.9 S-parameter definition

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] checkbox

Create a new trace when checked.

b) [Numerator] zone

Select the numerator for arbitrary ratio measurement or the measurement 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] checkbox

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] checkbox

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 4-11.

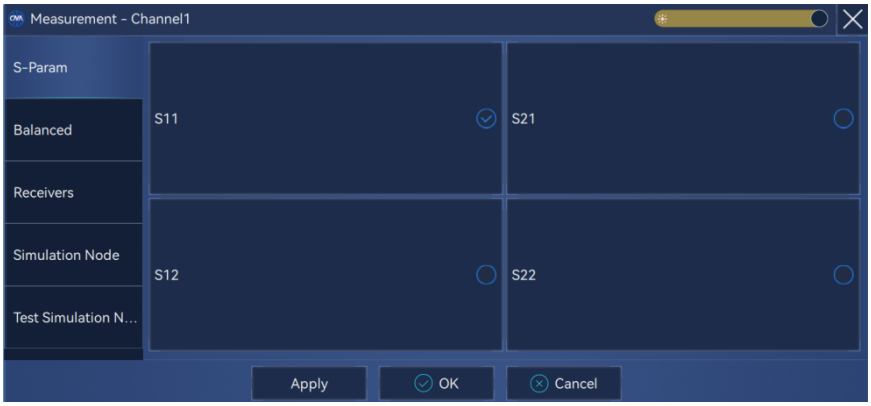


Figure 4. 11 Set the measurement type of S parameter of current active trace

4.3. Setting 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:

a) Specify start/stop frequency
b) Specify center frequency and frequency span

2) Set the start and stop frequencies

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

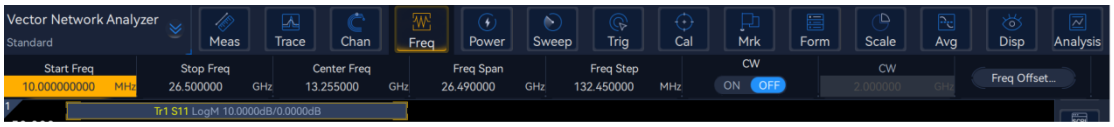


Figure 4.12 Set the start and stop frequencies

3) Set center frequency and frequency span

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

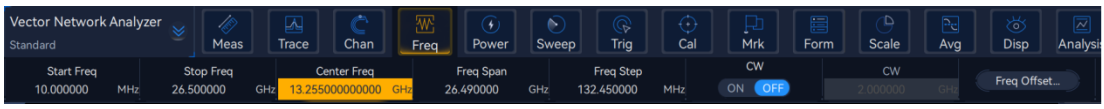


Figure 4.13 Set center frequency and frequency span

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 3674 series vector network analyzer port are as follows:

Table 4.2 Source output signal power level index

Frequency range:	Source power range (dBm)	Configuration
10MHz~9GHz/14GHz/20GHz/26.5GHz /32GHz/44GHz/50GHz/53GHz/67GHz	-25 ~ +20	Standard configuration
10MHz ~ 9GHz/14GHz/20GHz/26.5GHz	-95 ~ +20	Equipped with programmable step attenuator option
10MHz ~ 32GHz/44GHz/50GHz	-85 ~ +20	Equipped with programmable step attenuator option
10MHz ~ 53GHz/67GHz	-75 ~ +20	Equipped with programmable step attenuator option

1) Set the power level

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





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]** checkbox in the Power Setting dialog box.

3) Set attenuator 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

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] checkbox

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

n) [Port Powers Coupled] check box

Check the **[Port Powers Coupled]** checkbox 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] checkbox is unchecked, the analyzer allows the power level of each port to be set separately.

c) [State] column

The state is set to **[Auto]** by default, and in this case, the port power On/Off state 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 Attenuator] 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) [Attenuator] column

It is used to set the attenuator value.

g) [Leveling 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

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.....117
- Set Sweep Type.....118
- Sweep Time.....121
- Sweep Setting.....122

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.

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) Point frequency

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 Setting

1) Set sweep type

Menu path: [Swp]→[Swp TYP].

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

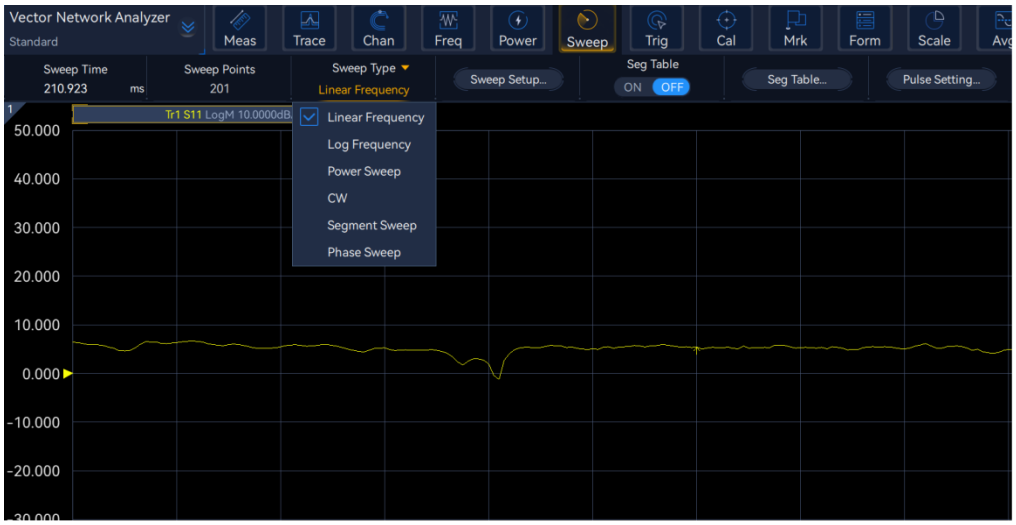




Figure 4. 15 Set the linear frequency sweep type

2) Set segment sweep type

Menu path: [Swp]→[Swp TYP].

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



Figure 4.16 Set segment sweep type

Description of dialog box settings:

a) **[Indep PwrLvl]** checkbox: When checked, each segment can set a separate power level, but it must be the same attenuator setting.

b) **[Indep IFBW]** checkbox: When checked, a separate IF bandwidth can be set in each segment.

c) [Indep Swp TM] checkbox: 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.



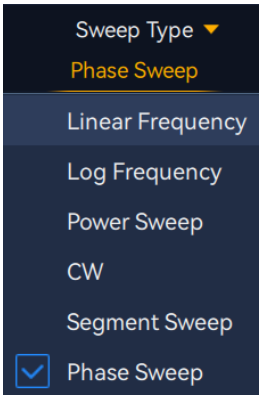
Figure 4.17 Insert and delete segments

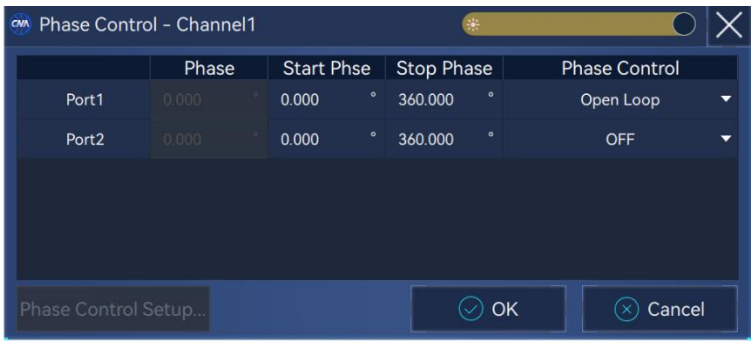
4) Edit segment table

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





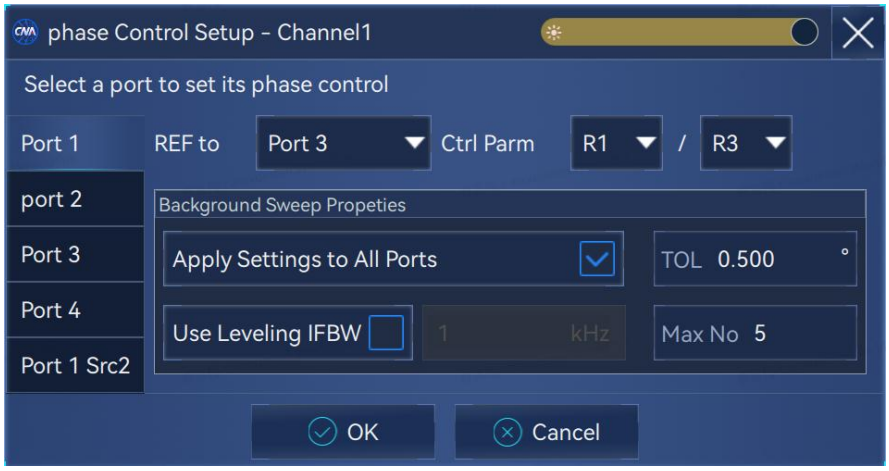


Figure 4. 18 Setting to phase sweep

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 sweep time

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.

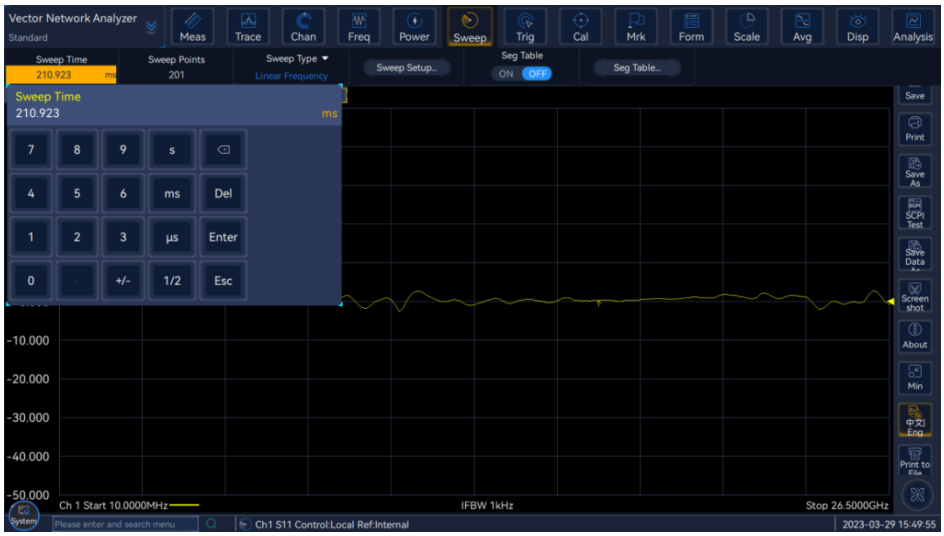


Figure 4.19 Set sweep time

4.5.4 Sweep Setting

1) Sweep setting

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

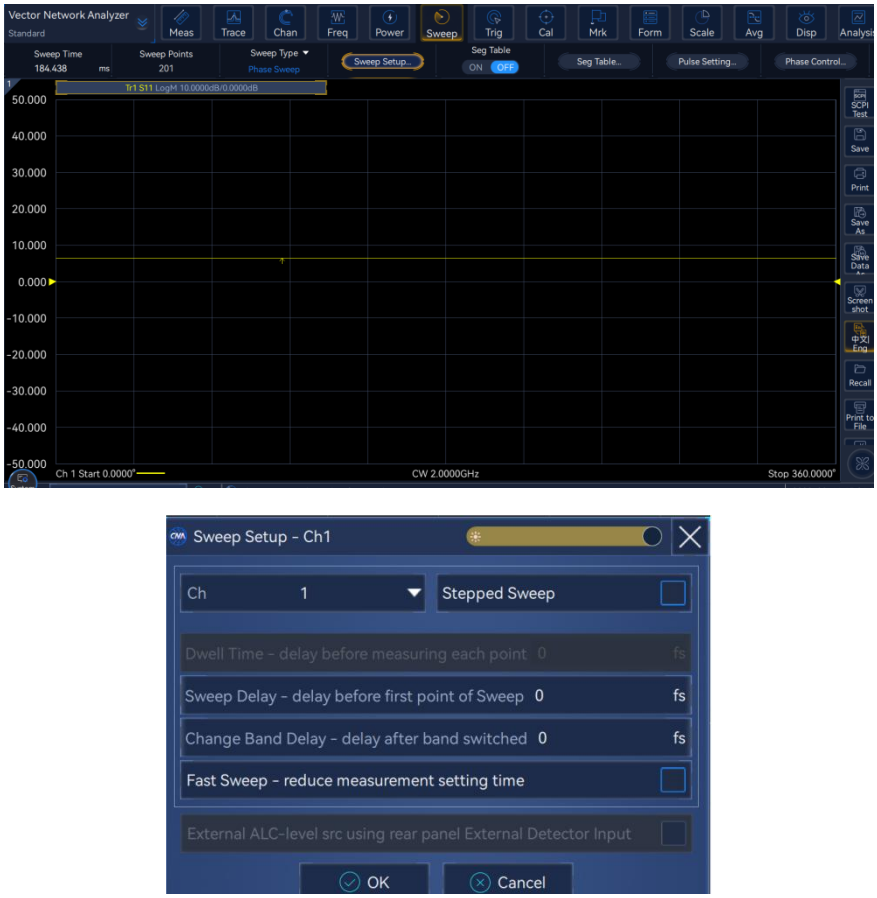


Figure 4.20 Sweep setting

2) Sweep setting dialog box

a) [Ch] box

Select the channel to which the sweep setting applies.

[b) [Step Swp] checkbox

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 checkbox 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 TM] 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 setting.....123
- Detailed trigger setting.....123
- Trigger dialog box.....124

4.6.1 Simple Trigger Setting

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 Setting

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



Figure 4.22 Trigger submenu

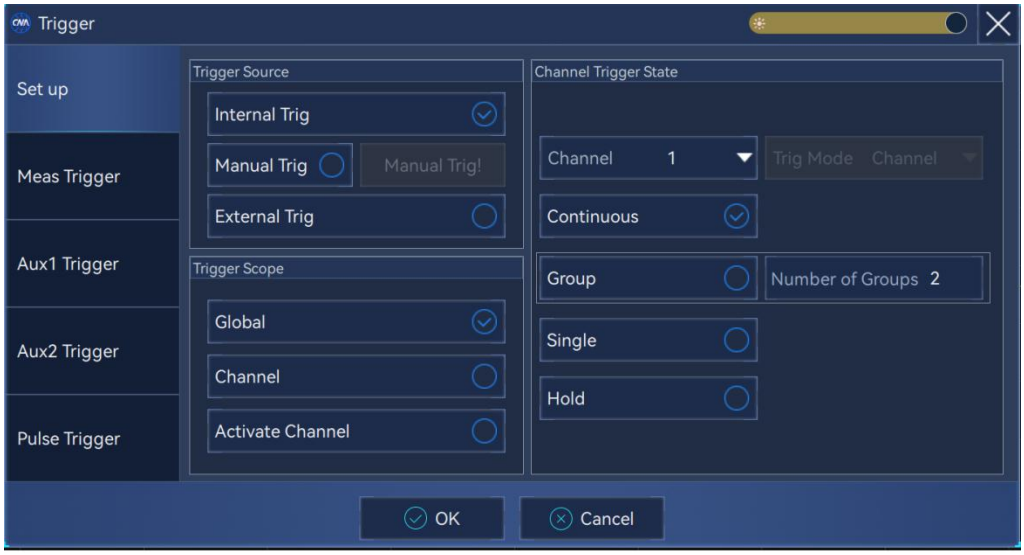


Figure 4.23 Trigger setting dialog box

4.6.3 Trigger Dialog Box

1) Trigger setting item

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 Scope zone

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 setting 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.

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

2) Meas 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.

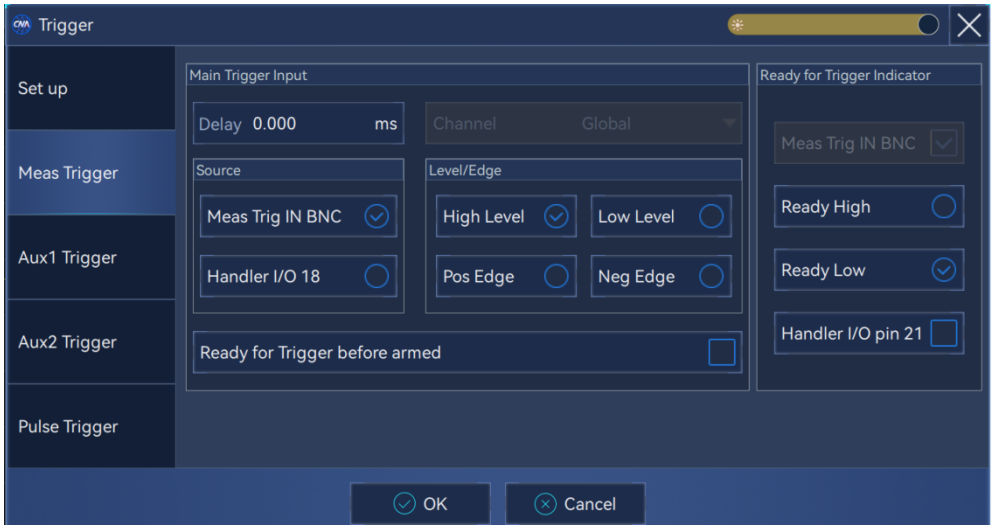


Figure 4. 24 Meas Trigger dialog box

3) Aux1/2 Trigger options

a) AUX TRIG OUT 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]**

  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.

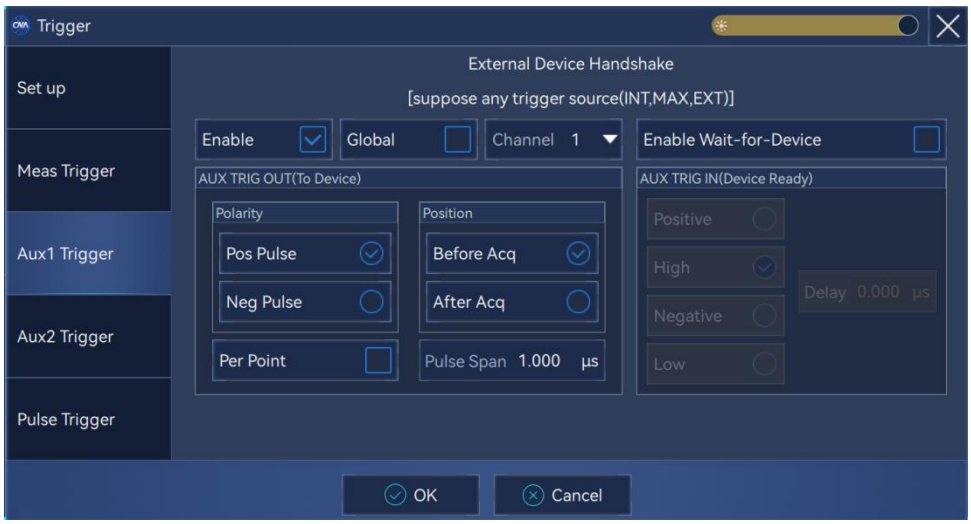


Figure 4. 25 AUX1/2 Trigger dialog box

4) Pulse Trigger options

a) Pulse Sync input area

It is for select to use an internal source or an external source to synchronize

the analyzer internal pulse generator.

- **[Source]**

  1) **Internal**: The pulse generator is synchronized by the periodic signal generated inside the analyzer, and the output pulse train period is specified by the pulse repetition period in the pulse measurement interface.

  2) **External**: When External is selected, the pulse repetition period specified by the user is ignored, and the analyzer outputs a series of pulse trains every time it detects a valid external pulse sync signal. The delay and span of the pulse generator can be set separately through the pulse measurement interface. External pulse sync input signal is input from the **pulse input/output interface** at the rear panel of the analyzer. The pulse sync input signal can input a configurable sync trigger signal to the analyzer. In case of level triggering, if the level remains valid after the analyzer outputs a series of pulse trains, the analyzer continues to output another series of pulse trains; in case of edge triggering, a valid edge signal triggers the analyzer to output a series of pulse trains. The burst duration is equal to the maximum (delay + span) of all pulse generators enabled.

- **[Level/Edge]**

  It is for setting the effective signal characteristics among High Level, Low Level, Pos Edge and Neg Edge when the analyzer receives the external pulse sync input signal.

b) Receiver output area

- **[Synchronize receiver to pulse]**

  It is for broadband sync pulse measurement, and when it is enabled, the pulse generator output is allowed to be synchronized with data acquisition.

- **[Sync Delay]**

  It is for setting the delay time before the analyzer acquires AD data**.**

- **[Pulse PRI]**

  It is for setting the pulse period for AD data acquisition of analyzer.

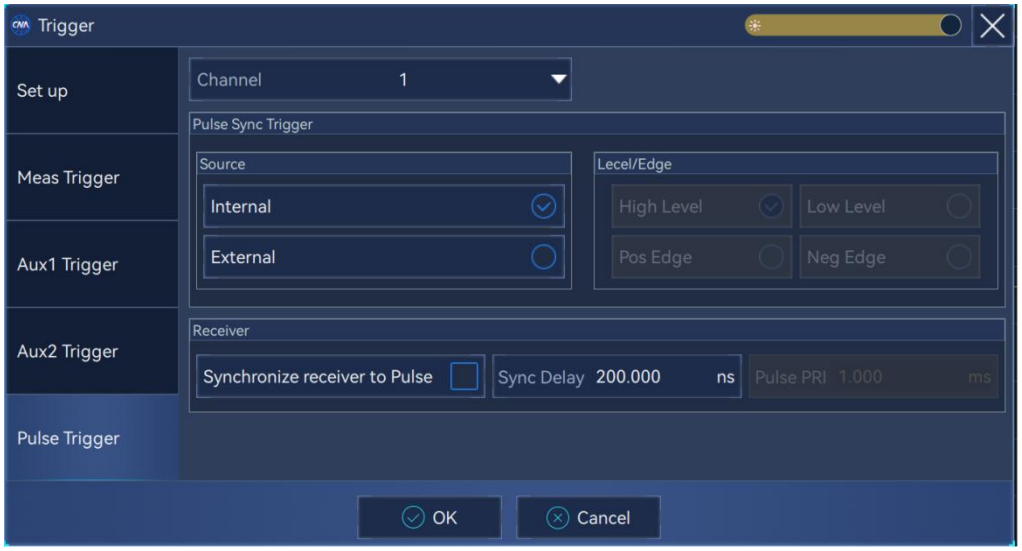


Figure 4. 26 Pulse Trigger dialog box

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.....129
- Set data format.....132
- Scale.....133

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 4.27 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.

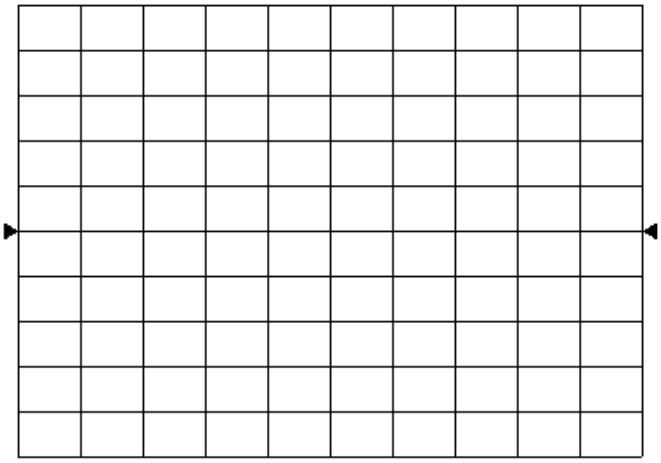


Figure 4.27 Cartesian format

a) Logarithm magnitude format
- Display amplitude information (no phase information)
- Y axle 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 magnitude 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
- Display the reflection measurement data calculated by equation (1+ρ)/(1-ρ), 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 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. 28, 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 marker. The default marker format is real and imaginary, or you can open the Marker dialog box through the Marker/Analysis menu and select another format in the Advanced Marker.

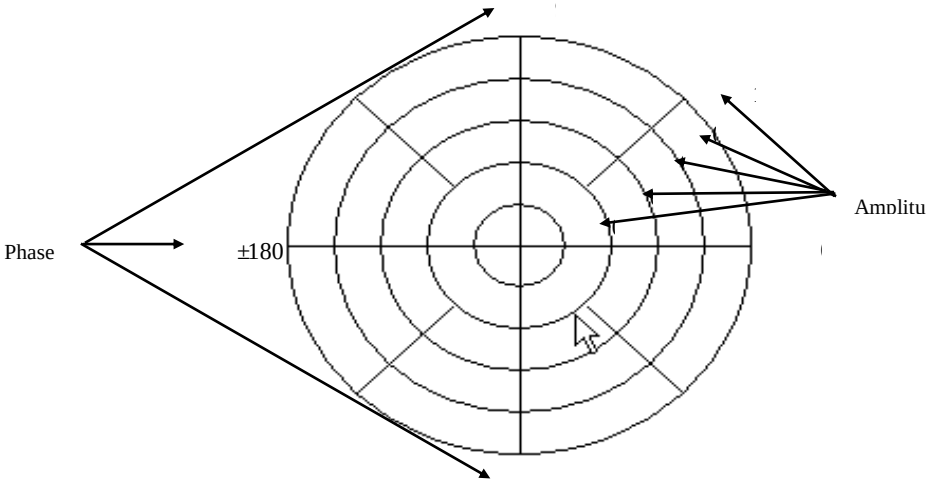


Figure 4. 28 Polar coordinate format

3) Smith chart format

The Smith chart shown in Figure 4.29 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 (±jX). The resistance value, reactance value, equivalent capacitance value and inductance value of the DUT can be read through the marker.

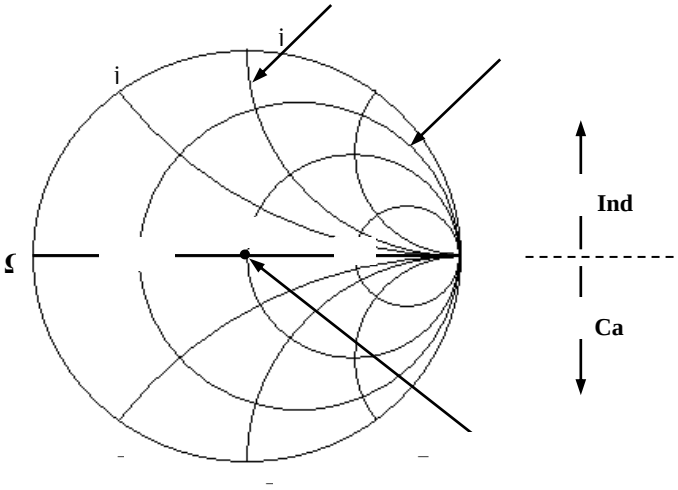


Figure 4.29 Smith chart format

a) 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.
b) The points on the circle intersecting the horizontal axis on the Smith chart have equal resistance values.
c) The points on the arc tangent to the horizontal axis on the Smith chart have the same reactance value.
d) The reactance of the upper half of Smith chart is positive, so it is an inductive zone.
e) 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.

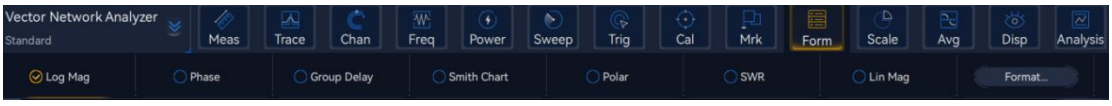




Figure 4.30 Set data format

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 4.31 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

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.

- Automatically Set to Multi-window Display.....136
- Manually Add Measurement Traces and Windows.....136

4.8.1 Automatically Set to Multi-window Display

Menu path: **Click [Display] → [Window Layout], and select window layout among Tiled/Stack-up/Overlay 1x/Stack 2x/Split 3x/Quad 4x/One Window Per Measurement/One Window Per Channel/Arrange In Rows.**

a). All the created windows are displayed in the manner of tiled/stack-up on the screen.

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

c). All created traces are displayed in the manner of One Window Per Measurement/One Window Per Channel/Arrange In Rows manner.

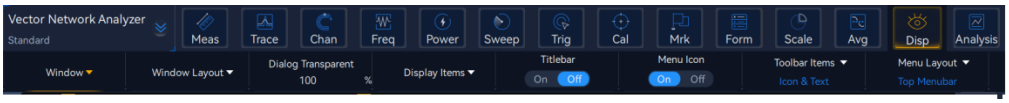


Figure 4.32 Setting pre-configured measurement

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

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

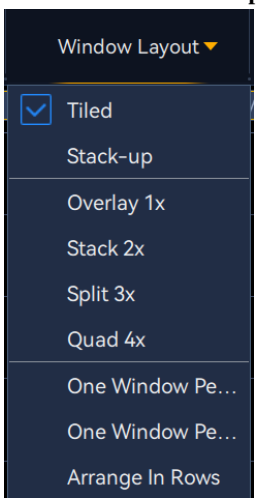


Figure 4.33 Set window arrangement

- Tiled

When this window layout is selected, all windows will be tiled on the screen.

- Stack-up

When this window layout is selected, all windows will be stacked up on the screen.

- Overlay 1x

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.

- One Window Per Measurement

When this window layout is selected, the traces of each window will be displayed in one window. The number of windows is equal to the number of traces, and the windows are arranged in the tiled manner by default.

- One Window Per Channel

When this window layout is selected, the traces of each channel will be displayed in

one window. The number of windows is equal to the number of channels, and the windows are arranged in the tiled manner by default.

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.....136
- Tool Bar.....136
- List.....137
- Display contents.....137

4.9.1 Status Bar



Figure 4.34 Status bar

1) When the Status bar is open, 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
   - 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.

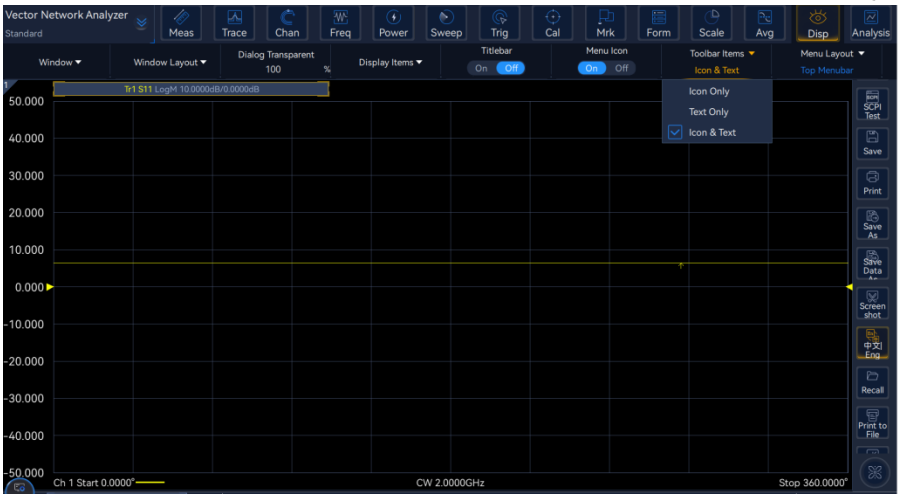


Figure 4. 35 Toolbar

4.9.3 List

a) Segment Table

	State	Start Stimulate	Stop Stimulate	Points	IF Bandwidth	Sweep Time	Power Level
1	Off	500.000 Hz	110.000 GHz	21	1.000 kHz	0 fs	-5.000 dBm
2	Off	110.000 GHz	110.000 GHz	21	1.000 kHz	0 fs	-5.000 dBm

Figure 4.36 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

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
- c) Frequency/stimulus state
- marker 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.

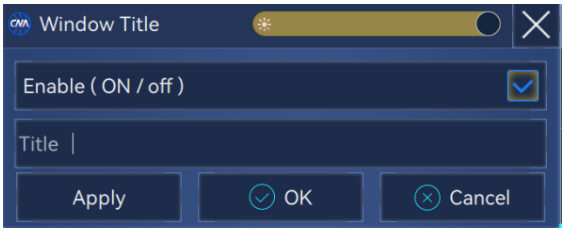


Figure 4.37 Title Input dialog box

b) Trace Status

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 Status

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

d) Marker display

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

- Marker number
- Stimulus value
- Response value

e) Display time state

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

2) Triggering Display Items

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.

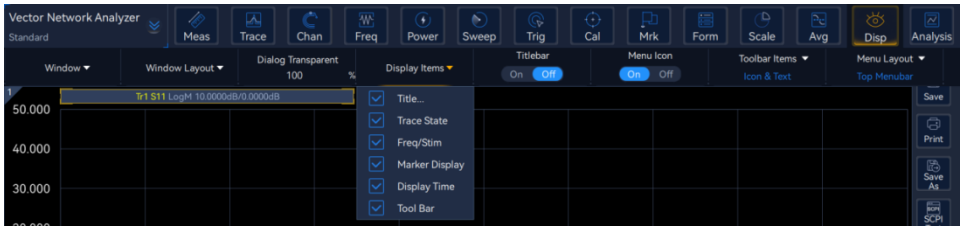


Figure 4.38 Triggering display content

5 Menus

Menu is the path for you to operate and set the vector network analyzer. The 3674 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 3674 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.....140
- Menu Description.....163

5.1 Menu Structure

The menus of different levels are as follows:

- Meas.....141
- Trace.....142
- Chan.....143
- Freq.....144
- Power.....145
- Sweep.....146
- Trigger.....147
- Cal.....148
- Mrk.....149
- Form.....150
- Scale.....151
- Avg.....151
- Disp.....153
- Analysis.....157
- System.....160

5.1.1 Meas



Note: Press the menu with an ellipsis (...) to open the corresponding dialog box, the same below.

Figure 5.1 Meas menu

5.1.2 Trace

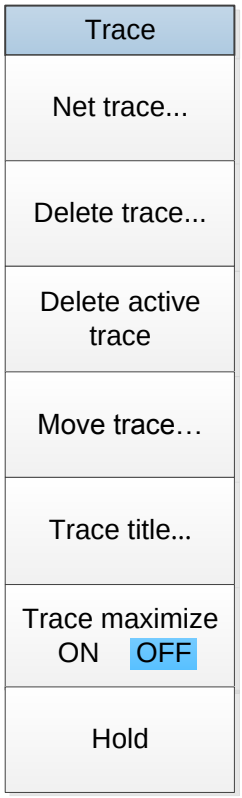


Figure 5.2 Trace menu

5.1.3 Channel

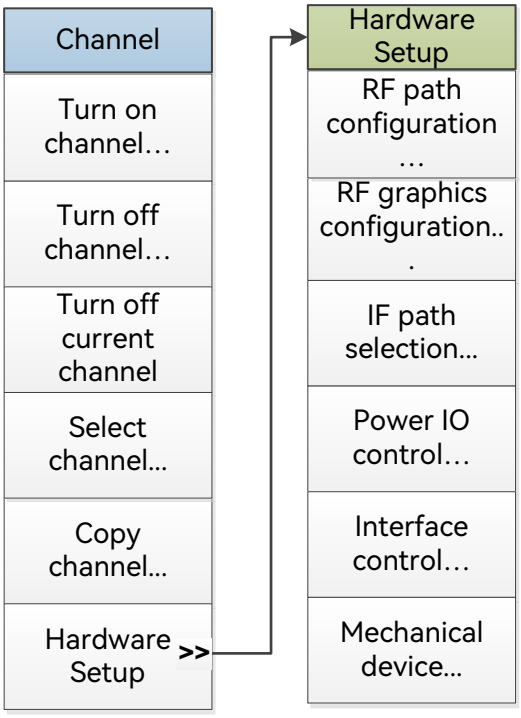


Figure 5.3 Chan menu

5.1.4 Freq

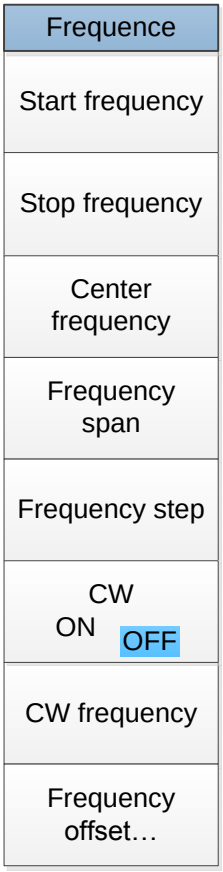


Figure 5.4 Freq menu

5.1.5 Power



Figure 5.5 Power menu

5.1.6 Sweep

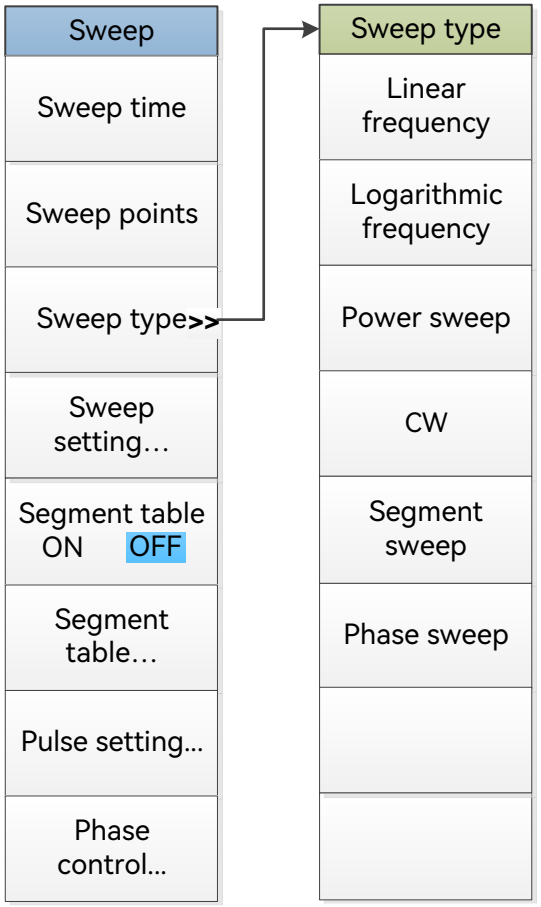


Figure 5.6 Sweep menu

5.1.7 Trigger

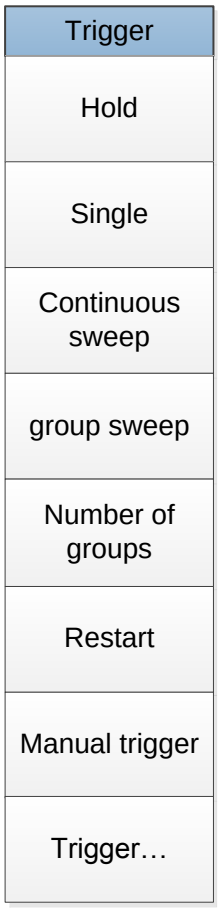


Figure 5.7 Trigger menu

5.1.8 Calibration

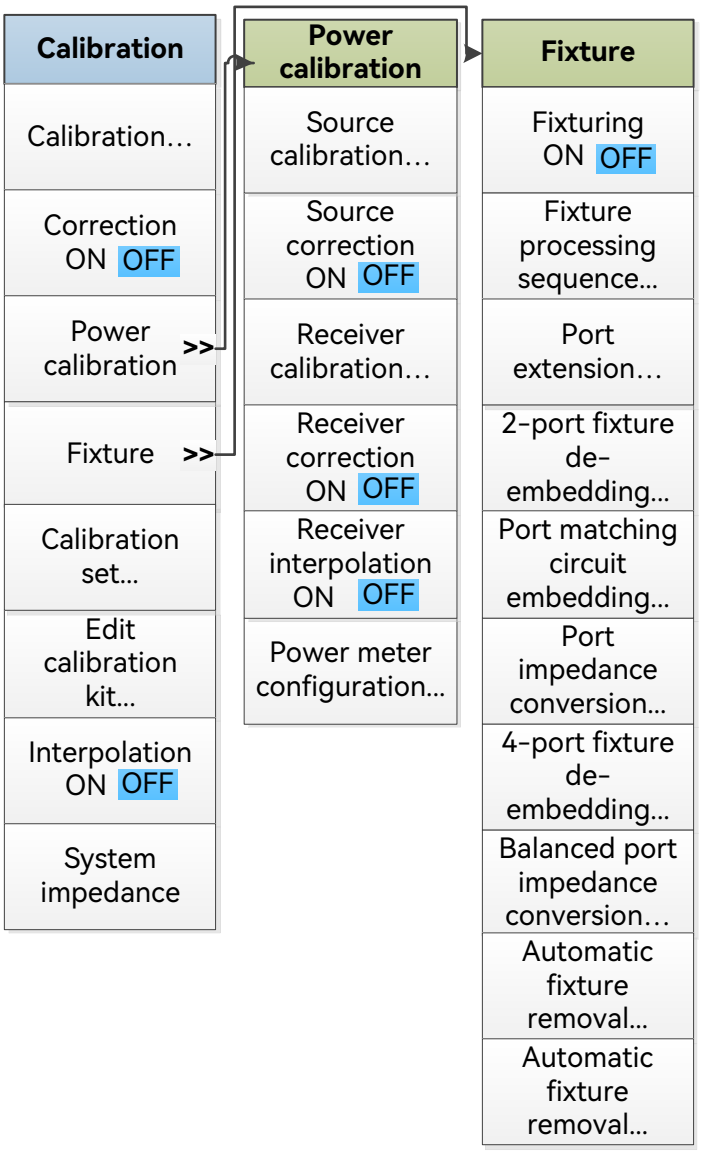


Figure 5.8 Calibration menu

5.1.9 Marker

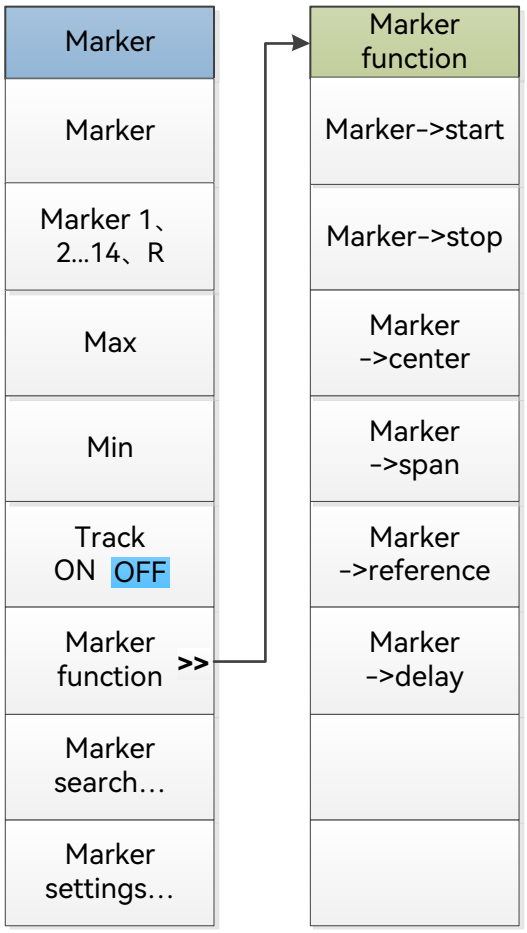


Figure 5.9 Mrk menu

5.1.10 Format



Figure 5.10 Format menu

5.1.11 Scale

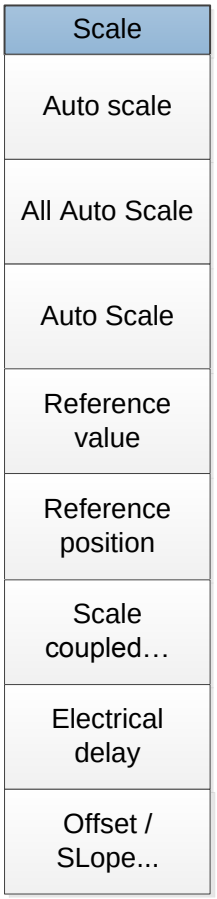


Figure 5.11 Scale menu

5.1.12 Average

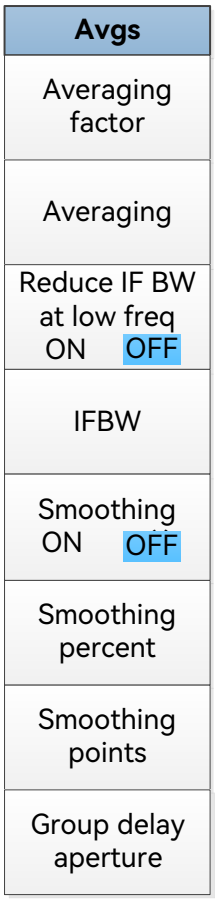


Figure 5.12. Avgs menu

5.1.13 Display

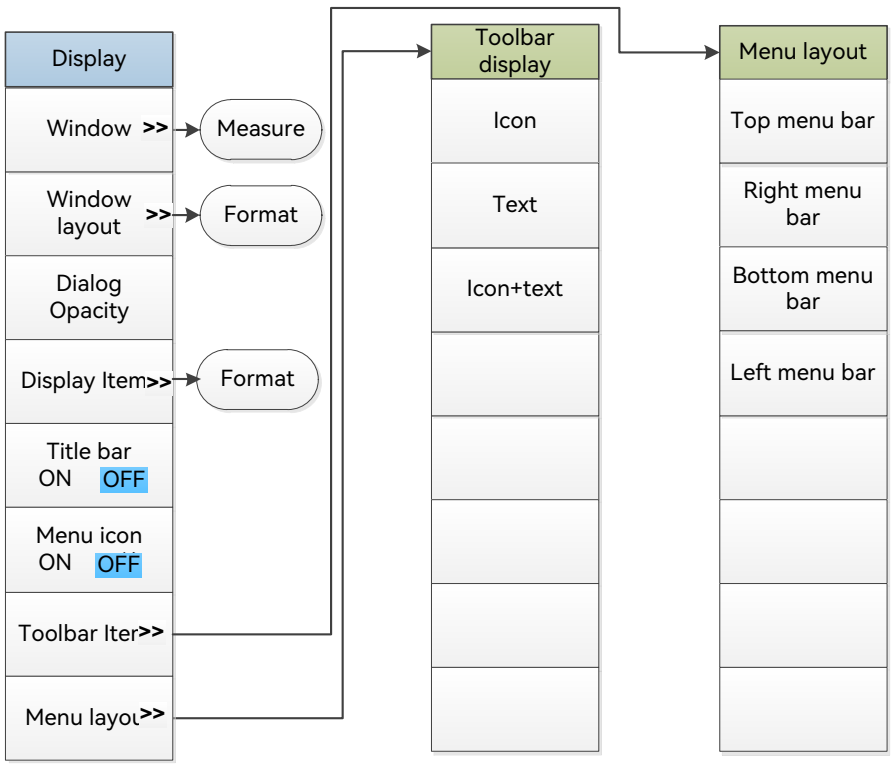


Figure 5.13 Display menu 1



Figure 5.14 Display menu 2

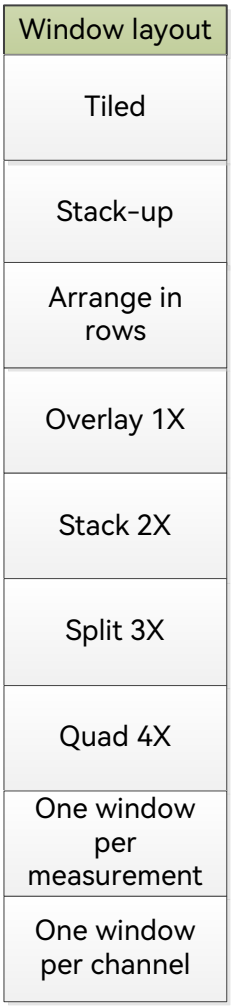


Figure 5.15 Display menu 3

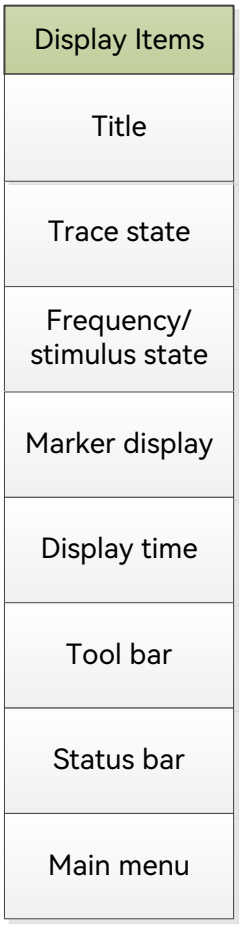


Figure 5.16 Display menu 4

5.1.14 Analysis

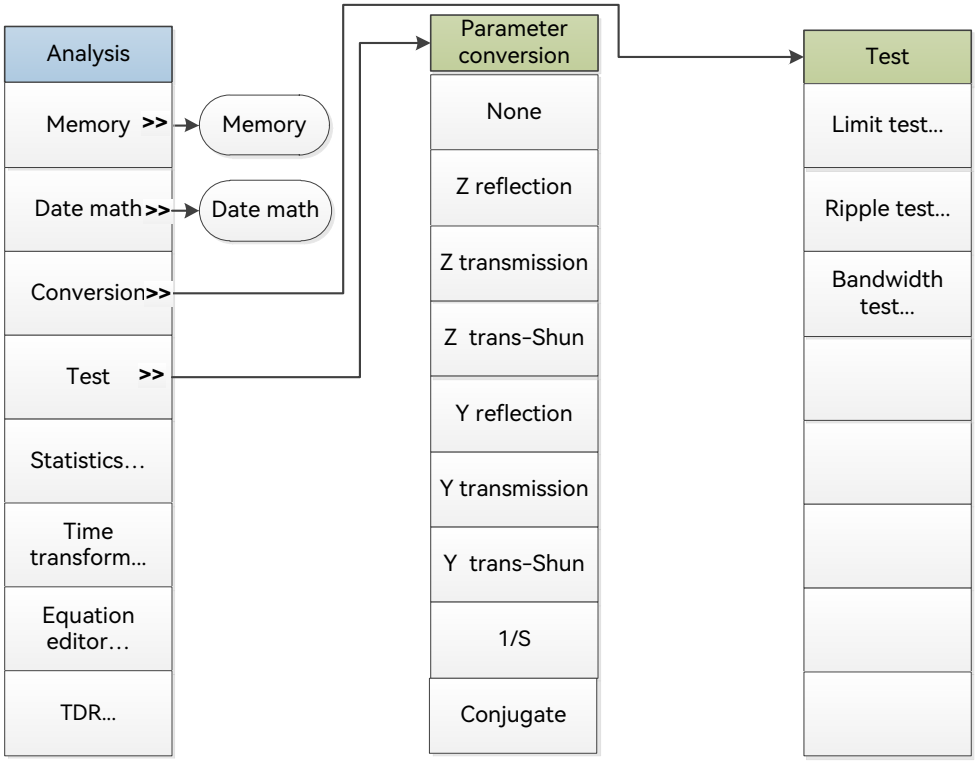


Figure 5.17 Analyzer menu 1

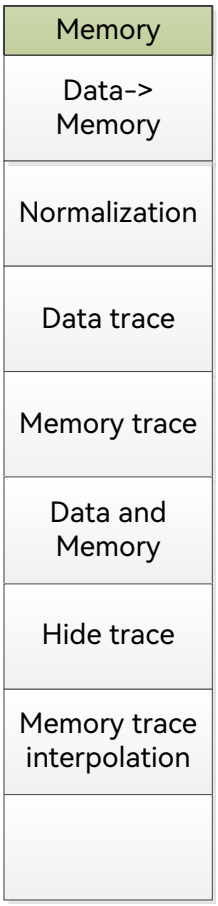


Figure 5.18 Analyzer menu 2

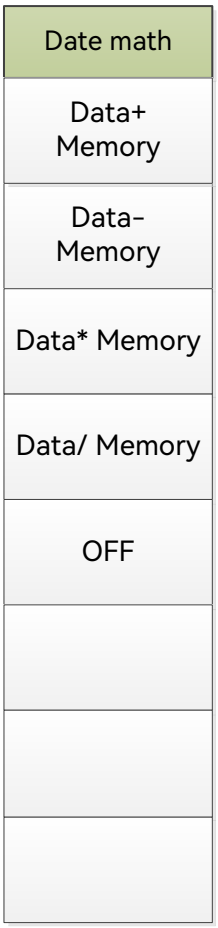


Figure 5.19 Analyzer menu 3

5.1.15 System

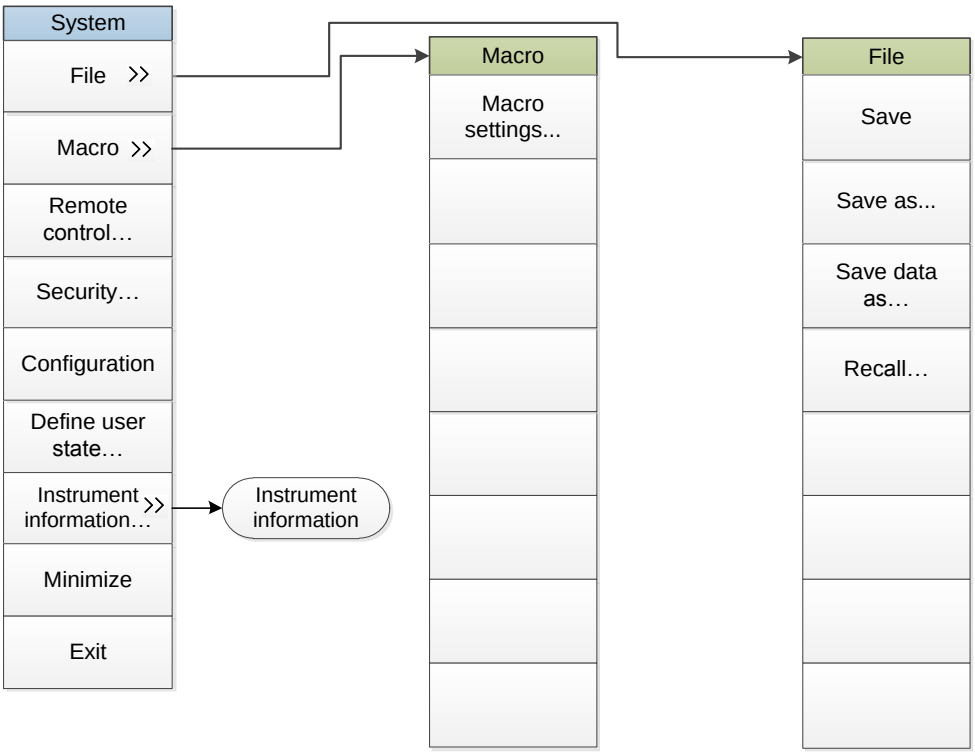


Figure 5. 20 System menu 1

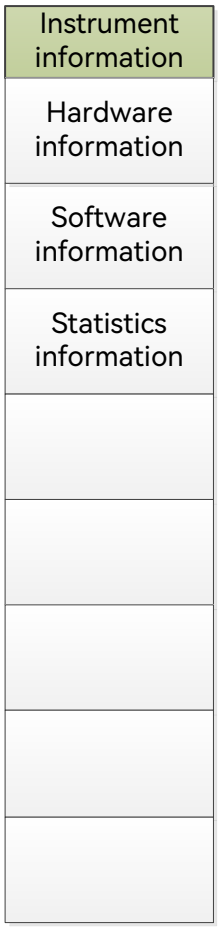


Figure 5.21 System menu 2

1) Main menu

The 3674 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 marker 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) Submenus

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.

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

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

- Meas.....163
- Trace.....165
- Chan.....166
- Freq.....169
- Power.....171
- Sweep.....173
- Trigger.....175
- Cal.....177
- Mrk.....182
- Form.....185
- Scale.....186
- Avg.....151
- Disp.....191
- Analysis.....191
- System.....196

5.2.1 Measurement

Press the [Meas] button on the front panel or click the [Meas] at the top of user interface to pop up a menu 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:

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] checkbox and click [OK] to complete the trace title change.

Parameter description:

None

5.2.2.6 Trace 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].

5.2.2.7 Hold

Function description:

Click this menu to enable the system to hold the display of the currently active trace. Optional values include [OFF **|** Max **|** Min **|** Clear], and the default value is Clear.

Parameter description:

OFF [OFF **|** Max **|** Min **|** Clear].

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:



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 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
- RF graphics configuration
- IF selection
- Extended interface control
- Interface control
- Mechanical device

1) RF path configuration

Function description:

Click this manual to configure the corresponding RF path.

Parameter description:

None

2) RF graphics configuration

Function description:

Click this menu to configure the corresponding RF graphics.

Parameter description:

None

3) IF selection

Function description:

Click this menu to configure the corresponding IF path.

Parameter description:

None

4) 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) Interface control

Function description:

Click this menu to configure the corresponding interface.

Parameter description:

None

6) Mechanical device

Function description:

Click this menu to configure the corresponding mechanical device.

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], [End 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

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.

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 Frequency

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:

500Hz/10MHz [500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

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:

9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz

[500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

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:

4.505GHz/7.005GHz/10.005GHz/13.255GHz/16.505GHz/22.005GHz/25.005GHz/26.505GHz/33.505GHz

4.50GHz/7.00GHz/10.00GHz/13.25GHz/16.50GHz/22.00GHz/25.00GHz/26.50GHz/33.50GHz (with 500Hz LF option)

[500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

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:

8.99GHz/13.99GHz/19.49GHz/26.49GHz/32.99GHz/43.99GHz/49.99GHz/52.99GHz/66.99GHz

9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz (with 500Hz LF option)

[500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

5.2.4.5 Frequency step

Function description:

Set the frequency span between two successive sweep 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:

44.95MHz/69.95MHz/99.95MHz/132.45MHz/164.95MHz/219.95MHz/249.95MHz/264.95MHz/69.95 MHz/334.95MHz

45MHz/70MHz/100MHz/132.5MHz/165MHz/220MHz/250MHz/265MHz/70MHz/335MHz (with 500Hz LF option)

[500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

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:

2GHz [500Hz/10MHz ~ 9GHz/14GHz/20GHz/26.5GHz/32GHz/44GHz/50GHz/53GHz/67GHz].

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



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

5.2.5.2 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

[-25 ~ 20dBm (without programmable step attenuator option)];

[-95 ~ 20dBm (programmable step attenuator option configured for 3674B/C/D/E)];

[-85 ~ 20dBm (programmable step attenuator option configured for 3674F/G/H)];

[-75 ~ 20dBm (programmable step attenuator option configured for 3674K/L)].

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:

-20dBm

[-25 ~ +20dBm (without programmable step attenuator option)];

[-95 ~ + 20dBm (programmable step attenuator option configured for 3674B/C/D/E)];

[-85 ~ +20dBm (programmable step attenuator option configured for 3674F/G/H)];

[-75 ~ + 20dBm (programmable step attenuator option configured for 3674K/L)].

5.2.5.5 Stop Power

Function description:

Set the stop power during power sweep.

Parameter description:

0dBm

[-25 ~ +20dBm (without programmable step attenuator option)];

[-95 ~ + 20dBm (programmable step attenuator option configured for 3674B/C/D/E)];

[-85 ~ +20dBm (programmable step attenuator option configured for 3674F/G/H)];

[-75 ~ + 20dBm (programmable step attenuator option configured for 3674K/L)].

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

[0 ~ 70dB (programmable step attenuator option configured for 3674B/C/D/E)];

[0 ~ 60dB (programmable step attenuator option configured for 3674F/G/H)];

[0 ~ 50dB (programmable step attenuator option configured for 3674K/L)].

5.2.5.7 Attenuator Mode Manual Auto

Function description:

This function is available only after the programmable step attenuator option is configured, and is used for switching the attenuator mode between Manual/Auto.

Parameter description:

Auto: The attenuation is automatically set during power level setting, and the control logic is as follows:

In case of -15dBm ≤P≤ 20dBm, attenuation value: 0dB

In case of -25dBm ≤P<-15dBm, attenuation value: 10dB

Set the power level -35dBm≤P＜-25dBm; attenuation value: 20dB

Set the power level -45dBm≤P＜-35dBm; attenuation value: 30dB

Set the power level -55dBm≤P＜-45dBm; attenuation value: 40dB

Set the power level -65dBm≤P＜-55dBm; attenuation value: 50dB

In case of -75dBm ≤ P<-65dBm, attenuation value: 50dB (K/L type); 60dB (B/C/D/E/F/G/H type)

In case of -85dBm ≤ P<-75dBm, attenuation value: 60dB (for 3674F/G/H); 70dB (for 3674B/C/D/E)

In case of -95dBm ≤ P<-85dBm, attenuation value: 70dB (for 3674B/C/D/E).

Manual: In manual attenuator mode, this function allows you to manually control the attenuation of the attenuator.

[0~70dB (3674B/C/D/E)];

[0~60dB (3674F/G/H)];

[0 ~ 50dB (3674K/L)].

5.2.5.8 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
✧ Sweep time ✧ Sweep points ✧ Sweep type ✧ Sweep settings ✧ **Segment Table ON OFF** ✧ Segment list ✧ Pulse setting ✧ **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.

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 Setting

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 ✧ Empty 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 Setting

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

5.2.7.4 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], [Power Calibration], [System Impedance]. 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
✧ Fixturing
✧ Processing Sequence
✧ Port Extensions
✧ 2-Port De-embed
✧ Port Matching Circuit Embed
✧ Port Z Conversion
✧ Four-port fixture de-embedding
✧ Balanced Z Conversion
✧ AFR
✧ Cal Plane Manager

1) Fixturing

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) 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) Dual-port fixture de-embedding

Function description:

Click this menu to enable the 2-port fixture de-embed function. For setting method of 2-port fixture de-embed function, see "7.9.4 2-port Fixture De-embed" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

5) Matching circuit embedding

Function description:

Click this menu to enable the matching circuit embed function. For setting method of matching circuit embed function, see "7.9.3 Matching Circuit Embed" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

6) Port Z 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 Z Conversion" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

7) 4-port De-embed

Function description:

Click this menu to enable the 4-port fixture embed/de-embed function.

Parameter description:

None

8) 4-port De-embed

Function description:

Click this menu to enable the 2-port fixture embed/de-embed function. For setting method of 4-port fixture embed/de-embed function, see "7.9.6 4-port Fixture De-embed" 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) Cal Plane Manager

Function description:

Click this menu to call the Cal Plane Manager Setting 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 include:

Table 5.8 Fixture

Menu
✧ Fixturing ON OFF
✧ Port Matching Circuit Embed
✧ 2-Port De-embed
✧ Port Z Conversion
✧ 4-port De-embed/Embed

1) Fixturing 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 embed function. For setting method of matching circuit embed function, see "7.9.3 Matching Circuit Embed" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

3) Dual-port fixture de-embedding

Function description:

Click this menu to enable the 2-port fixture de-embed function. For setting method of 2-port fixture de-embed function, see "7.9.4 2-port Fixture De-embed" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

4) Port Z 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 Z Conversion" in "7.9 Fixture Compensation Calibration".

Parameter description:

None

5) 4-port De-embed/Embed

Function description:

Click this menu to enable the 2-port fixture embed/de-embed function. For setting method of 4-port fixture embed/de-embed 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

5.2.8.9 System Impedance

Function description:

Click this menu to edit the currently calibrated system impedance, which defaults to 50 ohms.

Parameter description:

None

5.2.9 Mrk

Press the [Mrk] 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 marker, including [Mrk], [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 Marker setting. Click the menu to enter the submenu. The detailed menus include:

Table 5.9 Marker

Menu
✧ Marker 1/2/3/4/5/6/7/8/9/10/11/12/13/14 ✧ Reference marker 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 1/2/3/4/5/6/7/8/9/10/11/12/13/14function. Read measurement data, search for specific types of values, or change stimulus settings.

Parameter description:

None

2) Reference marker

Function description:

Select this menu to enable the reference marker 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:

The marker frequency point is defaulted to the center frequency in the frequency sweep range, and can be set to a value in the range between the start frequency and stop frequency.

5.2.9.3 Max value

Function description:

By selecting this menu, the marker 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 Tracking ON OFF

Function description:

Click this menu to enable the marker 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 Level
✧ 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) Marker → Ref Level

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 Setting

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
✧ Imag
✧ Polar
✧ Pos Phase
✧ Imped
✧ 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:

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 Coupled
✧ 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.

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. until 0 is returned.

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.

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 ~ $N_{Sweep\ Points}$* 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 ~ $N_{Sweep\ Points}$]

Percent: 5% [1% ~ 100%].

Frequency range: $F_{Frequency\ Span}$ [$F_{Two\text{-}point\ Frequency\ Span}$ ~ $F_{Frequency\ Span}$]

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:

Tiled: all windows are tiled on the screen.

Stack-up: all windows are stacked up on the screen.

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.

➢ One Window Per Measurement: the traces of each window are displayed in one window. The number of windows is equal to the number of traces, and the windows are arranged in the tiled manner by default.

One Window Per Channel: The traces of each channel are displayed in one window. The number of windows is equal to the number of channels, and the windows are arranged in the tiled manner by default.

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, Toolbar, State Bar and level 1 menu. For the meaning of each item, refer to software interface introduction in 3.3.

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 ✧ Memory Trace Interpolation

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.

Parameter description:

None

5) Hide trace

Function description:

Select this menu to hide the data trace and memory trace.

Parameter description:

None

6) Memory Trace Interpolation

Function description:

Click this menu to enable memory trace interpolation.

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 limit 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 ripple 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 BW 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 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, GPIB address, network socket port number, and command monitoring enable state.

Parameter description:

GPIB address: default 16, value range: 0 - 30.

Network port number: default 5025, value range: 1024~65535.

Note: If the analyzer is reset after the GPIB address and port number are successfully changed, the changes are still valid and will not be reset to the original state.

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 MMW Spread Spectrum Configuration

Function description:

Click this menu to call the MMW Spread Spectrum Configuration dialog box.

Parameter description:

None.

5.2.15.5 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.

5.2.15.7 Minimize

Function 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.

5.2.15.8 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:

- Reducing the impact of accessories.....201
- Improving the reflection accuracy of low-loss two-port devices.....202
- Add dynamic range.....204
- Improve Measurement Results of Electrical Length Devices.....207
- Improve the phase measurement accuracy.....209
- Reduce Trace Noise.....215
- Increase Data Points.....218
- Improve Measurement Stability.....219
- Improve Sweeping Speed.....221
- Improving multi-state measurement efficiency 225
- Accelerating data transmission.....227
- Use Macro.....228

6.1 Reducing 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) Cable loss compensation by power slope

If there are long cables or other accessories in a measurement settings, power loss will occur across the entire frequency band, and the power slope feature can be applied, which can increase the source power of the analyzer with the defined rate (dB/GHz). The operation procedure of cable loss compensation is as follows:

Menu path: **[Power]**→**[Power and Attenuators**...**]**.

If the power slope feature is not active, check the **[Power Slope]** check box. Enter the desired increase rate of source power upon frequency sweep in **dB/GHz** box.

2) 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]** 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.

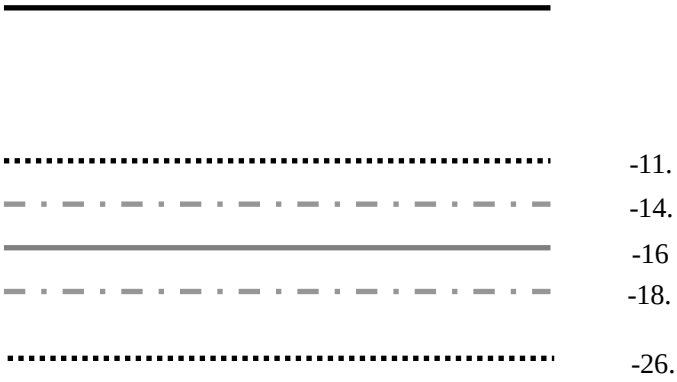


Figure 6.1 Comparison of measurement instability

Representative filter reflection
Measurement uncertainty without a 10dB precision attenuator.
Measurement uncertainty with a 10dB precision attenuator

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.

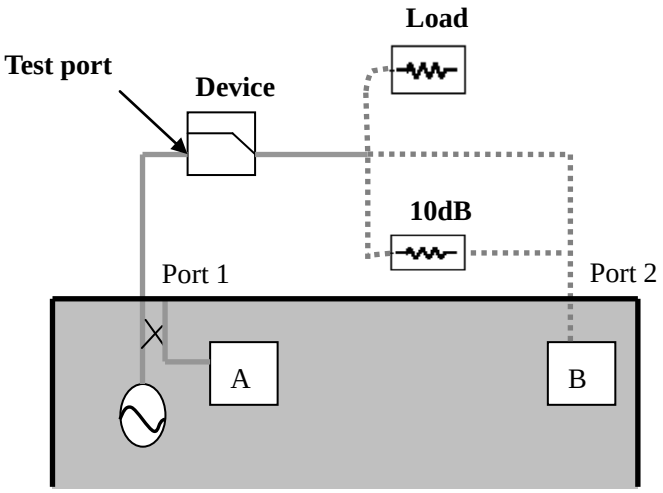


Figure 6.2 Precise reflection measurement of dual-port device

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**

ρ_{NA} $= (F_{IL}) * (NA_{LM}) * (F_{IL})$ $= (F_{IL}) * (A_{IL}) * (NA_{LM}) * (A_{IL}) * (F_{IL})$

$= (.891) * (.126) * (.891)$ $= (.891) * (.316) * (.126) * (.316) * (.891)$

$=.100$ $=.010$

$\rho_{Attenuator}$ NA $= (F_{IL}) * (A_{SWR}) * (F_{IL})$

$= (.891) * (.126) * (.891)$

$=.019$

Error in worst case (E_{WC}) ρ_{NA} $= \rho_{NA} + \rho Attn.$

= . 1 =. 01.019 =. 029

Uncertainty=-20log(F_{RL}) (E_{WC}) (NA_D) =-20log(F_{RL}) (E_{WC}) (NA_D)

upper limit=-20log(.158) (.100) (.010) =-20log(.158) (.029) (.010)

= 11.4dB = 14.1dB

Uncertainty=-20log(F_{RL})-(E_{WC})-(NA_D) =-20log(F_{RL})-(E_{WC})-(NA_D)

lower limit=-20log(.158)-(.100)-(.010) =-20log(.158)-(.029)-(.010) = **26.4dB** = **18.5dB**

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.

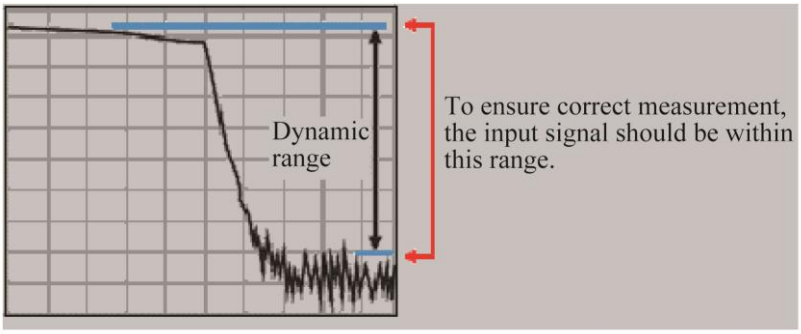


Figure 6.3 Dynamic range

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) Apply 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) Reduce IF bandwidth

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.

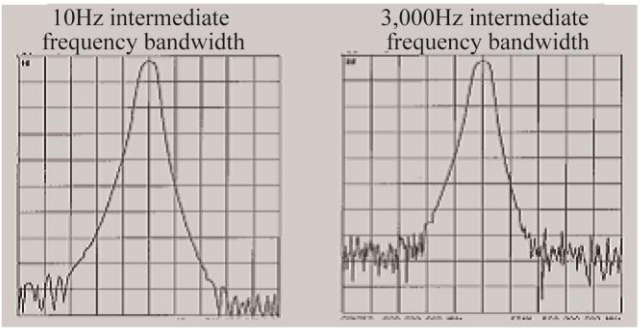


Figure 6.4 Effect of IF bandwidth on measurement results

a) Set IF bandwidth

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.

Figure 6.5 shows the effect of sweep average on measurement results.

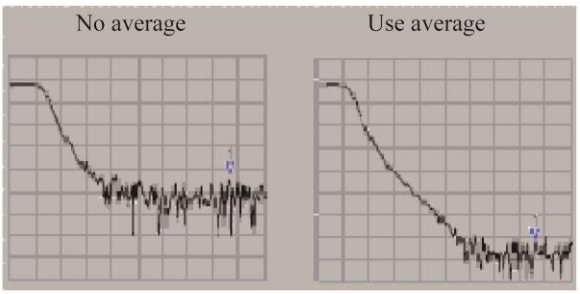


Figure 6.5 Effect of sweep average on measurement results

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 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.

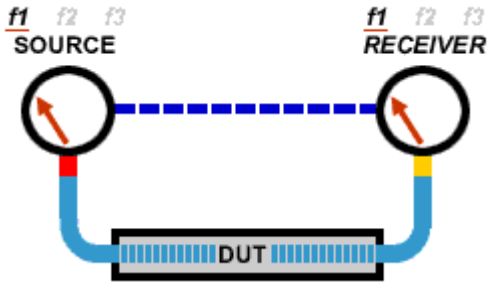


Figure 6.6 Long test cable

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 IF BW

Menu path: **[Avg]→ [IFBW]**.

Enter a value directly or press the arrow button to select a value.

c) Increase the 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.

- Electrical delay.....209
- Port Extensions.....209
- Phase Offset.....212
- Spacing of frequency points.....213
- Specific Operations.....214

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 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...]**.

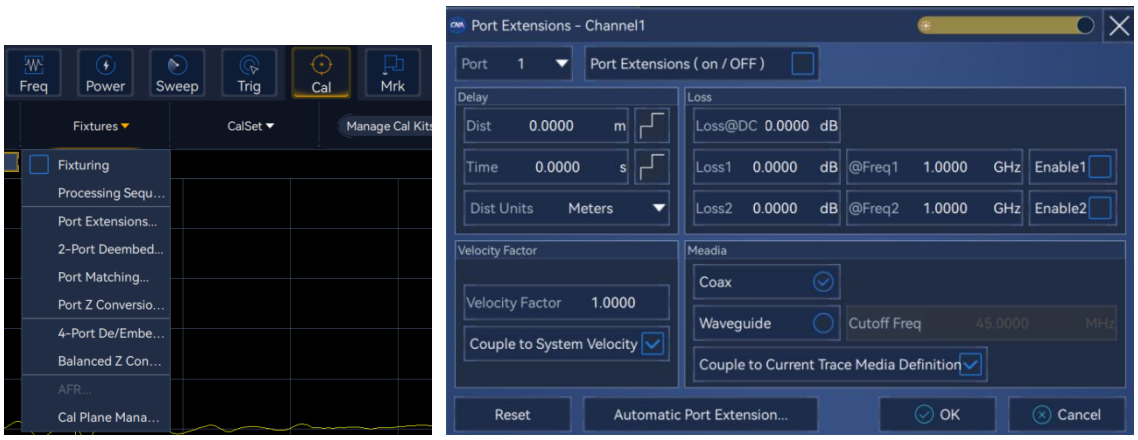


Figure 6.7 Set Port Extension

The Port Extension dialog box is displayed. Click the [Port EXT] checkbox. 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:

  Loss (f) = Loss1*(f/Freq1)^0.5

  When both Loss 1 and Loss 2 are selected, compensation is performed using the following equation:

  Loss (f) = Loss1*(f/Freq1)^n , where

  n = log10(|Loss1/Loss2|)/log10(Freq1/Freq2)
- [Velocity Factor]: Set the velocity factor. Note that the velocity factor set here is the same 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.

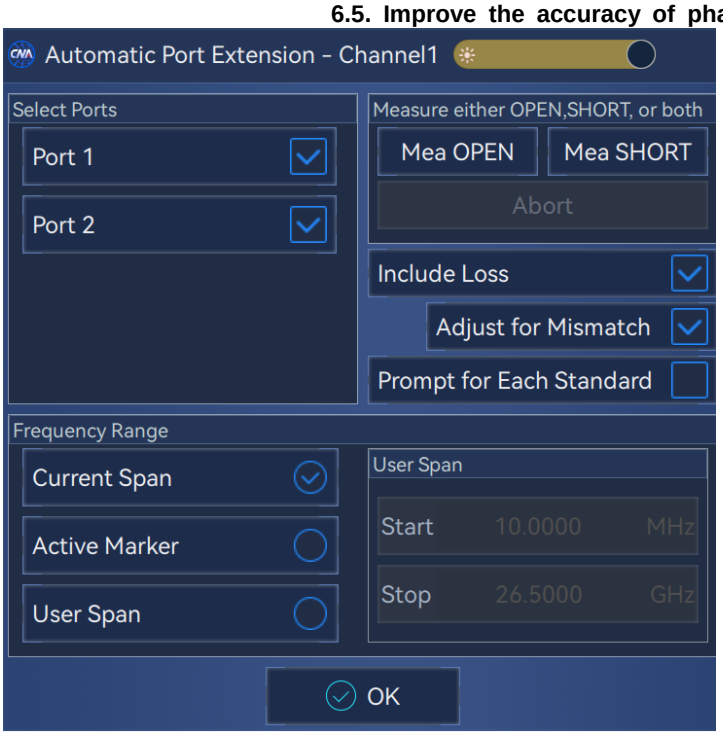


Figure 6.8 Auto port extension

- [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.

  [DEF Range]: The same frequency range as the current measurement setting.

  [Mkr Scope]: Automatic port extension calculation using the data between the active marker 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 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.

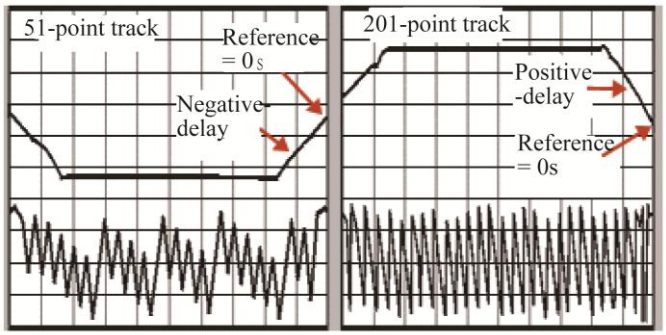


Figure 6.10 Effect of frequency spacing on phase measurement

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
- 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 expansion 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 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……216
- Trace Smooth……216
- IFBW……217

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.

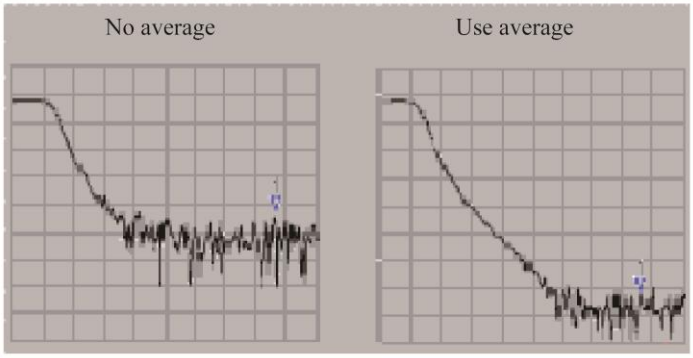


Figure 6.12 Effect of sweep average on measurement results

3) Sweep Average Setting

Menu path: **[Averaging]**→［**Averaging On Off**］.
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

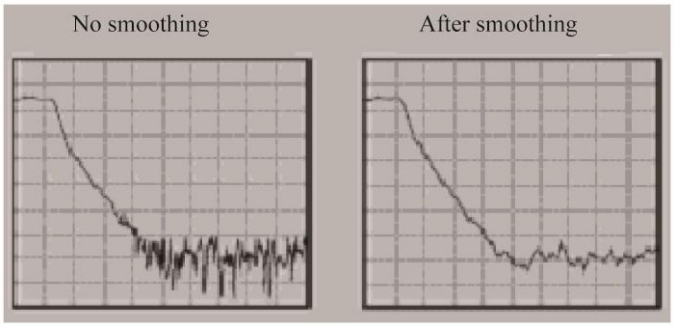


Figure 6.14 Effect of trace smoothing on measurement results

2) Trace smoothing setting

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:

a) In **Smoothing Percent** box, enter a percentage value for smoothing. (max. 25%)

b) 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)

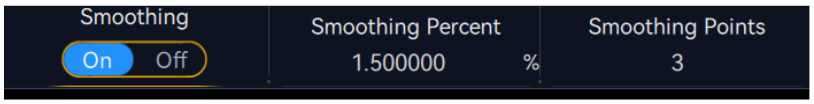


Figure 6.15 Setting smoothing

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:

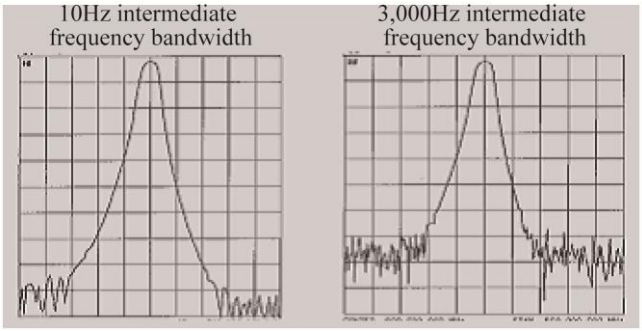


Figure 6.16 Effect of IF bandwidth on measurement results

2) Setting 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 allows for better sweeping of the trajectory and thus better definition of 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 ± 1 ℃.

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) Equipment connector

Good connectors are a must for repeatable measurements. For a good connection, the 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 ± 10ppm
b) Input level: -15 ~ +20dBm
c) Input impedance: 200 ohms

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 ℃ ± 3 ℃ (unless otherwise specified).
c) Ensure that the deviation between the ambient temperature and the measured calibration temperature is ± 1 ℃.

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 sweeping speed

The fastest sweeping speeds will increase measurement efficiency, and the following settings are optimized for the microwave analyzer to achieve the fastest sweeping speed:

1) Sweep setting

Get the fastest sweeping 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) Noise reduction setting

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) Selecting 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**
- **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.

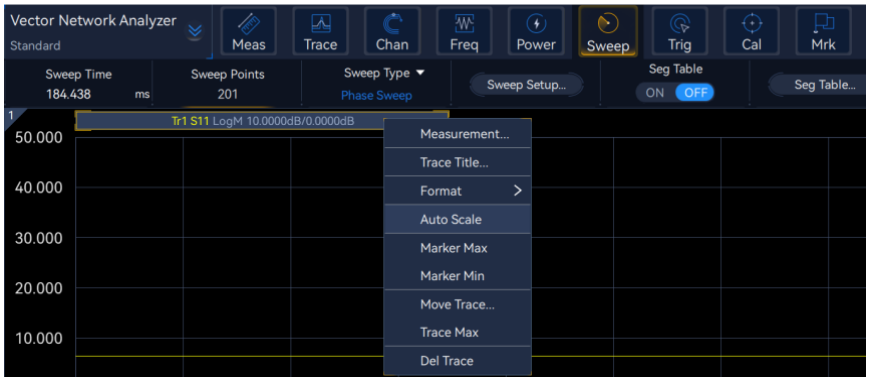


Figure 6.19 Delete undesired traces

b) Disable the unused marker

Menu path: **[Mrk]→[Mrk]**;
Select the marker to be closed and uncheck.



Figure 6.20 Closing an unused marker

c) Disable the smoothing function

Menu path: **[Avg]→[Smooth]**;
Close **[Smooth]** switch to deactivate function.

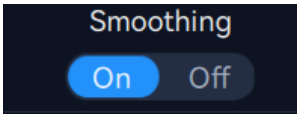


Figure 6.21 Turning-off smoothing function

d) Turn off limit test

Menu path: **[Analy]→[Test]→[LIM Test...]** to display the **Limit Test** dialog box.
Click [LIM Test ON/off] to clear the checkbox to turn off the limit test function.

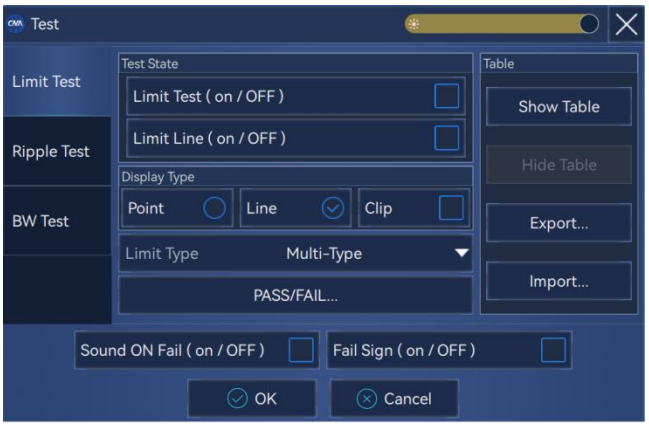


Figure 6.22 Turn off limit test

e) Turn off the trace operation function

Menu path: **[Analysis]→ [Trace Operation]**;
Select **Off**.

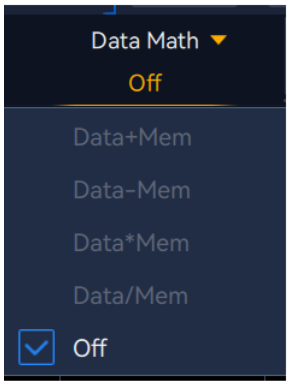


Figure 6.23 Turning-off the trace operation function

6.10 Improving 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.

- Improving measurement efficiency via measurement settings.....225
- Automatic change of measurement settings.....226
- Fast call of measurements.....226

6.10.1 Improving 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 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) Usage 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) Using 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.

Select the **[INTL Trig]** radio box in the **Trigger Source** zone.

In **Channel Trigger State** zone, click **[Channel]** box and select **Channel**1.

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.....227
- Transmit As Little Data As Possible.....227
- Use Real Format.....227
- Use LAN.....227
- Use COM Programs.....228

6.11.1 Using 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 Transmitting 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 Using 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 Using 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 3674 series vector network analyzer supports up to 10 macros.

- New Macro.....228
- Macro Settings dialog box.....229

6.12.1 New Macro

Menu path: **[SYS]→[Macro]→[Macro Set...]** to display the **Macro Setting** dialog box.

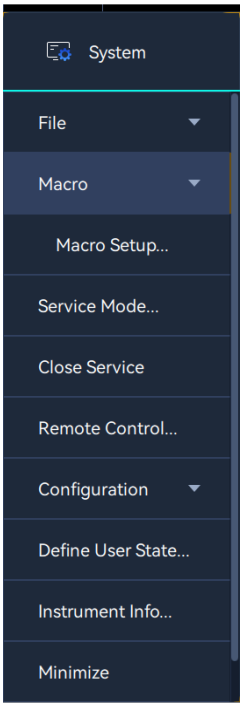


Figure 6.24 Set macro

In **Macro Settings** dialog box, click to activate the blank line below the existing macro settings, and check**[Edit]** to edit the macro.

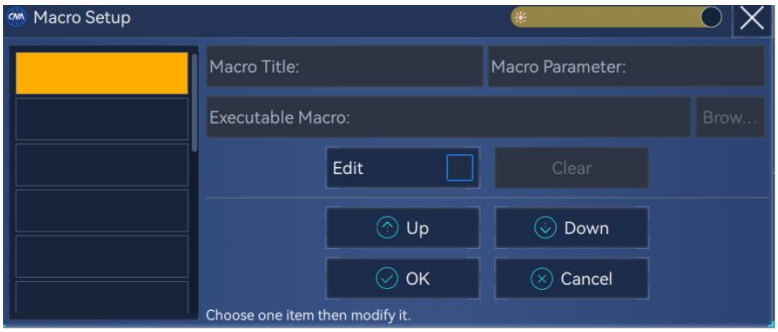


Figure 6.25 Macro Settings dialog box

- 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 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 Parameter] 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.....231
- Select Calibration Type.....233
- Calibration Wizard.....236
- Calibration Set.....245
- Measurement Error.....250
- Edit Calibration Kit Definition.....257
- Calibration Standard.....268
- TRL Calibration.....270
- Fixture Compensation Calibration.....271

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.....231
- Significance of Calibration.....232
- Applications of Calibration.....232
- Simple Process of Calibration.....232

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".

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 Selecting calibration type

There are 9 calibration types commonly used by 3674 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

c) Required calibration standard: through device, two loads (one load per port)
d) Corrected system error:
   - Transmission tracking
   - crosstalk
e) Measurement application:
   - 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 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 application:
   - 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 application:
   - Only one-side port and through standard measurements are performed to complete all error correction of full dual ports.
   - Simplify reflection standard measurement

8) Full dull-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 application:
   - Measurement of all S parameters
   - It is necessary to improve the measurement accuracy through 12 error corrections.

9) Full dull-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

- Transmission tracking

e) Measurement application:

- 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 program 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.....236
- Perform Calibration Settings.....237
- Modify Calibration Method.....239
- Perform Source Calibration Settings.....241
- Measurement Standard.....242
- Complete Calibration.....244
- Save As User Calibration Set.....244

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.

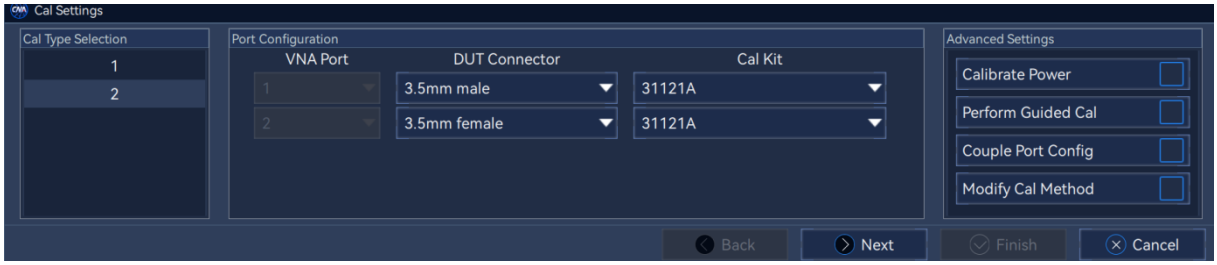


Figure 7.2 Cal Settings interface (with an electronic calibration kit connected)

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.

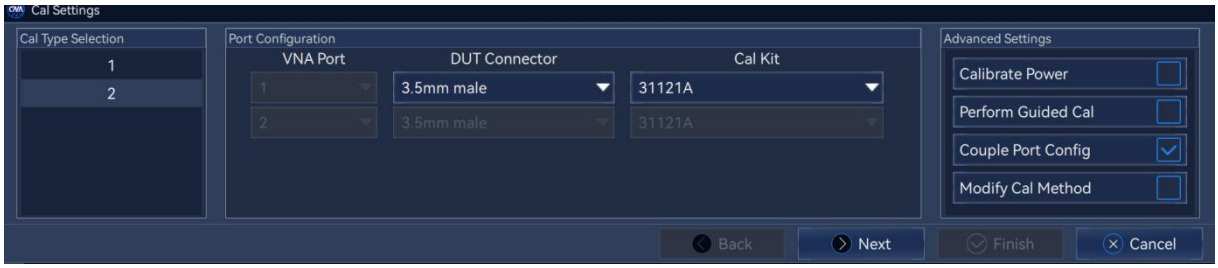


Figure 7.3 Cal Settings interface (with no electronic calibration kit connected)

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".

[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 guidance 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]**.

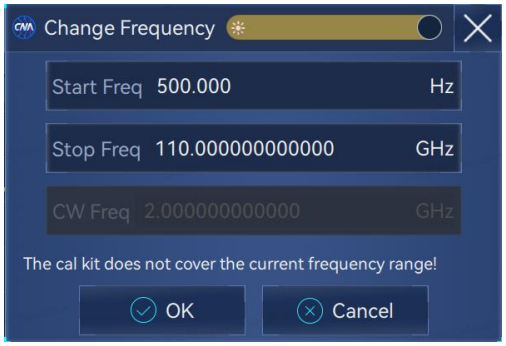


Figure 7.4 Change Frequency interface

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.

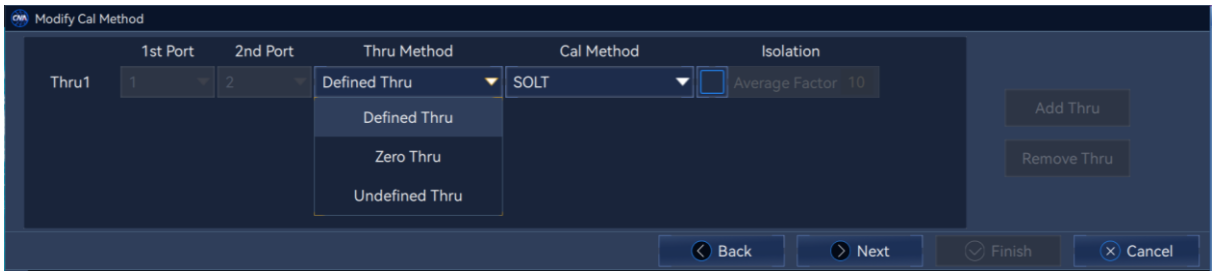


Figure 7.5 Modify through method

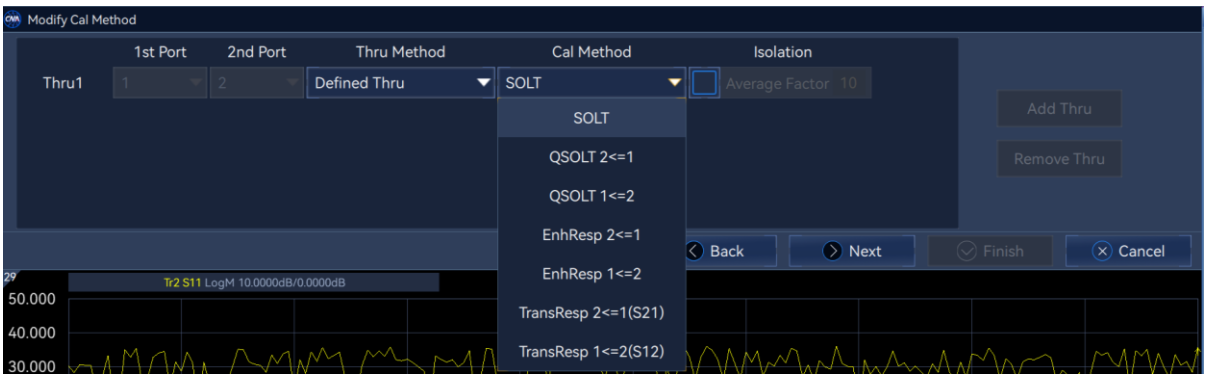


Figure 7.6 Modify calibration method

When **[Calibration Ports]** is set to N, the minimum number of measurement through is N-1, the maximum number of measurement through is N×(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(ECal): 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(ECal): 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(ECal), 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(ECal). 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(ECal). 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 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-embed 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-embed 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-embed 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-embed calibration will not be performed to the adapter even if the **[De-embed Adapter]** is

checked.

[Cal Kit]: When **[Connector]** is set as the connector type of the existing calibration kit, the 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 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 (with measurement in progress)

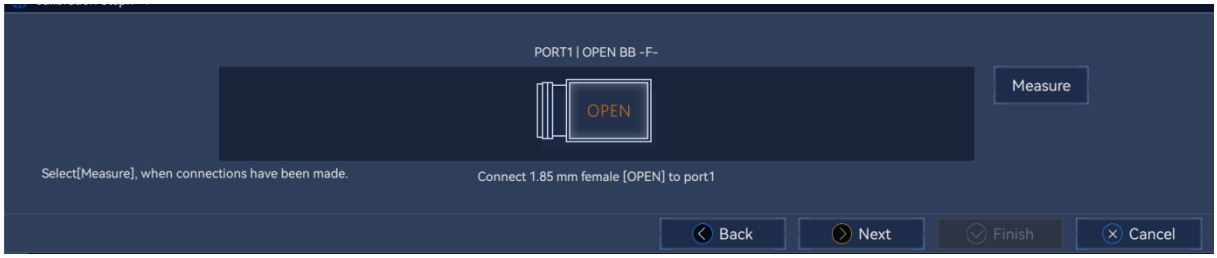


Figure 7.10 Calibration Kit Measurement (Step-by-step) interface

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.

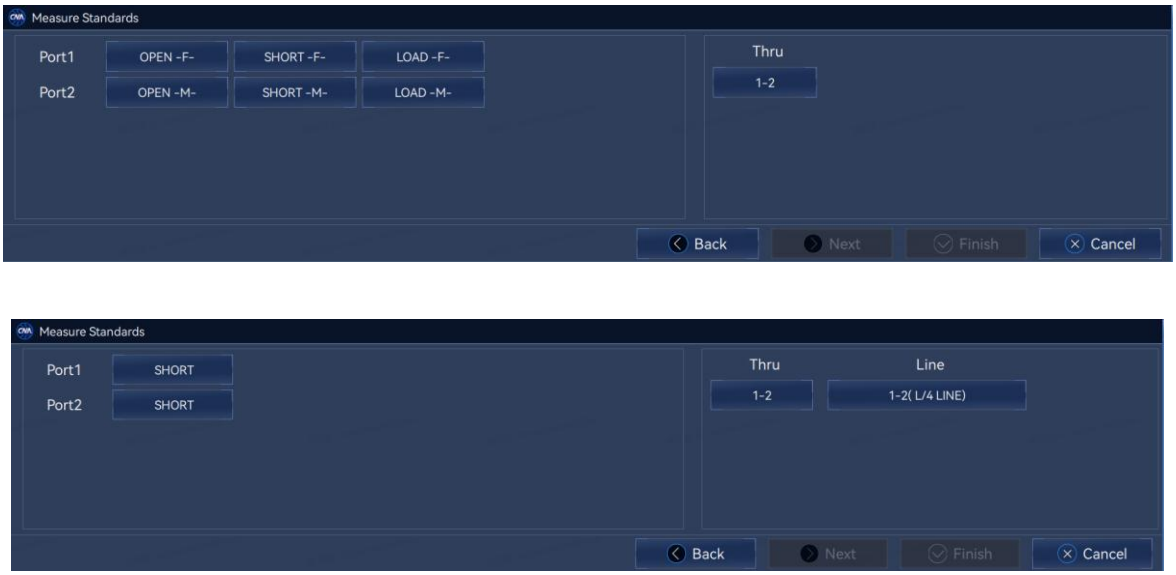


Figure 7.11 Calibration Kit Measurement (Simplified) interface

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.

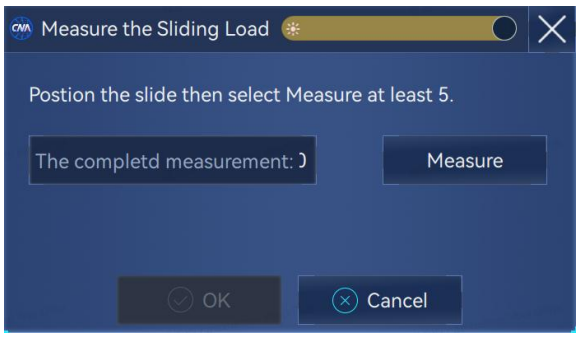


Figure 7.12 Calibration Kit Measurement (Sliding Load) interface

When a Calibration Kit Measurement interface contains a sliding load standard, the Calibration Kit Measurement (Sliding Load) interface will pop up as shown in Figure 7.12 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 Finish 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.

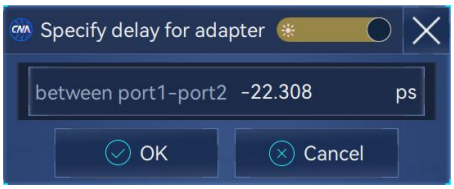


Figure 7.14Adapter Delay Confirmation interface

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.

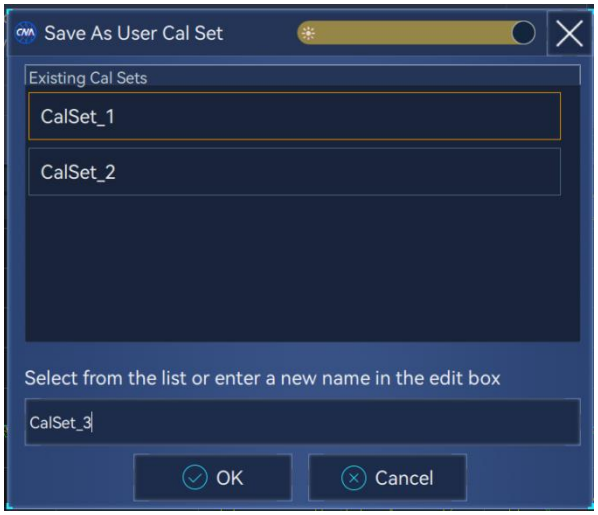


Figure 7.15 Save as User Cal Set interface

[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.

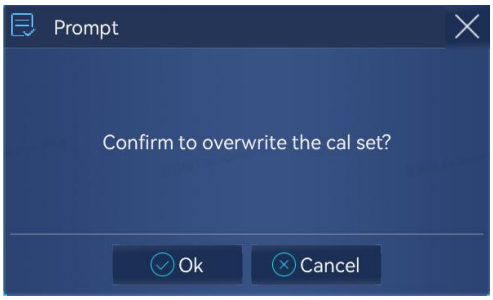


Figure 7.16 Prompt to overwrite previous calibration set

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.

- Definition of Calibration Set.....246
- Display Calibration Set Properties.....247
- Save Calibration Set As.....248
- Delete Calibration Set.....249
- Delete All Calibration Sets.....249
- Apply/Unapply Calibration Set.....249

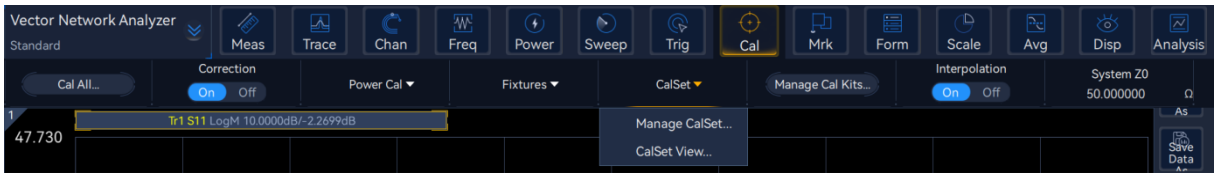


Figure 7.17 Open calibration set

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.

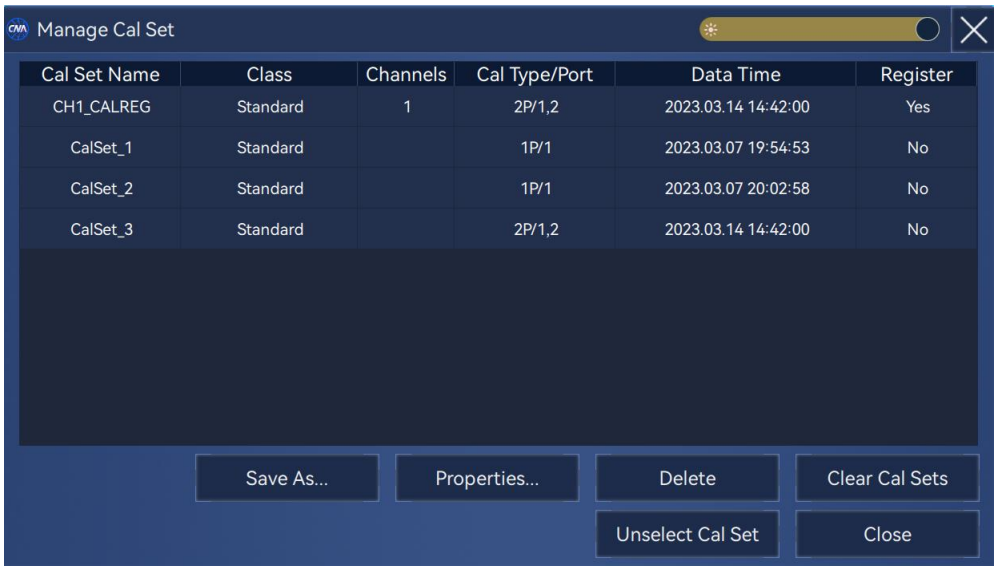


Figure 7.18 Calibration Set dialog box

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.

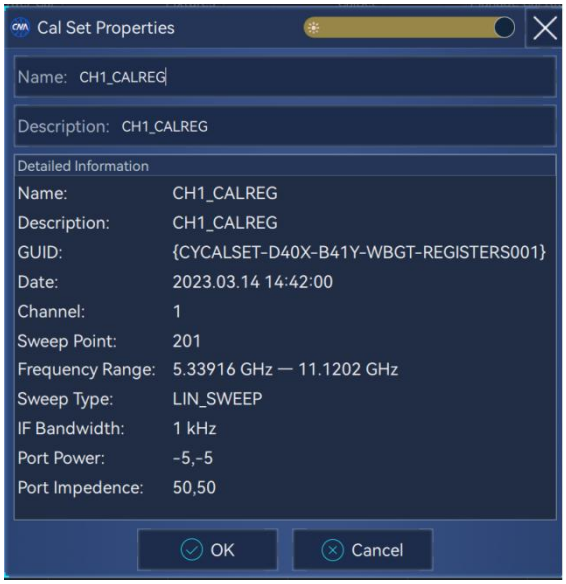


Figure 7.19 Cal Set Properties dialog box

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.

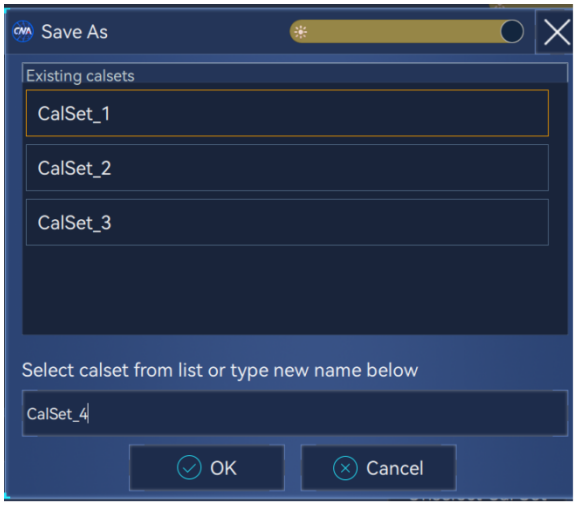


Figure 7.20 Save Calibration Set As

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 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.

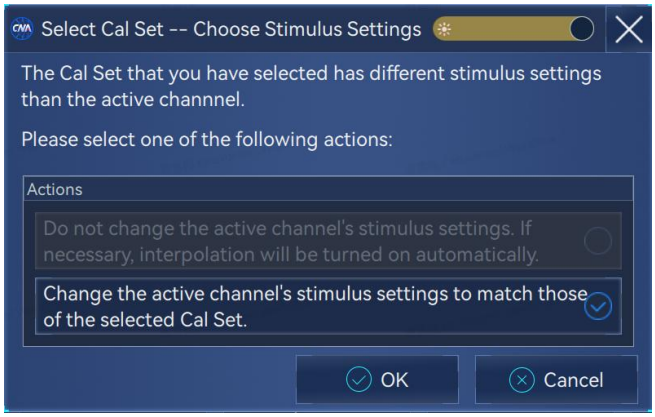


Figure 7.21 Choose stimulus settings

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.

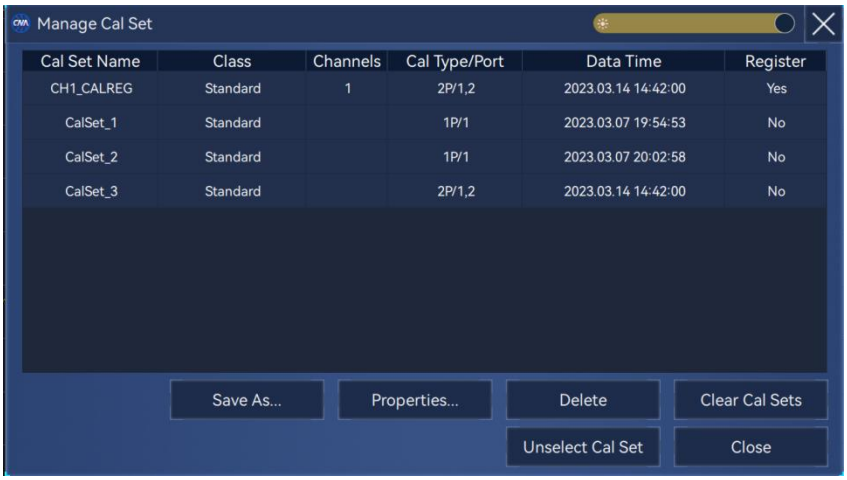


Figure 7.22 Unapply calibration set

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.....251

- Random error.....251
- Systematic error.....252
- High-precision Measurement Calibration.....252

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

For transmission measurements, the relevant residual errors include:

- crosstalk
- Effective load matching
- Effective transmission tracking

1) Directivity 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.

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 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 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 introduces 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.....257
- User-defined Calibration Kit.....257
- Create Calibration Kit.....258
- Edit Calibration Kit.....259
- Edit Calibration Kit Dialog Box.....260
- Edit Connector Dialog Box.....261
- Class Information Dialog Box.....262
- Add Standard Dialog Box.....263
- OPEN Dialog Box.....264
- SHORT Dialog Box.....265
- Load Dialog Box.....266
- Thru/Line Dialog Box.....267
- Databased Standard.....267

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:

   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], [SHORT], [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 **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…]

Calls the **Open** dialog box to import the calibration kit definition on the hard disk or other drive.

4) [Save…]

Open the **Save As** dialog box to save the selected calibration kit definition.

5) [Add…]

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) [Restore Defaults]

Reinstall all default calibration kits of the vector network analyzer.

7.6.5 Edit Calibration Kit Dialog Box

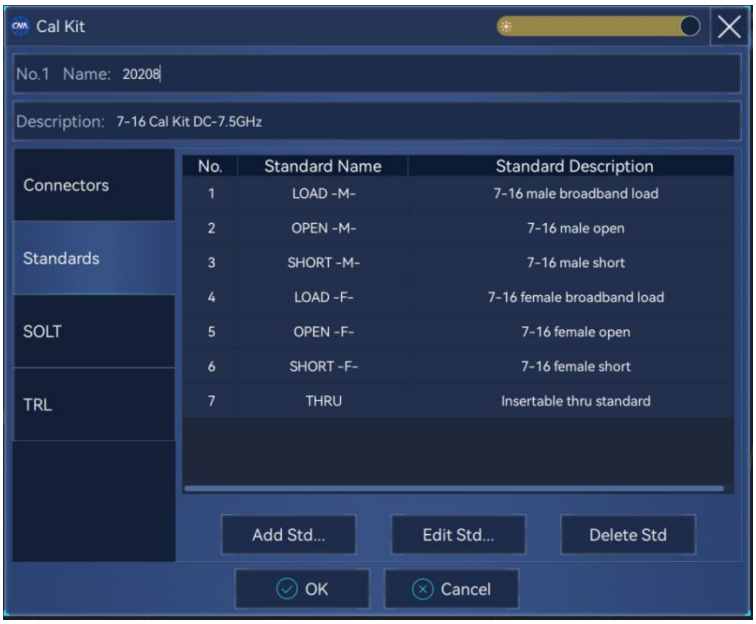


Figure 7.24 “Edit calibration kit” dialog box

1) Cal Kit ID area

a) **[**Calibration Kit Name] box
Display or edit calibration kit SN and name.

b) [**Cal Kit Description**] box
Display or edit the characteristic description of the calibration kit.

2) Connector Type area

a) **[Connector]** box
Click the **[Connector]** box or the **Arrow** button to select the connector type.

b) **[Add...]**
Click the **[Add Connector...]** button to display the **Add Connector** dialog box to add a new connector type for the calibration kit.

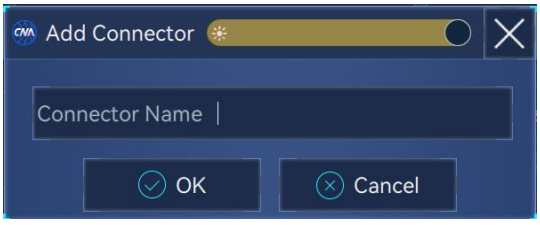


Figure 7.25 Add Connector dialog box

3) Class area

a) **[Calibration Method]** box
Click **[SOLT]** box or **[TRL]** box to select the calibration method assigned by class.

b) **[Class]**
Click the **[Cal Class]** button to display the **Class Information** dialog box to set the class of calibration kits.

4) Std area

a) **[Add Std…]**
Click this button to display the **Add Std** dialog box to define the new calibration standard.

b) **[Edit Std…]**
Click the **[Edit Standard...]** button to display the Cal Std **(such as OPEN, SHORT, Load, THRU/Air Line)** dialog box to modify the definition of the selected calibration standard.

c) **[Delete Std]**
Delete the definition of the selected calibration standard.

7.6.6 Edit Connector Dialog Box

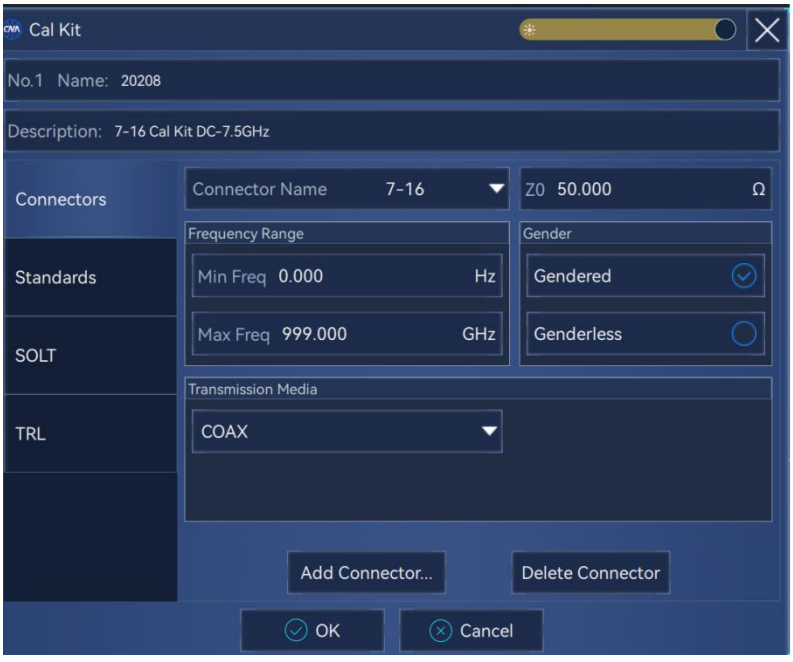


Figure 7.26 Add or Modify Connector dialog box

1) Characteristics zone

a) **[Connector]** box

It is used to define the name of calibration kit connector.

2) Frequency range zone

a) **[**Min. 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 area

Select the gender of the calibration kit connector between **[Gendered]** and **[Genderless]**.

4) Impedance area

Define the characteristic impedance of the calibration standard.

5) Media area

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

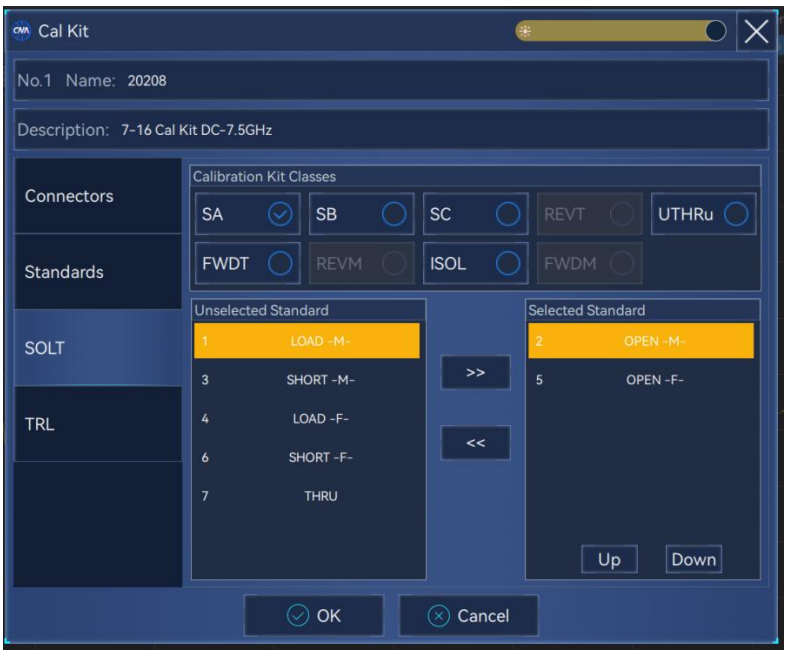


Figure 7.27 Class Information dialog box

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 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) [>>] and [<<] 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

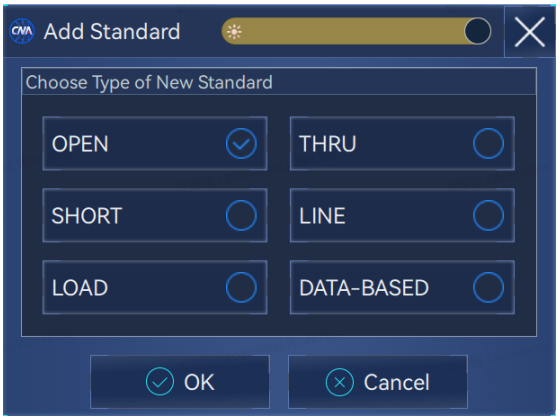


Figure 7.28 Add Std dialog box

Standards supported by the calibration kit include **[OPEN], [SHORT], [LOAD], [THR], [LINE]** and **[DATA-BASED]**.

7.6.9 OPEN Dialog Box



Figure 7.29 OPEN 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) **[**Min. 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 area

Define the OPEN connector, which can be selected in the drop-down box**.**

4) Open characteristic area

[C0], **[C1]**, **[C2]** or **[C3]** are used to define the fringe capacitance of the open circuit breaker.

5) Delay characteristic area

a) **[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:

$$Loss(\frac{\Omega}{S}) = \frac{Loss(\mathrm{dB}) \times Z_0(\Omega)}{4.3429(\mathrm{dB}) \times \mathrm{Delay(S)}}$$

c) **[Impedance]** box

Define the impedance of the standard.

7.6.10 SHORT Dialog Box



Figure 7.30 SHORT dialog box

1) Short characteristic area

[L0], **[L1]**, **[L2]** and **[L3]** are used to define the residual inductance of the short circuit breaker.

7.6.11 Load Dialog Box

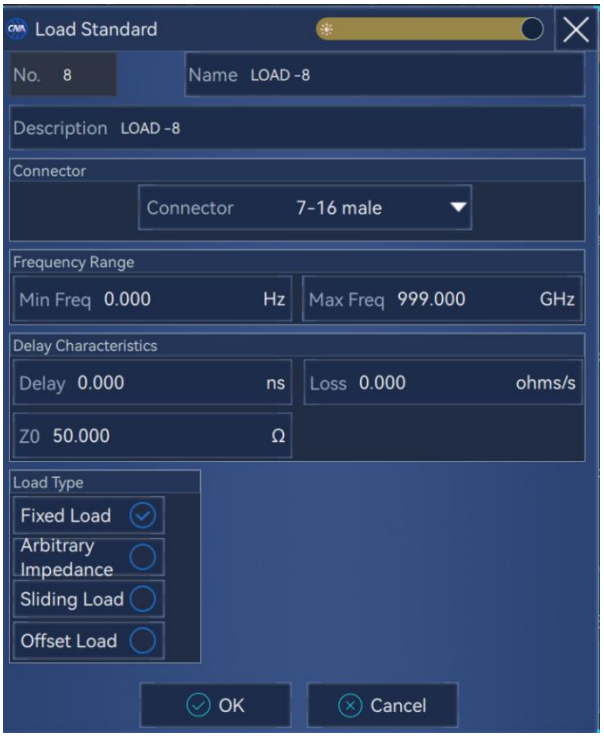


Figure 7.31 Load dialog box

1) Load Type area

a) **[Blocked 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 Z0. Only when **[Specific Impedance]** is selected, the setting of **Complex Impedance** area is effective.

2) Complex Impedance area

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



Figure 7.32 Through/Line Standard Dialog Box

1) Connector area

[Connector1] box and **[Connector2]** box define the types of the connectors of the two standard ports.

7.6.13 DataBased Standard Dialog Box



Figure 7.33 DataBased Standard dialog box

1) Update Standard Data area

[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 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	∞Ω
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) Min. frequency

Specifies the minimum frequency that a standard can use during calibration.

b) Max 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, SHORT, 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:

$$Loss(\frac{\Omega}{S}) = \frac{\mathrm{Loss(dB)} \times \mathrm{Z_0}(\Omega)}{4.3429(\mathrm{dB}) \times \mathrm{Delay(S)}}$$

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×F)+(C2×F2)+(C3×F3) (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×F)+(L2×F2)+(L3×F3) (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) Why 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

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.....272
- Fixturing.....273
- Processing Sequence.....273
- Port Extensions.....274
- 2-Port De-embed.....279
- Port Matching Circuit Embed.....281
- Port Z Conversion.....285
- 4-Port De/Embed.....285
- AFR.....288
- Cal Plane Manager.....293

The entire fixture simulation process is shown in 0.

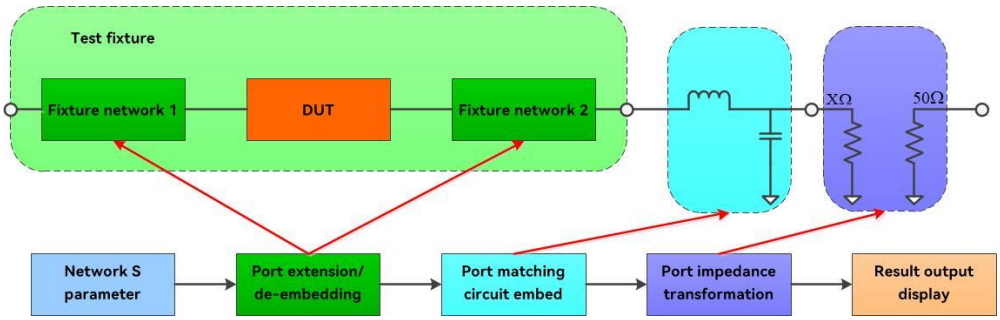


Figure 7.34 Fixture simulation process

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 embed/de-embed 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 Fixturing

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 empty, indicating that the **Fixture Simulation** function is off.

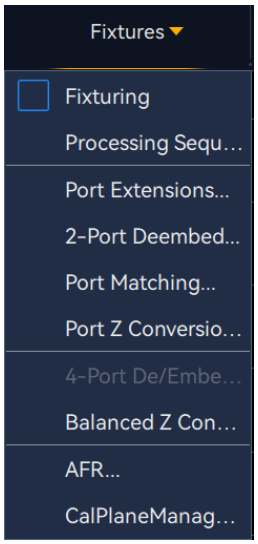


Figure 7.35 Enable fixture simulation function

7.9.3 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.

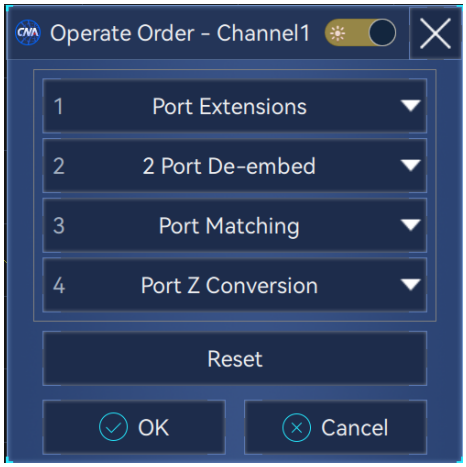


Figure 7.36 Fixture processing sequence

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 Extensions dialog box

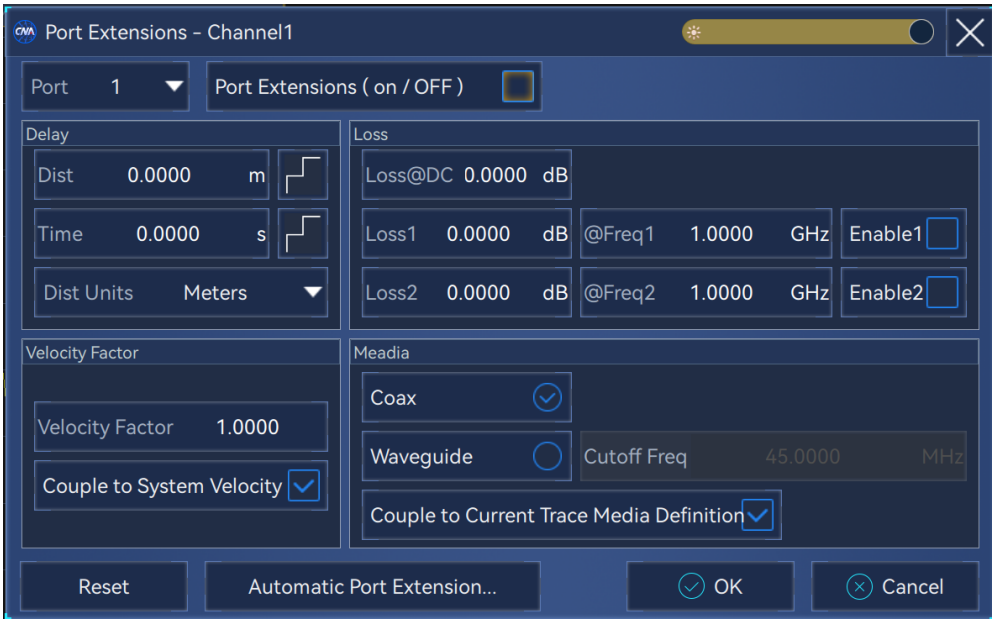


Figure 7.37 Set Port Extension

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 Extensions (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) **[Dist]**

Set the physical length of port extensions for the current port.

e) **[Dist Units]**

Set the distance calculation unit of port extensions for the current port among meters, feet and inches.

f) **[Loss@DC]**

Set the full band loss offset of the current port.

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:

Loss (f) = Loss1*(f/Freq1)^0.5

When both Loss 1 and Loss 2 are selected, compensation is performed using the following equation:

Loss (f) = Loss1*(f/Freq1)^n , where

n = log10(|Loss1/Loss2|)/log10(Freq1/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.

l) **[Couple to System Velocity]**
Set to couple the velocity factor of current port to the velocity factory of the system

j) **[Coax]**
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 Freq]**
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) Port extension steps

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Port Extensions...]** to display the **Port Extensions** dialog box.

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



Figure 7.38 Auto port extension

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) **[Mea OPEN]**

Click **[Mea OPEN]** when the automatic extension port is connected with the open standard.

c) **[Mea SHORT]**

Click **[Mea SHORT]** when the automatic extension port is connected with the short standard.

d) **[Abort]**

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) **[Adjust for Mismatch]**

Automatically compensate the error introduced by the mismatch.

g) **[Prompt for Each Standard]**

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) **[Active Marker]**

Automatically set the frequency range to the frequency between the two active markers.

j) **[User Span]**

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) Auto Port Extend steps

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-embed

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.

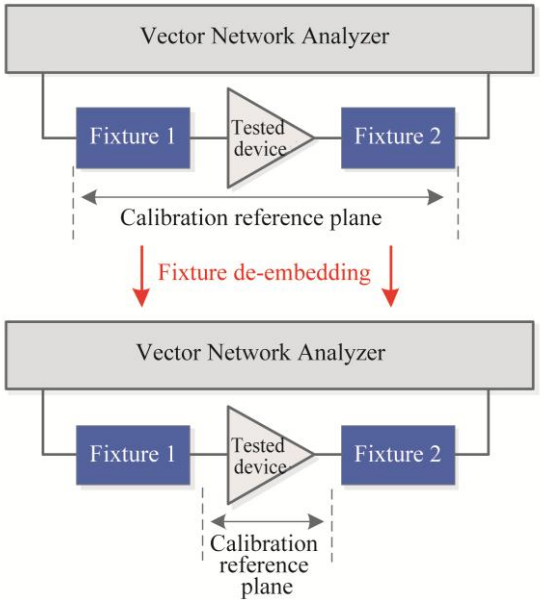


Figure 7.39 Example of fixture de-embedding

1) 2-port De-embed dialog box

a) **[Select Ports]** drop-down menu

Select the port number of the network analyzer requiring fixture de-embedding.

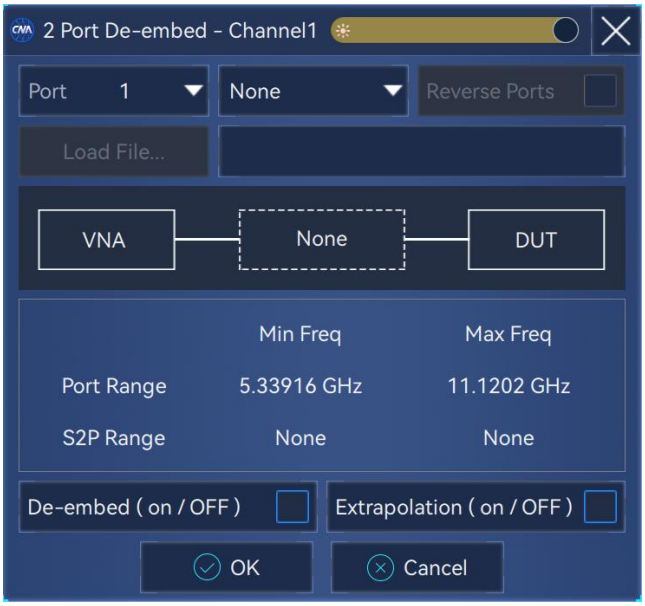


Figure 7.40 2-port De-embed dialog box

b) **[De-embed Type]** drop-down menu
In the drop-down menu to the right of the **Select Port**s drop-down menu, select the de-embed type. This network analyzer provides **User** mode for the de-embed type. When a different type is selected, the corresponding display in example diagram will changes.

c) **[Load File]**
This button is ineffective by default. When **[User]** is selected in the **[De-embed 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 area 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-embed 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) **[De-embed on/OFF]** check Box
When checking, turn on the two-port fixture de-embedding function.

f) **[Extrapolation on/OFF]** check Box
When this check box is checked, the extrapolation function is turned on.

g) **[OK]**
Click **[OK]** to apply the settings in this dialog box to the currently active channels.

h) **[Cancel]**
Click **[Cancel]** to exit the 2-port de-embed function.

2) 2-port fixture de-embedding steps

Menu path: Click **[Cal]→[Fixture]**, and in the **[Fixture]** submenu, click **[2-Port De-embed...]** to display the 2-Port De-embed 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 Port**s drop-down menu, select the de-embed type. This network analyzer provides **User** mode for the de-embed type.

When **[User]** is selected as the de-embed 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-embed on/OFF]** in the dialog box to turn on the 2-port de-embed function.

Check **[Extrapolation on/OFF]** in the dialog box to turn on the extrapolation function.

Finally, click **[OK]** to complete the setting.

7.9.6 Port Matching Circuit Embed

Function introduction: As shown in 0, the port matching circuit embedding function can convert the 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.

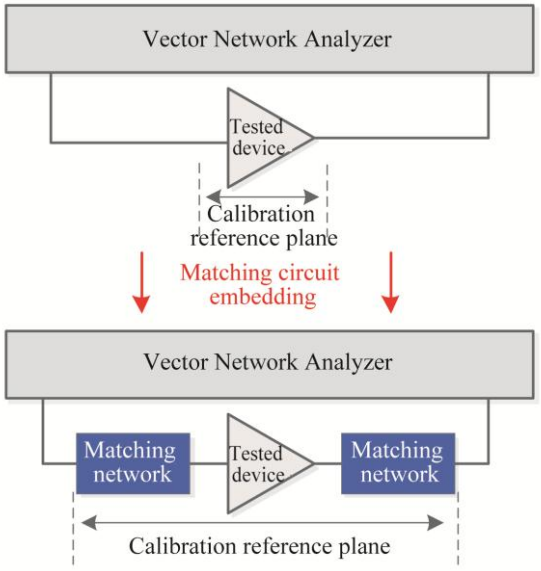


Figure 7.41 Port matching circuit embedding

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]**

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.

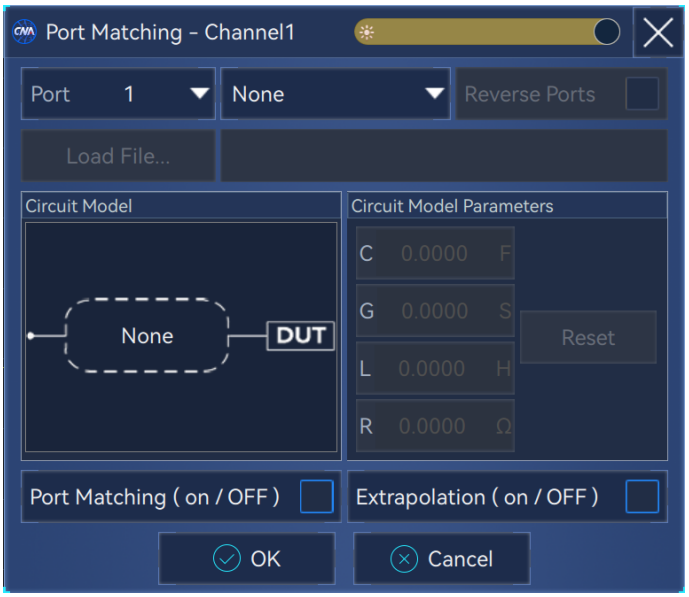


Figure 7.42 Port Matching Circuit Embed interface

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.

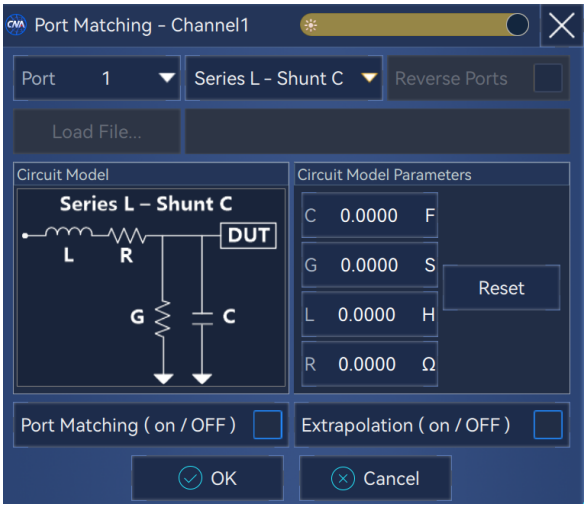
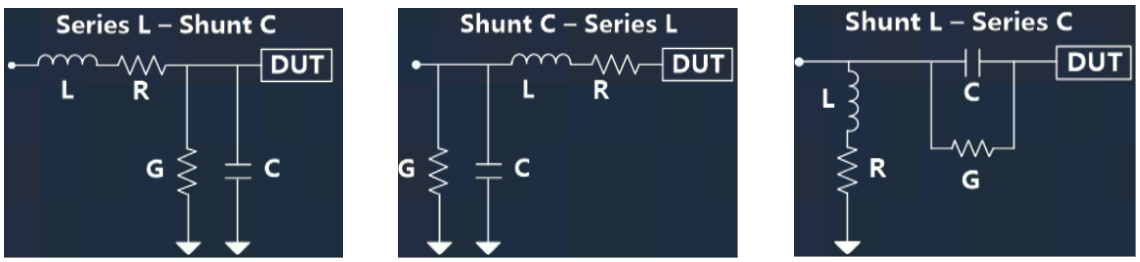
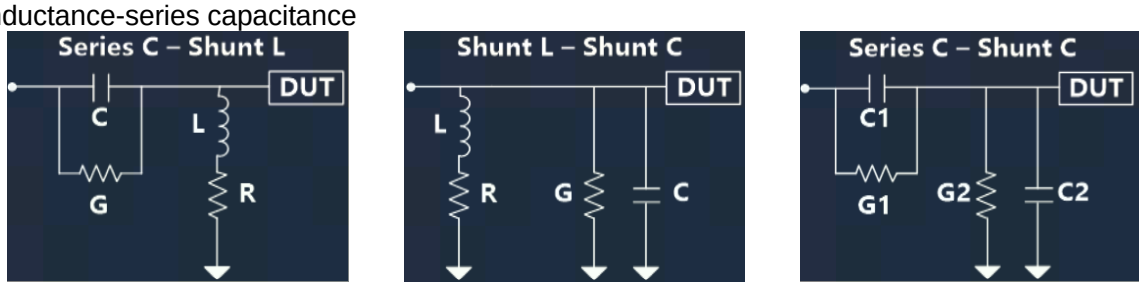


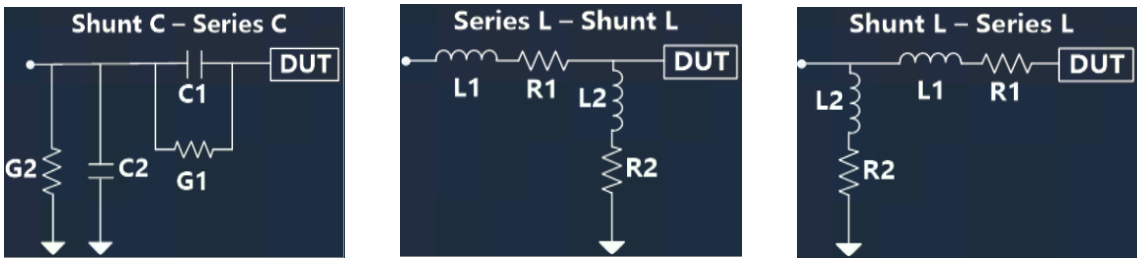
Figure 7.43 Matching Circuit Type drop-down menu



(1) Series inductance-shunt capacitance (2) Shunt capacitance-series inductance (3) Shunt inductance-series capacitance



(4) Series capacitance-shunt inductance (4) Shunt inductance-parallel capacitance (5) 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 Setting]** edit boxes

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) **[Reset]**

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) **[Port Matching on/OFF]** check box

Check this box to turn on the port matching circuit embedding function.

h) **[Extrapolation on/OFF]** check Box

When this check box is checked, the extrapolation function is turned on.

i) **[OK]**

Click [OK] to apply the settings in this dialog box to the currently active channels.

j) **[Cancel]**

Click **[Cancel]** to exit the port matching circuit embedding function.

2) Port matching circuit embedding steps

Menu path: Click **[Cal]**→**[Fixture]**, and in the **[Fixture]** submenu, click **[Port Matching...]** to display the Port Matching Circuit Embed dialog box.

Click **[Select Ports]** drop-down menu, and select the port requiring matching circuit embedding.

In the drop-down menu to 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**a nd **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 **[Port Matching on/OFF]** in the dialog box to turn on the port matching circuit embedding function.

Check **[Extrapolation on/OFF]** in the dialog box to turn on the extrapolation function.

Click **[OK]** to complete the setting.

7.9.7 Port Z Conversion

Function introduction: Port z conversion converts the measured port impedance equal to system impedance to any impedance.



Figure 7.45 Port Z Conversion interface

1) Port Z Conversion dialog box

a) **[Select Ports]** drop-down menu
Select the port number of the network analyzer requiring port Z conversion.

b) **[Real/Imaginary]** edit boxes
Modifying the parameters in these edit boxes will set the impedance to be converted to.

c) **[Port Z Conversion on/OFF]**check Box
Check this check box to turn on the port Z conversion function.

d) **[OK]**
Click [OK] to apply the settings in this dialog box to the currently active channels.

e) **[Cancel]**
Click **[Cancel]** to exit the port Z conversion function.

2) Port Z conversion steps

Menu path: Click **[Cal]→[Fixture]**, and in the **[Fixture]** submenu, click **[Port Z Conversion...]** to display the **Port Z 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 Z Conversion on/OFF]** in the dialog box to turn on the port Z conversion function.

Click **[OK]** to complete the setting.

7.9.8 4-port De-embed/Embed

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.

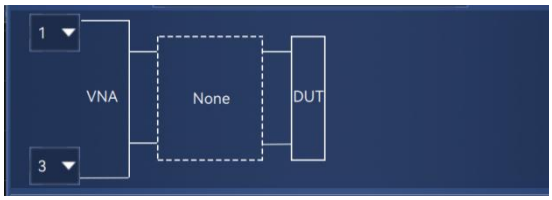
1) 4-port De-embed/Embed dialog box

a) **[Select Models]** drop-down menu

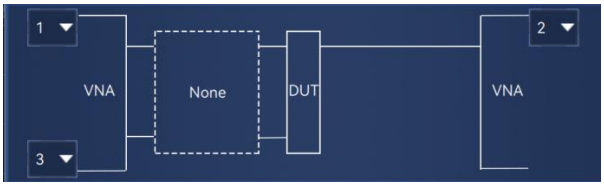
Select the fixture de-embedding/embedding model. As shown in 0, the vector network analyzer is designed with 3 network models (i.e. **Model A, B, C**).



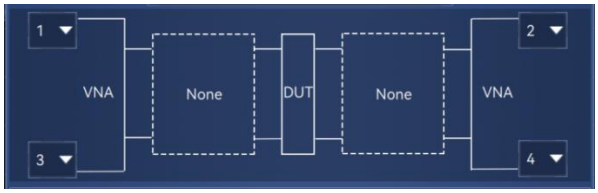
Figure 7.46 4-port De-embed/Embed interface



Model A



Model B



Model C

Figure 7.47 Three models for 4-port fixture de-embedding/embedding

b) **[Network De-embed/Embed Type]** drop-down menu

The **Network1** (**Network2** is on when Model C is selected) (i.e. the first 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]**

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 Embed/De-embed on/OFF]** check box

Check the 4-port de-embedding/embedding function of fixture.

f) **[Extrapolation 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]**

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) 4-port de-embedding/embedding steps

Menu path: Click **[Cal]→[Fixture]**, and in the **[Fixture]** submenu, click **[4-Port De-embed/embed...]** to display the 4-Port De-embed/Embed 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 on when Model C is selected).

When the network is in **Embed** or **De-embed** 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 **[Extrapolation 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.

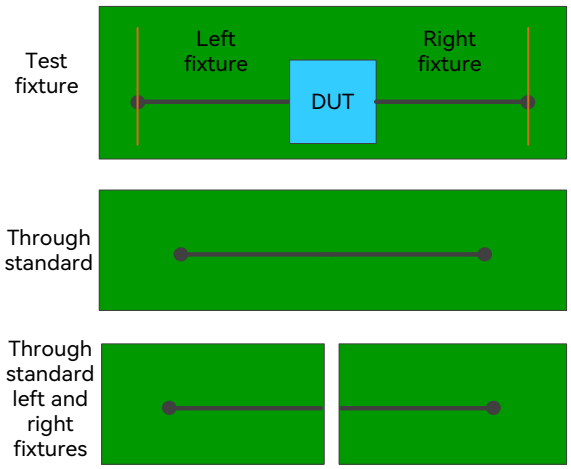


Figure 7.48 Single-ended fixture

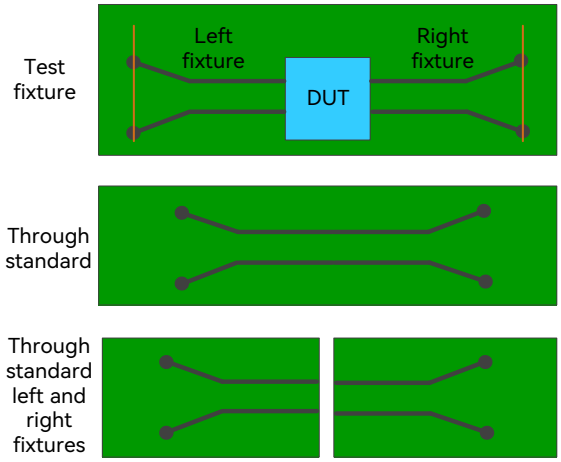


Figure 7.49 Differential fixture

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.

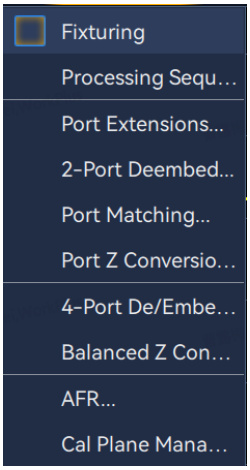


Figure 7.50 Enable AFR

The specific operation steps are as follows:

1) Describe fixture.



Figure 7.51 Describe Fixture 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.

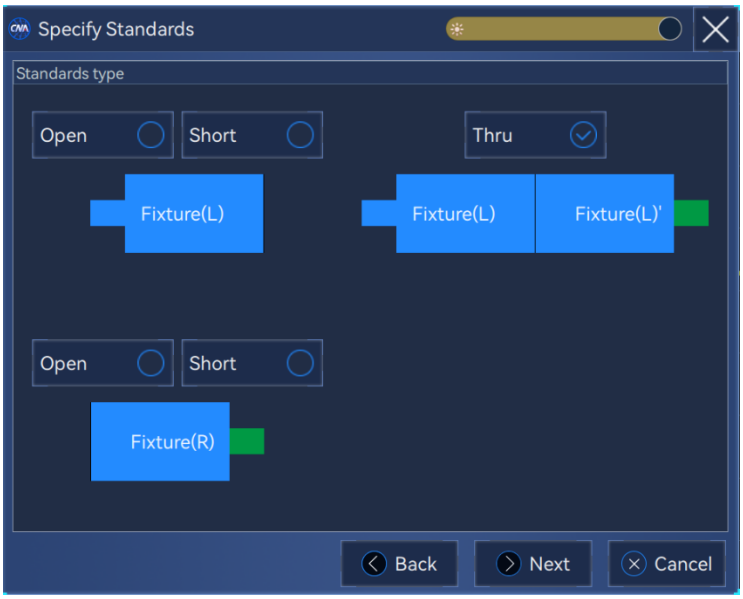


Figure 7.52 Specify Standards interface

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) Measure standards.

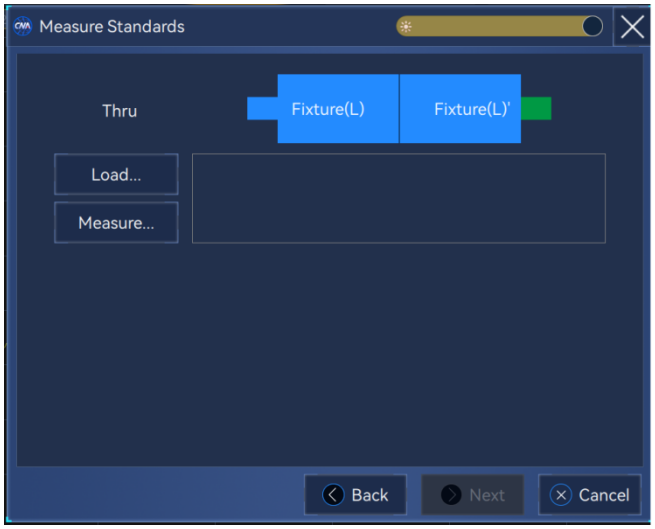


Figure 7.53 Measure Standards Interface

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 Embed interface.

4) Remove the fixture.

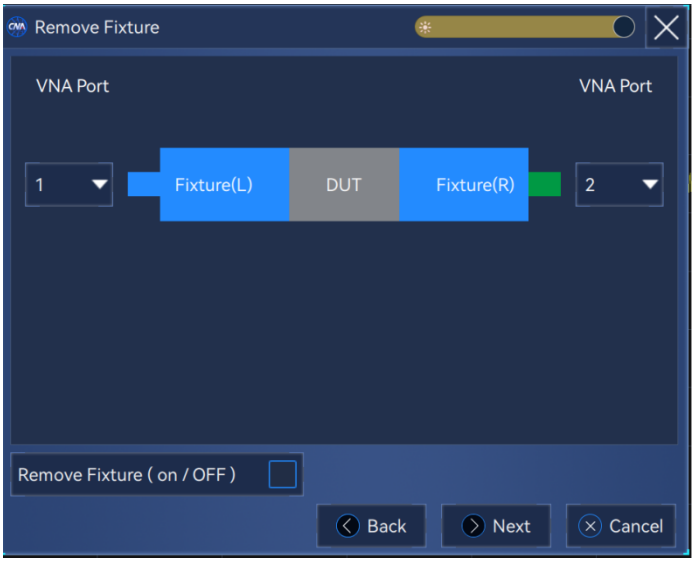


Figure 7.55 Remove Fixture interface

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.

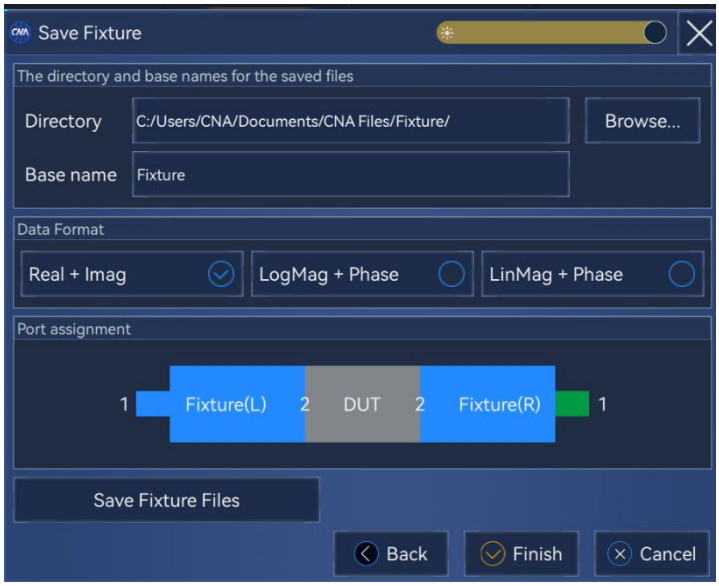


Figure 7.56 Save Fixture interface.

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 (S21 = S12).

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.

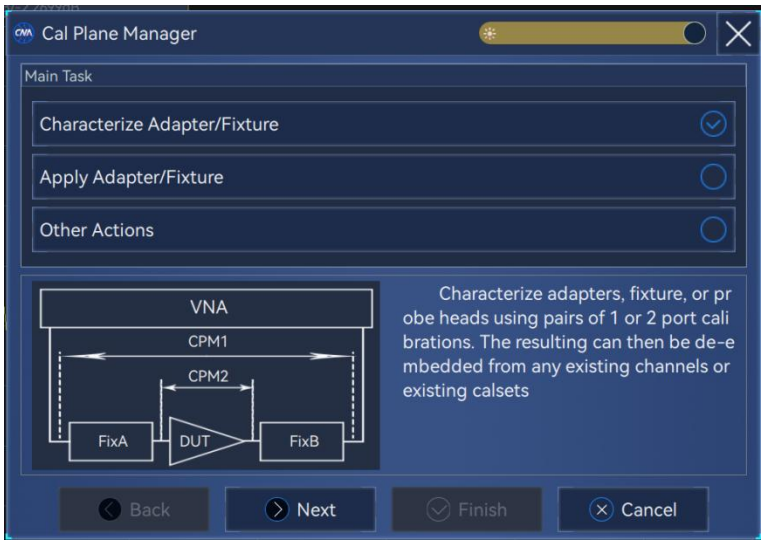


Figure 7.57 Cal Plane Manager interface

- 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

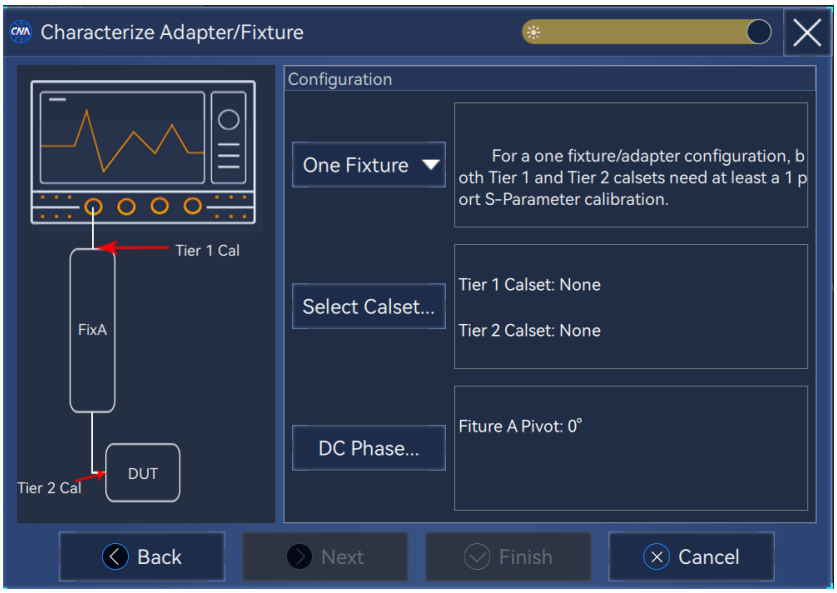


Figure 7.58 Characterize Adapter/Fixture interface

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

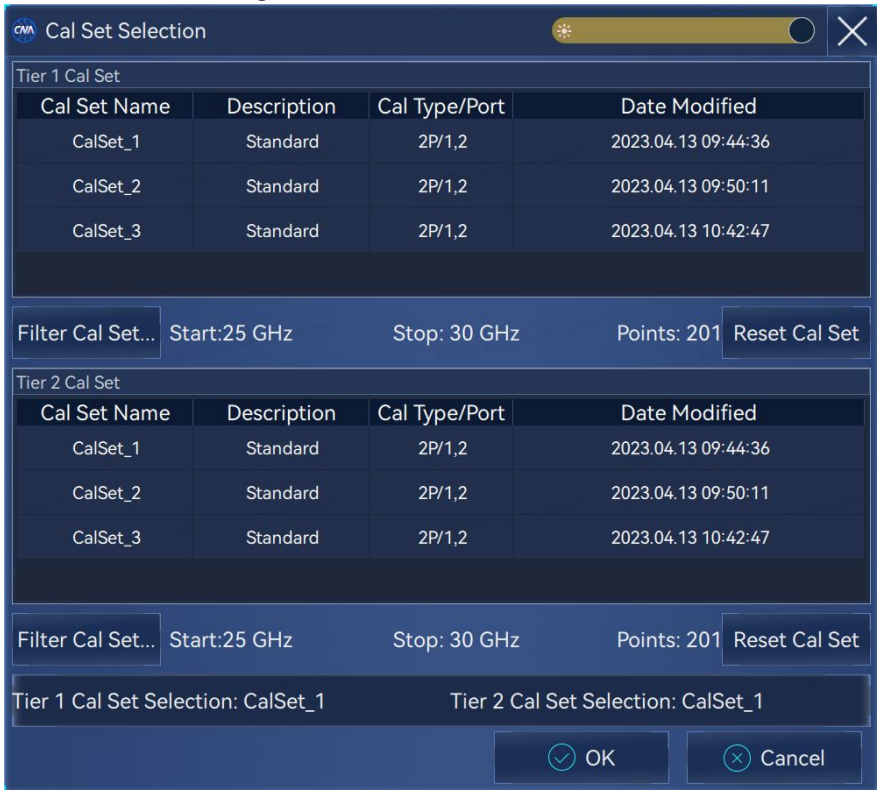


Figure 7.59 Cal Set Selection dialog box

[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 area and Tier 2 Cal Set area 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.

[Tiler 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

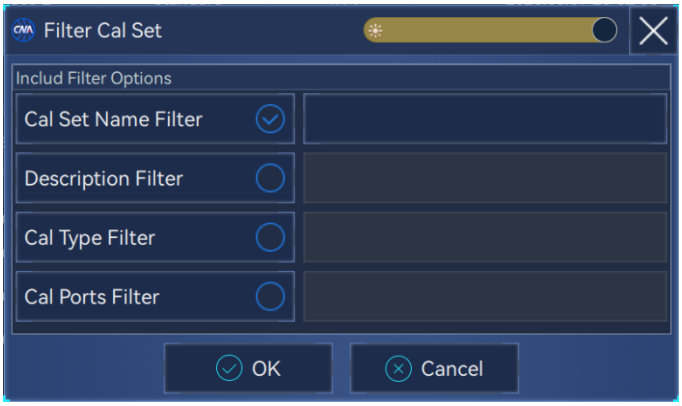


Figure 7.60 Filter Cal Sett Interface

[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

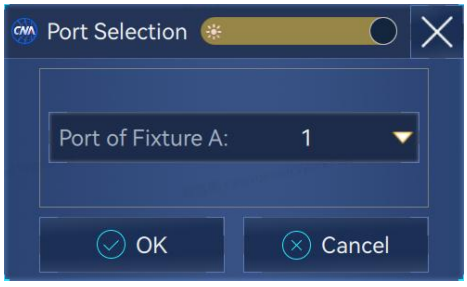


Figure 7.61 Port Selection interface

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

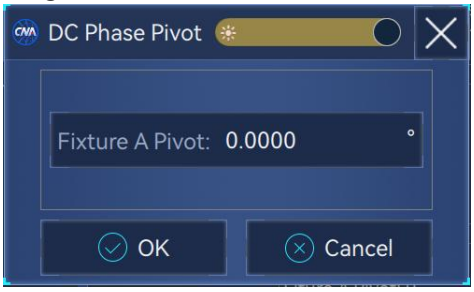


Figure 7.62 DC Phase Pivot interface

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

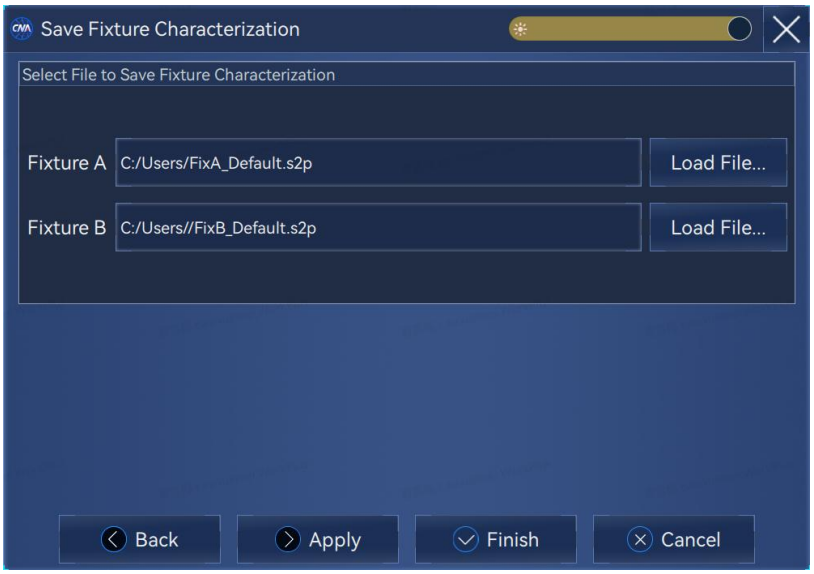


Figure 7.63 Save Fixture Characterization interface

[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.

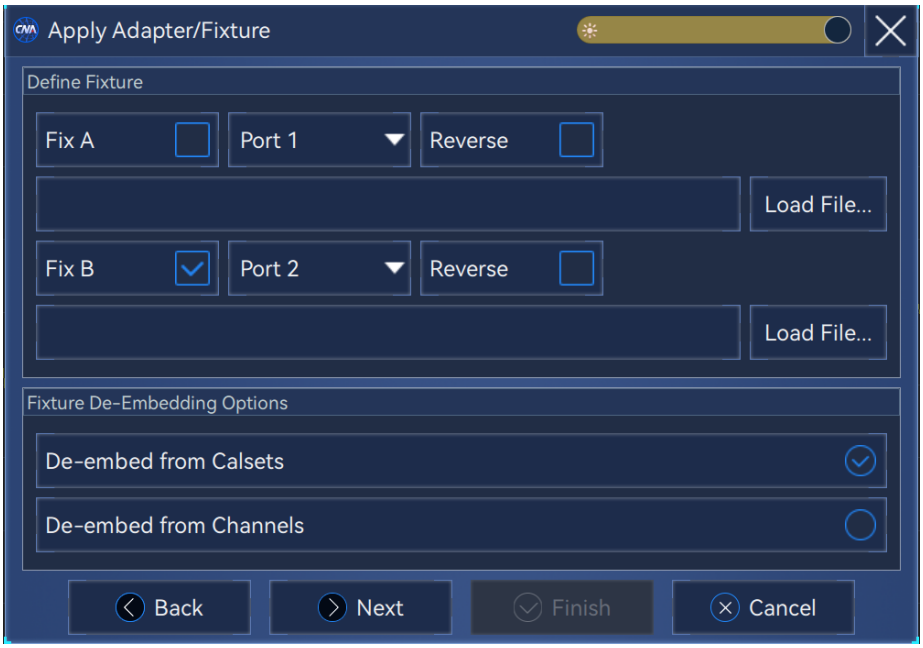


Figure 7.64 Apply Adapter/Fixture interface

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-embed from calsets/channels, the **[Next]** button is enabled, and you can click the **[Next]** button to enter the De-embed from Calsets/Channels dialog box.

[Cancel]

Click to exit the Cal Plane Manager function.

b) De-embed from Calsets dialog box

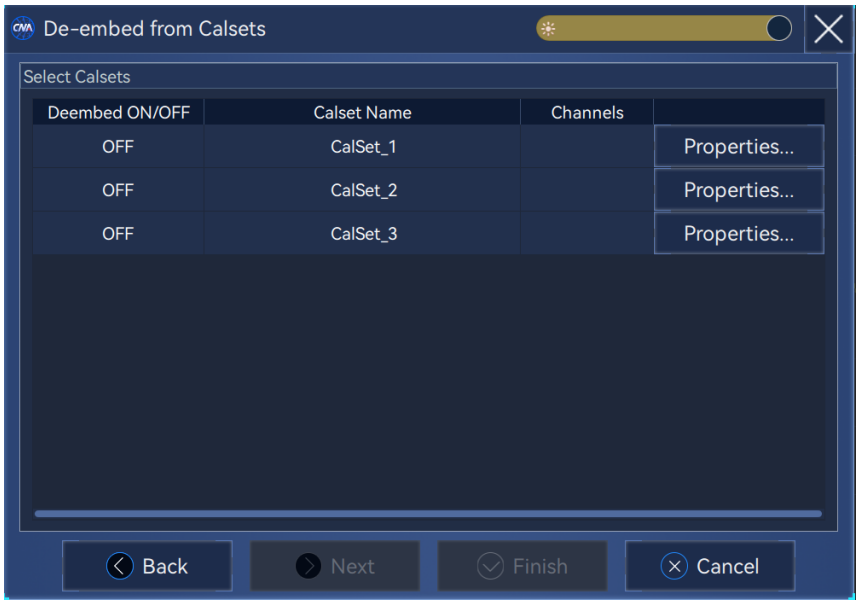


Figure 7.65 De-embed from Calsets interface

[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

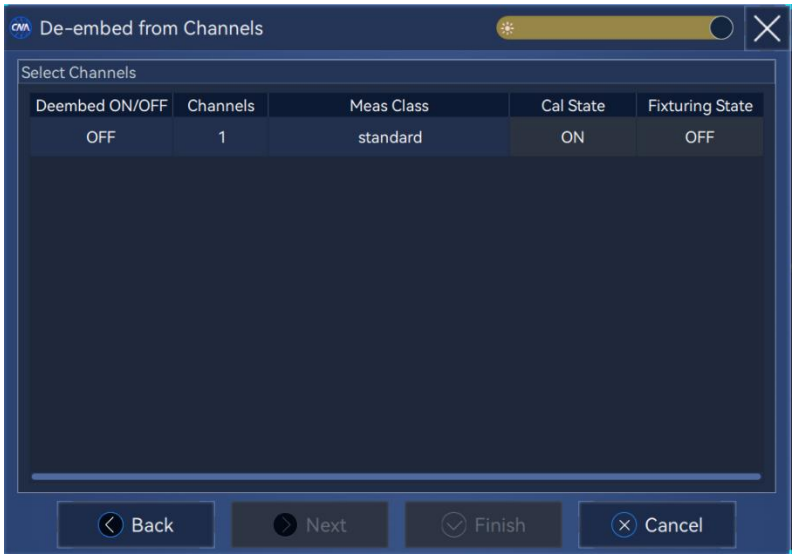


Figure 7.66 De-embed from Channels dialog box

[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



Figure 7.67 Other Actions interface

b) Reverse the Port Order of an S2P File dialog box

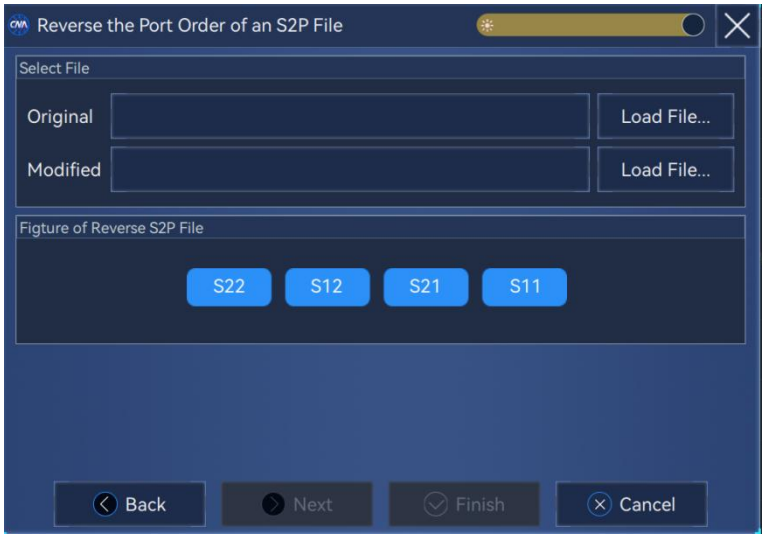


Figure 7.68 Reverse the Port Order of an S2P File dialog box

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

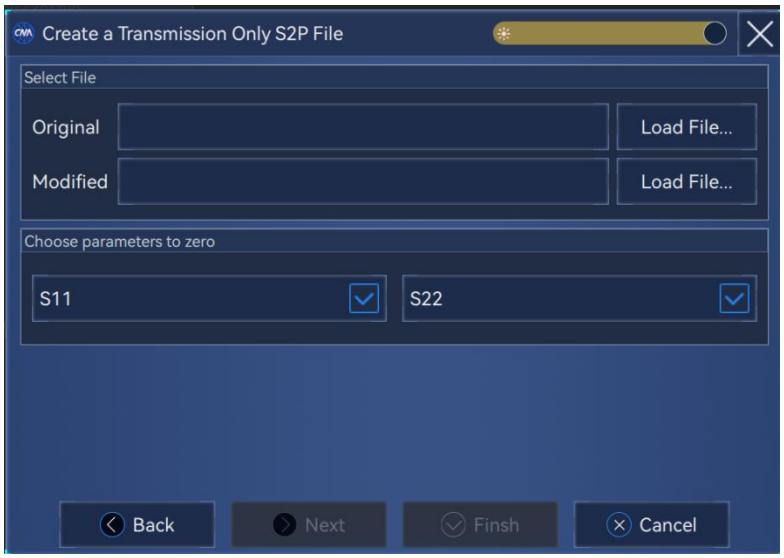


Figure 7.69 Create a Transmission Only S2P File interface

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 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 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

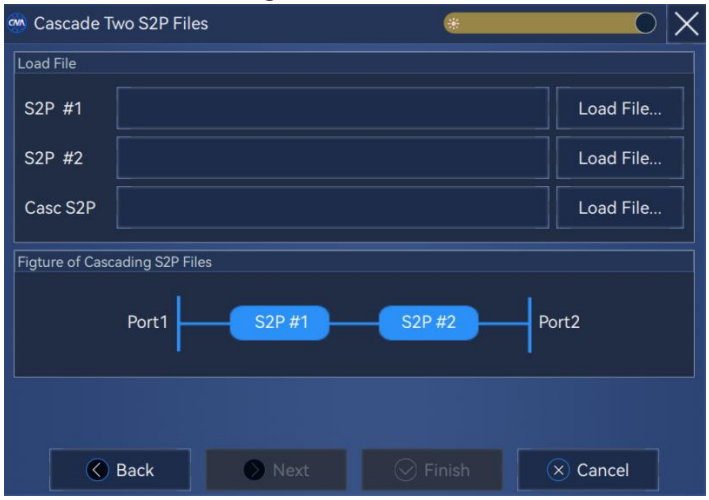


Figure 7.70 Cascade Two S2P File interface

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]

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.

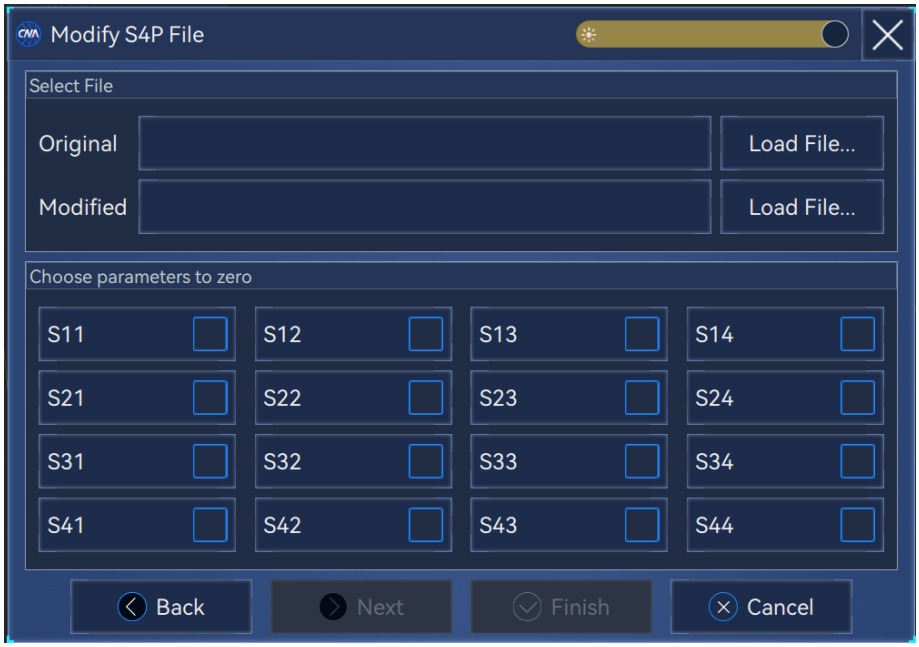


Figure 7.71 Modify S4P File interface

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.

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 .

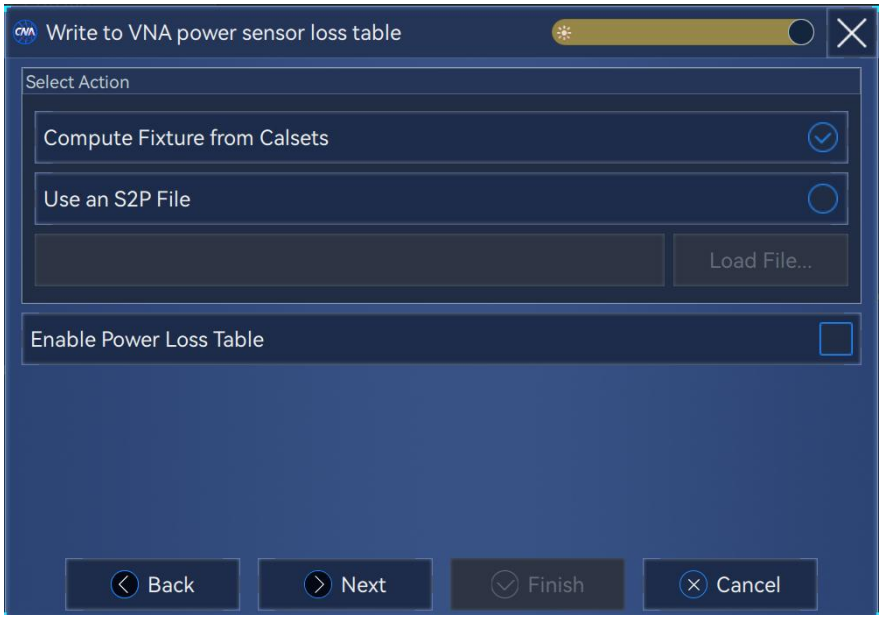


Figure 7.72 Write to VNA power sensor loss table interface

[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

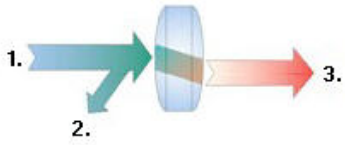
This chapter includes the following contents:

- Reflection measurement.....309
- Phase measurement.....313
- Amplifier parameter description.....315
- Complex impedance.....318
- Group delay.....320
- Absolute output power.....324
- AM-PM Transform.....326
- Linear Phase Deviation…..329
- Reserve isolation.....331
- Small signal gain and flatness.....333

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

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……310
- Summary of reflection measurement expression ……312

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, ρ=0. If the transmission line is terminated with an open circuit breaker or a short circuit breaker, all energy is reflected, ρ=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

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:

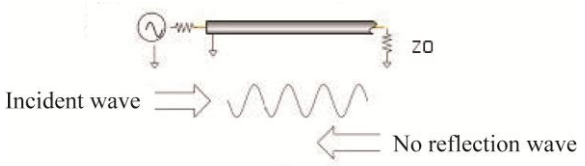


Figure 8.2 Signal transmission along a transmission line terminated with load

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 (ρ= 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.

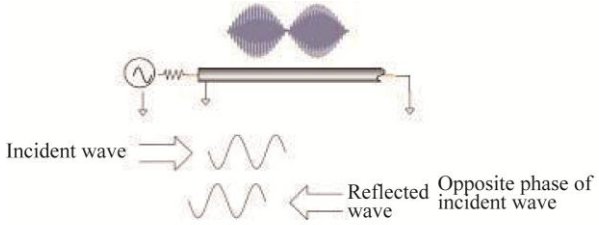


Figure 8.3 Signal transmission along a transmission line terminated with short circuit breaker

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 (ρ＝1), no current flows through the open circuit breaker, and the reflected voltage wave is in phase with the incident voltage wave.

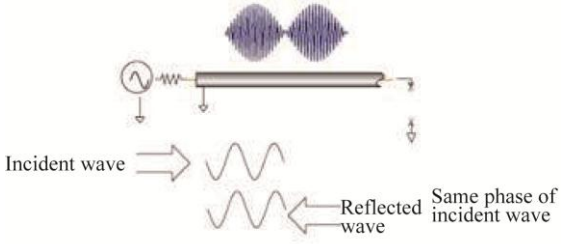


Figure 8.4 Signal transmission along a transmission line terminated with open circuit breaker

When the transmission line is terminated with a 25 Ω resistor, a portion of the energy is

absorbed and a portion is reflected back to the signal source. The amplitude of the reflected 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.

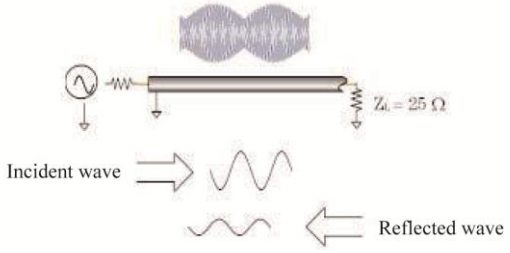


Figure 8.5 Signal transmission along a transmission line terminated with 25Ω resistor

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:

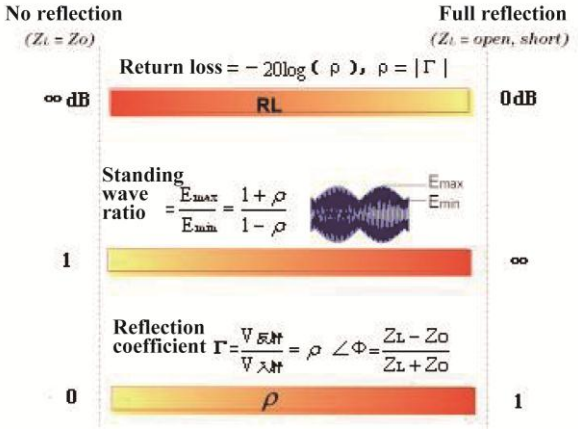


Figure 8.6 Relationship between reflection expressions

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 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.

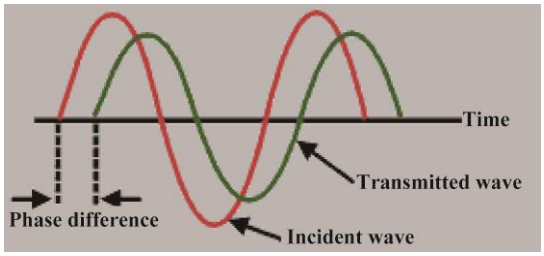


Figure 8.7 Phase offset between signals

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.

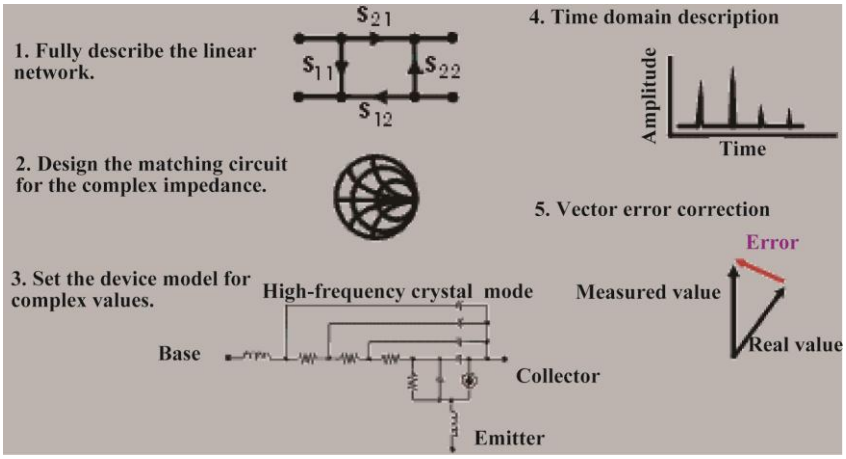


Figure 8.8 Reasons to make accurate measurements of amplitude and phase

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 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 Using 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 ± 180° between the reference signal and the response signal. When the phase value varies between +180 ° and -180 °, the analyzer displays the sawtooth phase measurement trace. The sawtooth waveform does not always reach +180 ° and -180 ° because the measurements are carried out at discrete frequency points, and the data points at +180° and -180° may not be in the sweep measurement points.

Phase format

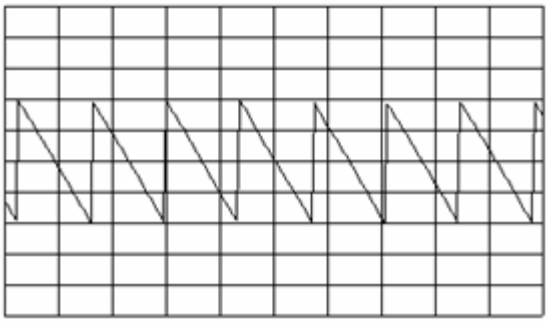


Figure 8.9 Phase format of analyzer

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 "°/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 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 delay measurement:
   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:

$$\tau = \frac{V_{trans}}{V_{inc}}$$

$$Gain\ (dB) = -20\log_{10}|\tau|$$

$$Gain\ (dB) = P_{out}\ (dBm) - P_{in}\ (dBm)$$

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:

$$\tau_g\,(\mathrm{sec}) = -\frac{\Delta\theta}{\Delta\varpi}$$

$$= -\frac{1}{360} * \frac{\Delta\theta}{\Delta f}$$

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 Z0 at the input or output port, and it is calculated by the following formula:

$$\Gamma = \frac{V_{refl}}{V_{inc}} = \rho\angle\theta$$

Reflection coefficient = ρ

Return loss (dB) = $-20\log_{10}\rho$

$$SWR = \frac{1+\rho}{1-\rho}$$

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:

$$Z = \frac{1+\Gamma}{1-\Gamma} * Z_0$$

$$= - R + jX$$

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.

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.

$$Z_{\Lambda} = [(1 + \Gamma) / (1 - \Gamma)] * Z_0$$

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 marker is used in Smith chart or polar format, higher measurement accuracy can be obtained in discrete marker 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.

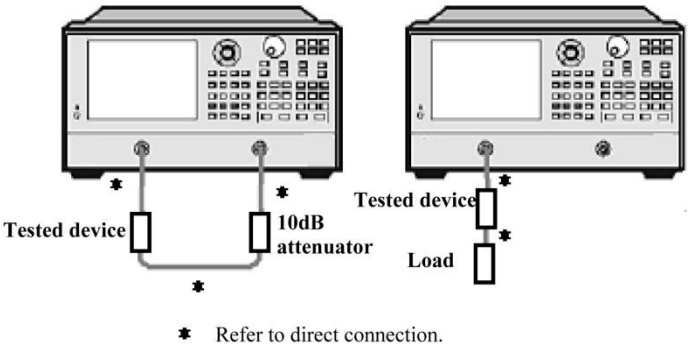


Figure 8.10 Impedance measurement connection

1) If the DUT is connected between the two ports of the analyzer after the single-port calibration, it is recommended to connect a 10 dB 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 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 marker and move the marker 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 marker and select the marker format as G+jB to read admittance data.
   b) Scale the displayed measurements for optimal viewing.
   c) Move the marker 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 marker and select the marker format as logarithmic/phase or linearity/phase.
c) Scale the displayed measurements for optimal viewing.
d) Move the marker 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.

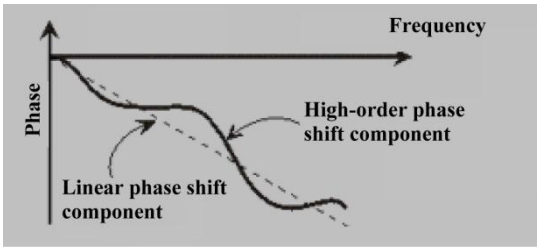


Figure 8.11 Phase offset and group delay

$$\text{Group delay} = \frac{-d\emptyset}{d\omega} = \frac{-1}{360°} \cdot \frac{d\theta}{df}$$

$\emptyset$ is in radians, ω is in radians/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.

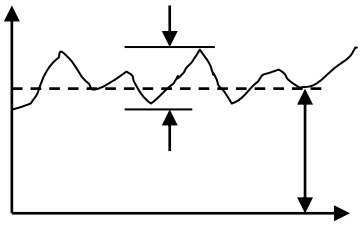


Figure 8.12 Group delay

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:

$$\text{Group delay} = \frac{-d\emptyset}{\mathrm{d}\omega} = \frac{-1}{360°} \cdot \frac{\mathrm{d}\theta}{\mathrm{df}}$$

$\emptyset$ is in radians, ω is in radians/second, θ is in degrees, and f is in hertz.

1) Calculate the phase change (-d$\emptyset$) from the phase data.
2) The specified frequency aperture is a function of frequency (dω).
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:

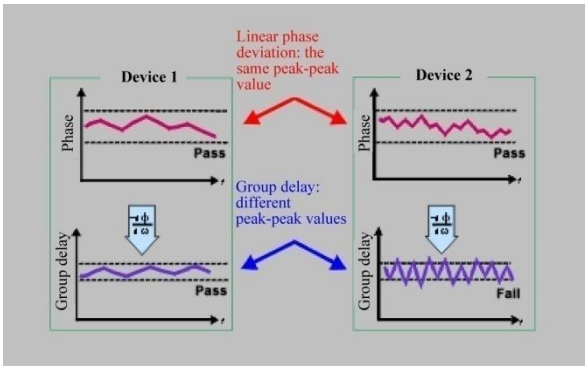


Figure 8.13 Comparison of group delay and linear phase deviation

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.

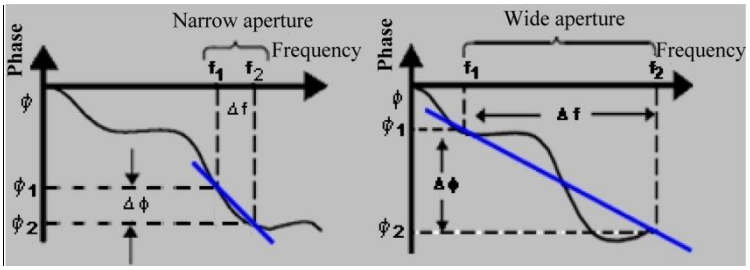


Figure 8.14 Effect of group delay aperture on measurement results

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) Using the smoothing function of the analyzer
   Perform a single sweep, turn on the smoothing function, and change the percentage of 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

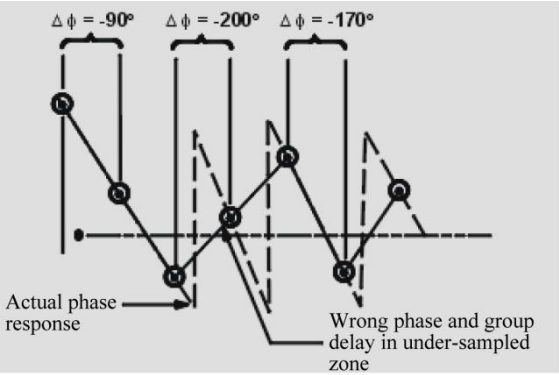


Figure 8.15 Effect of under-sampling on group delay measurement

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

(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 device under test, as shown in Figure 8.16;

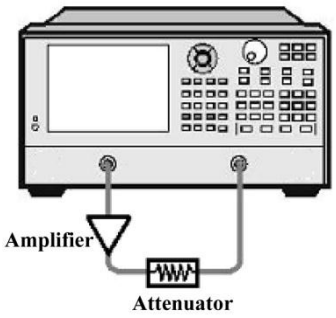


Figure 8.16 Measurement connection of group delay

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 device under test, perform calibration.
6) Connect the device under test again.
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 marker 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.

8.6.1 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:

0dBm = 1mW

-10dBm = 0.1mW

+10dBm = 10mW

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 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.

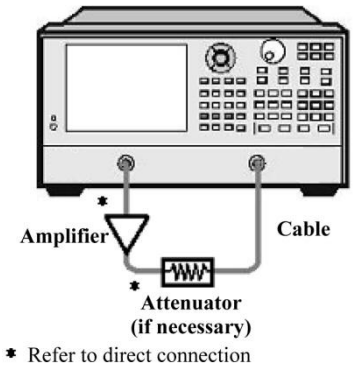


Figure 8.17 Absolute output power measurement connection

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 marker 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 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 "°/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
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.

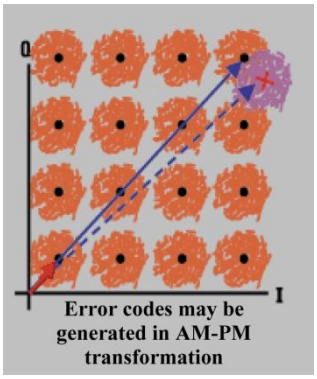


Figure 8.18 AM-PM conversion causes bit error

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 sweeping, 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.
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 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.

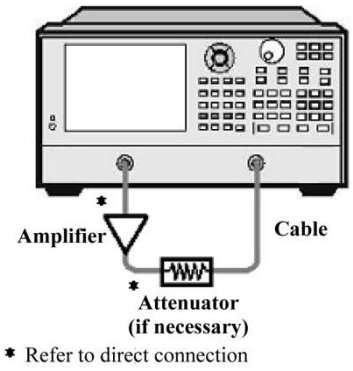


Figure 8.19 AM-PM transformation measurement connection

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 marker R, place it at the 1 dB gain compression point of the amplifier, then turn on the second marker, activate Δ marker mode, and set the stimulus value of the second marker to -1 dBm.
9) Change the S21 measurement from logarithmic amplitude format to phase format (no recalibration required).
10) Read the phase difference between the markers, 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 Using 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.

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.

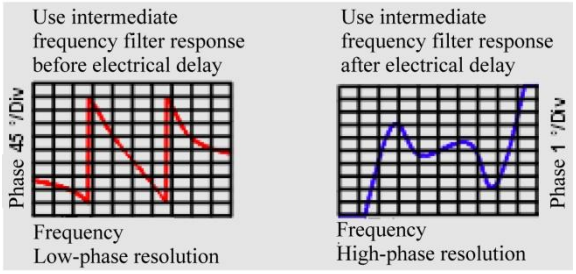


Figure 8.20 Observing linear phase deviation with electrical delay function

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 device under test, as shown in Figure 8.21.

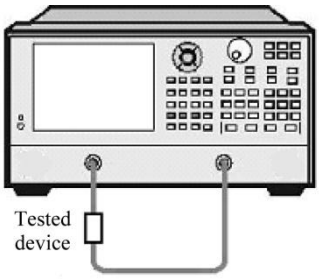


Figure 8.21 Linear phase deviation measurement connection

4) Set the analyzer according to the DUT, including setting the format as phase.
5) Remove device under test, 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 marker.
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 S12.

8.9.2 Why measure 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 analyzer.

2. Select S12 measurement.

3) Connect the amplifier and provide DC bias as shown in Figure 8.22.

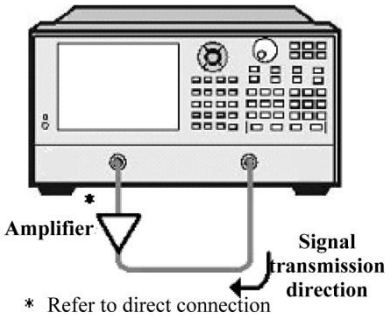


Figure 8.22 Reserve isolation measurement connection

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.
6) Reconnect the amplifier and scale the measurement display for optimal viewing.
7) Use the marker 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 Why measure 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:
   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 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.

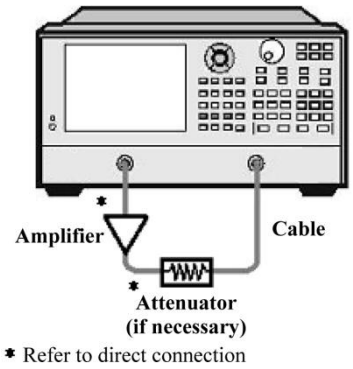


Figure 8.23 Small signal gain and flatness measurement connection

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 marker to read the small signal gain at the desired frequency point.
10) Use markers 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 program control foundation, remote control interface and configuration method and basic VISA interface programming method of the 3674 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.....337
- Instrument Program Control Port and Configuration.....356
- Basic VISA interface programming method.....363
- I/O Library.....371

9.1 Remote Control Basics

9.1.1 Remote Control Interface

This analyzer supports two kinds of remote control interfaces: LAN and GPIB, 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 network analyzer with the network port on the rear panel of the network analyzer. For details of the protocol, please refer to: 9.2.1 LAN Interface

GPIB (IEC/IEEE Bus Interface)	GPIB::primary address[::INSTR]	The controller realizes remote control by connecting the network analyzer with the GPIB interface on the rear panel of the network analyzer. The bus interface standard IEC 625.1/IEEE 418 is met. For details, please refer to: 9.2.2 GPIB Interface
USB (Universal Serial Bus)	USB::<vendor ID>::<product_ID>::<serial_number> [::INSTR]	Instrument rear panel port. For details, please refer to: 9.2.3 USB Interface

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.

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:

TC PIP::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);
  - No.0 equipment selects VXI-11 protocol;
  - More recent high speed LAN instrument protocol is selected for No.0 high speed LAN instrument;
- 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 remote 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.

9.1.1.2 GPIB Interface

As an instrument remote control interface widely used at present, GPIB interface is connected to different types of instruments through GPIB cable, so as to build a test system with the host computer. In order to realize remote control, the host computer should be equipped with GPIB bus card, driver and VISA library in advance. During communication, the host computer first addresses the controlled instrument through the GPIB bus address. The user may set the GPIB address and ID query string, and the GPIB communication language may be in the form of SCPI by default.

GPIB and its associated interface operations are defined and described in detail in ANSI/IEEE Standard 488.1-1987 and ANSI/IEEE Standard IEEE Standard 488.2-1992. For details of the standards, please refer to the IEEE website:http://www.ieee.org/

GPIB processes information in bytes, and the data transmission rate can reach 8MBps, which is relatively fast. Since the data transmission rate is limited by the distance between the device/system and the computer, the following points should be noted when connecting GPIB:

- Up to 15 instruments may be built through GPIB interface.

- The total length of the transmission cable should not be more than 15m or twice the number of instruments in the system. In general, the maximum length of the transmission cable between the devices cannot exceed 2 meters.
- If multiple instruments are connected in parallel, a GPIB cable is required.
- The end of the IEC bus cable shall be connected to the instrument or master computer.

9.1.1.3 USB Interface

To implement USB programming, a computer and vector network analyzer need to be connected via a USB B-type port with the VISA library installed in advance. VISA automatically detects and configures the instrument to establish a USB connection without the need to enter the instrument address string or install a separate driver.

USB address:

Addressing string format: USB::<vendor ID>::<product ID>::<serial number>[::INSTR]

Where:

<vendor ID> represents the manufacturer code;

<product ID> represents the network analyzer code;

<serial number> represents the serial number of the network analyzer;

Example:

USB0::0x3399::0x3000::2022020::INSTR

0x3399: Manufacturer code;

0x3000: Instrument code;

2022020: serial number of the network analyzer.

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. Only instruments with GPIB bus function can send interface message.

2) Instrument message

The network analyzer message can be divided into two types as per the transmission direction, namely, command and instrument response. Unless otherwise stated, all remote control interfaces apply instrument message in the same way.

a) Commands:

Commands (programming messages) are messages sent by the host computer to the network analyzer 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 network analyzer:

-- 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) Remote control commands are oriented to test function rather than describing instrument operation;
2) Remote 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) Remote control commands are independent of programming methods and languages. The test program of SCPI is easy to transplant.
5) Remote control commands are scalable and can adapt to different scale of measurement control.

6) SCPI have been a “living” standard for their scalability.

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 3674 series vector network analyzer, please refer to:

1) “3 Remote control commands” of the Remote Control Manual;
2) “Annex A Zoom table of SCPI commands classified as per subsystems” of the Remote 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.

Device

A device is any device that supports SCPI. Most of the devices are electronic measurement or stimulus devices.

Program message

A program message is the combination of one or more SCPI commands that have been correctly formatted. Program 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 remote 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, the power subsystem (:POWer) contains power-related commands, while the status subsystem (:STATus) contains commands for the status control register.

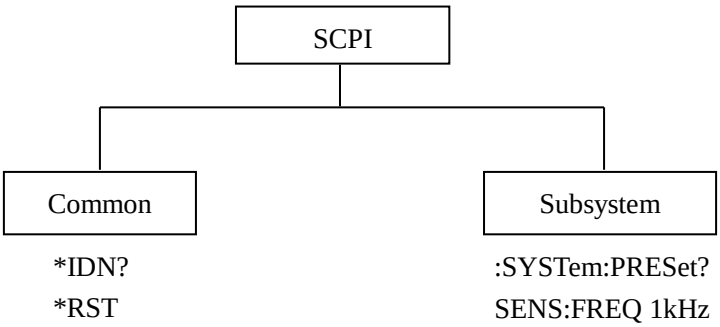


Figure 9.1 Types of SCPI

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 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<cnum>: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, which does not affect the current path.	:FREQ uency or :POWer :LEVel6.2 is not allowed. :LEVel and 6.2 must be separated by a space, namely, :POWer:LEVel 6.2.

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:

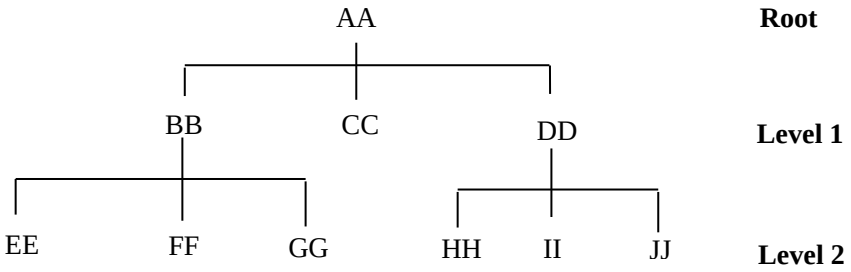


Figure 9.2 Simplified command tree

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 (reseting) the instrument, current command path is reset to root.

5) Command parameters and responses

SCPI define different data formats in the use of program 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.

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 -5.00000000000e+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

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.

Note: 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

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 state 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

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:

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 initial 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 program 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 setting is completed. During programming, the computer may force a period of time to synchronize certain events. The descriptions are shown below:

- **Master program using multi-threading**

  Multi-threading is used to wait for command completion and synchronization between the UI and program control, that is, to wait for *OPC? Completion in separate threading

without interfering GUI or program 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 until 1 is returned. 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 State 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 System 312

9.1.5.1 Structure of State Register

State registers are described by classification below:

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: STATus:OPERation and STATus: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

It stores the messages returned by the network analyzer to the master. 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 program control manual.

Please refer to Figure 9.3 for the hierarchical structure of the state register:

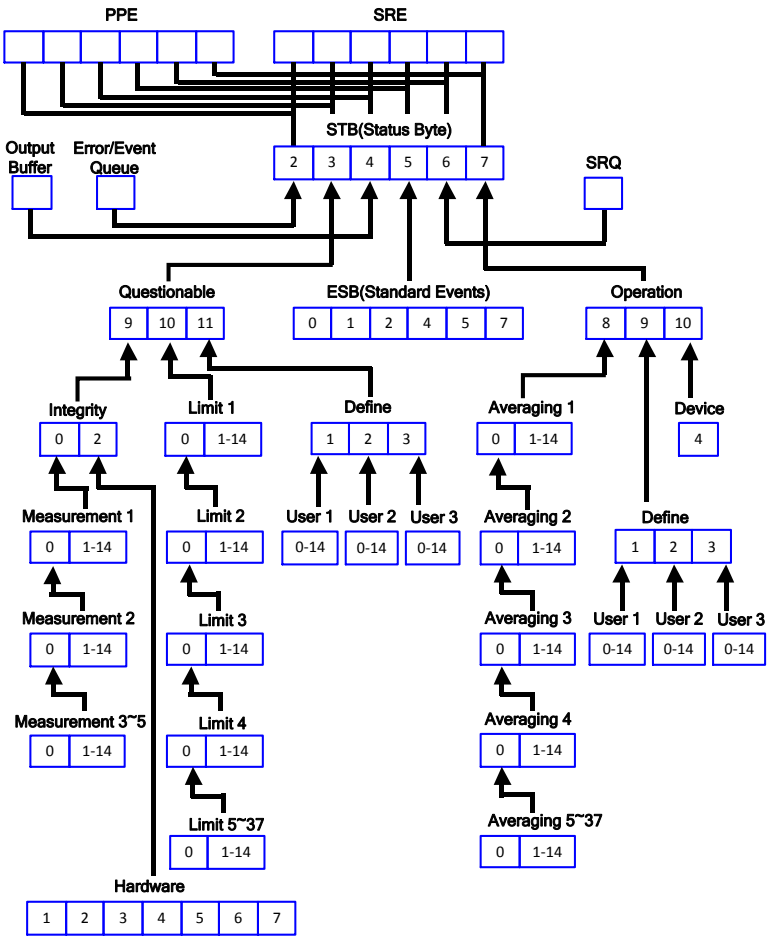


Figure 9.3 Hierarchical structure of the state register

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 State Reporting System

The state 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;

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) Remote 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 network analyzer 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 state 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 Program Port and Configuration

- LAN.....356
- GPIB.....359
- USB.....361

9.2.1 LAN

SICL-LAN is used by LAN program control system to control 3674 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 3674 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 program control of the instrument.

9.2.1.1 Connection Establishment

Connect the 3674 series vector network analyzer and external controller (computer) to the LAN with network cable, as shown in Figure 9.4:

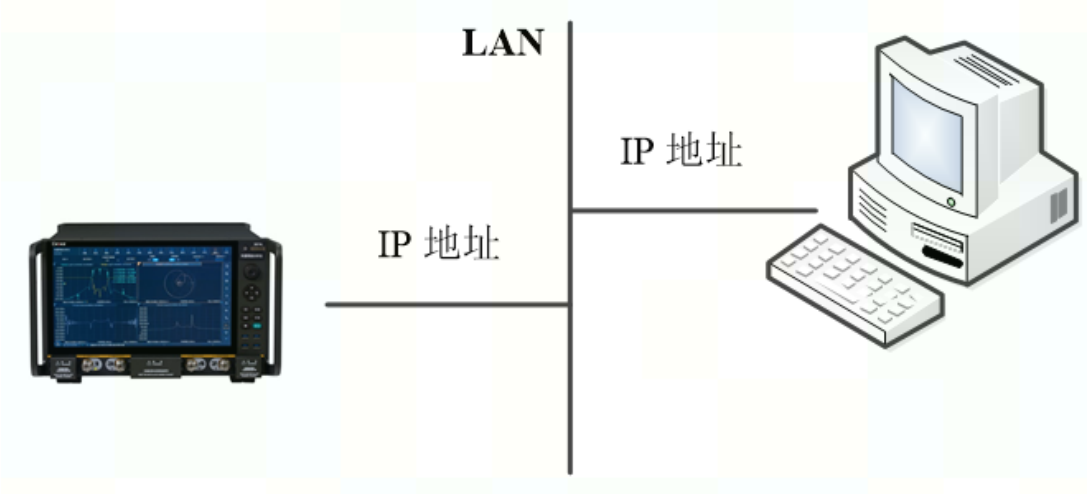


Figure 9.4 LAN interface connection

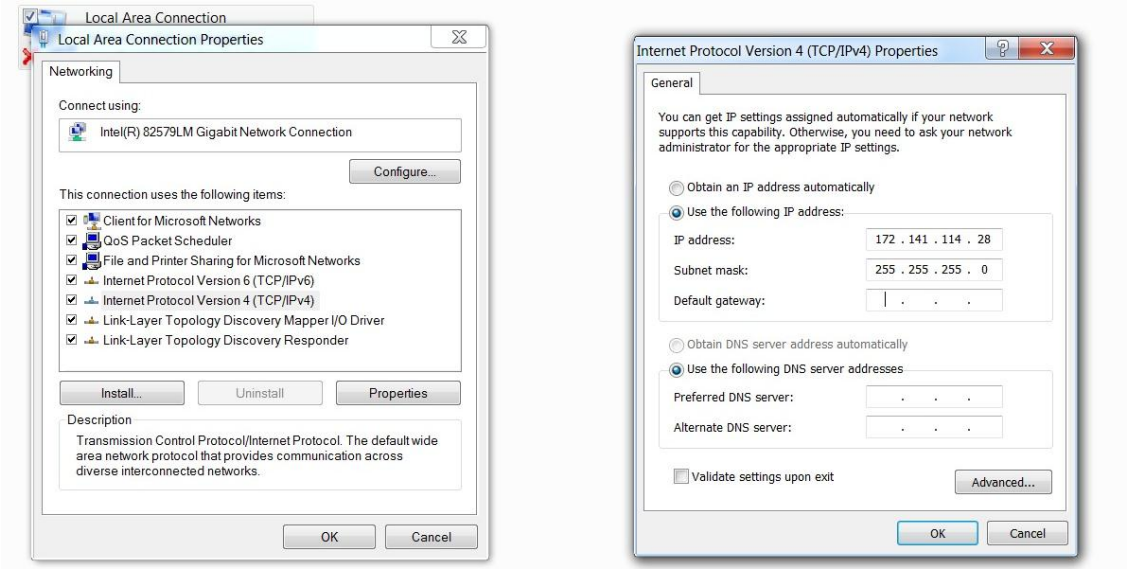


Figure 9.5 Instrument rear panel LAN interface

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.



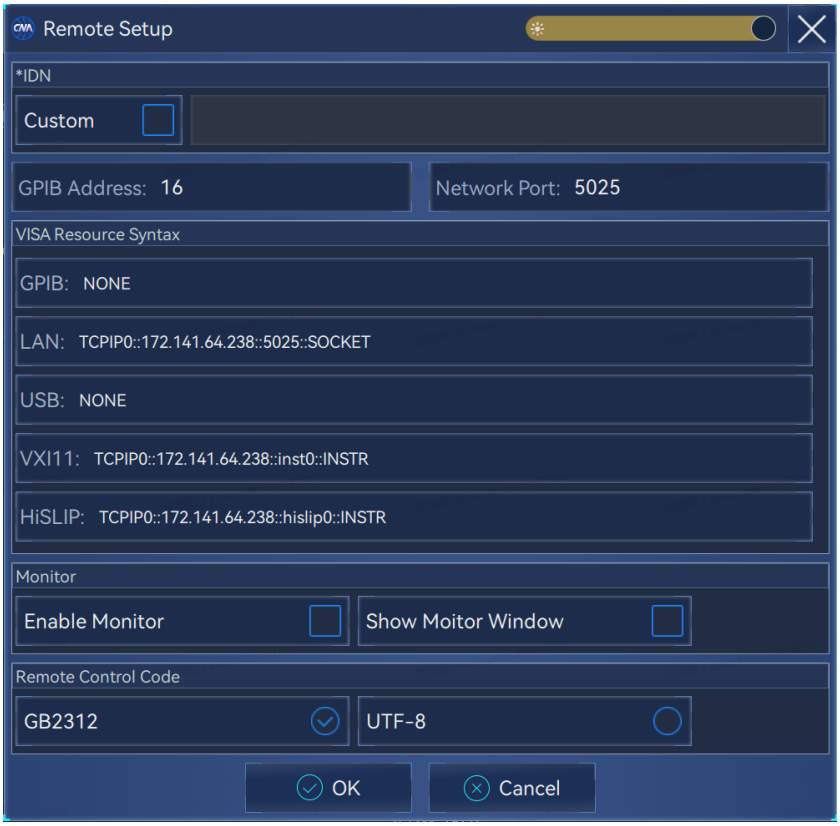


Figure 9.6 LAN interface setting

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.

9.2.2 GPIB

9.2.2.1 Connection Establishment

Use GPIB cable to connect 3674 series vector network analyzers with external controller (computer), as shown in Figure 9.7:

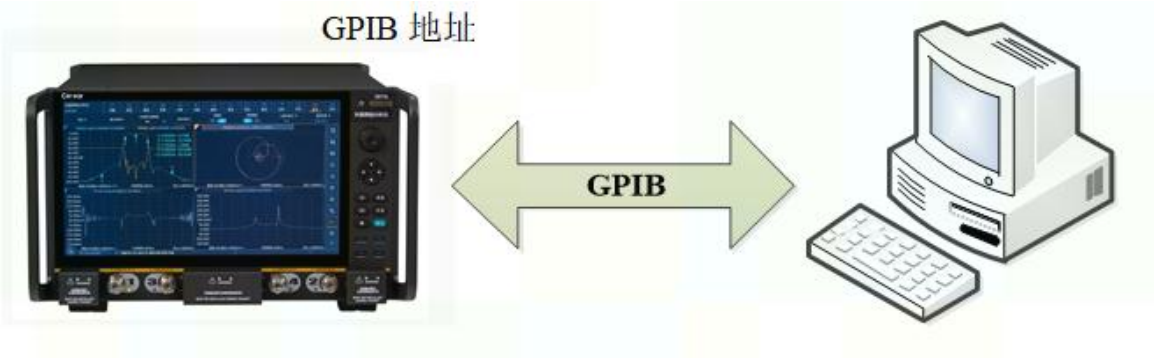


Figure 9.7 GPIB interface connection



Figure 9.8 Instrument rear panel GPIB interface

9.2.2.2 Interface Configuration

Users may need to modify the GPIB address when using the vector network analyzer to establish the system. The default GPIB address of the machine is 16. Methods for modification of the GPIB address are described below:

Click [System] -> [Remote Control…] to go to the interface shown in Figure 9.9, and then use keyboard, mouse or the number key on the front panel to modify in the GPIB address input box of the machine.

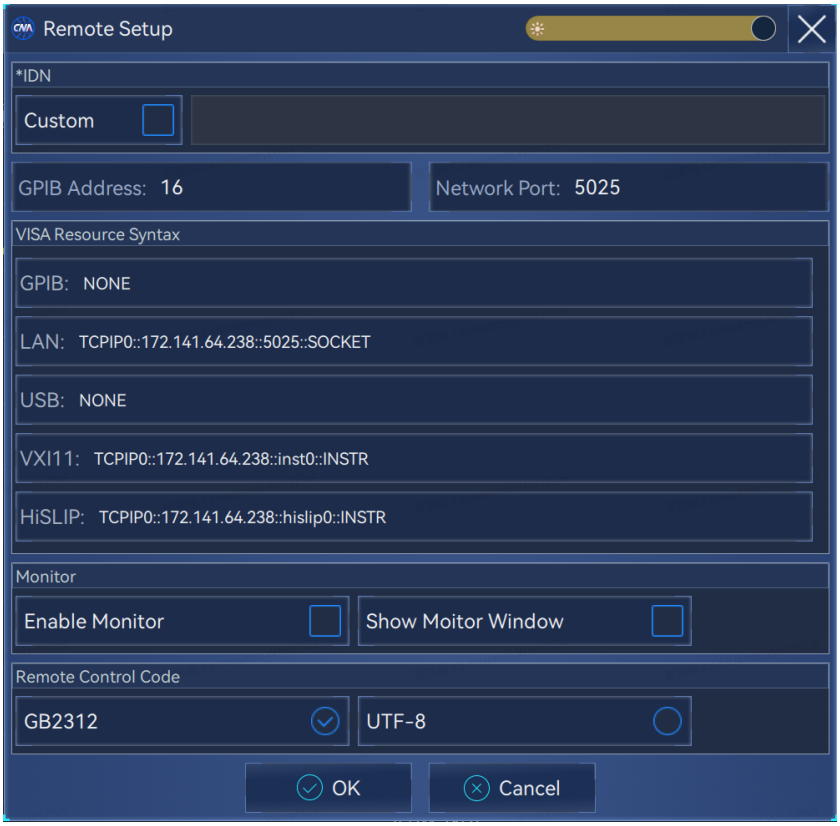


Figure 9.9 GPIB interface setting

9.2.3 USB

2.2.3.1 Connection Establishment

Use USB cable to connect 3674/3672/3671/3650 series vector network analyzers with external controller (computer), as shown in Figure 9.10:



Figure 9.10 USB connection



Figure 9.11 Instrument rear panel USB interface for remote control

The sockets of USB connectors support type-A configuration (4 contacts embedded, with contact 1 on the left) and type-B configuration (connectors of type-A configuration supports the connection with USB mouse, keyboard, or other devices with USB interface. The connectors of type-B configuration are mainly for realization of control, and through those connectors, the network analyzer can be controlled via SCPI to achieve interconnection with external computers or remote devices. One interface of this kind is provided on the rear panel, as shown in Figure 9.11.

2.2.3.2 Interface Configuration

When building a system with vector network analyzer, users need to obtain the VID and PID of USB address as well as the serial number of instrument. By default, the VID is 0x3399, the PID is 0x3000, and the serial number is the same as the factory-set serial number of the instrument. Users can view the USB port related information on the host computer by going to [USB Test and Measurement Device (IVI) Properties] -> [Details] -> [Device Instance Path] via the computer device manager, as shown in Figure 9.12. The VID, PID and serial number information are generally not changed under USB programmable conditions.



Figure 9.12 USB interface information

9.3 Basic VISA interface programming

Take the following as an example to describe the basic method for realizing instrument program control programming via the VISA library. Take the C++ language as an example.

- VISA Library……363
- Initialize and Set Default State……364
- Send Setting Command……366
- Read the State of Measuring Instrument……368
- Read Frequency Maker……369
- Read Trace Data…369
- Command Synchronization……370

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 program control of the instrument. By initializing the addressing string ("VISA resource string"), a connection to an instrument with a program port (LAN, USB, GPIB etc.) 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, GPIB, LAN and USB interfaces, so as to facilitate users' direct loading. The vector network analyzers support the following programming interfaces: GPIB, LAN and USB. **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 GPIB interface as an example to show the relationship between remote control interface, VISA libraries, programming languages and vector network analyzers.

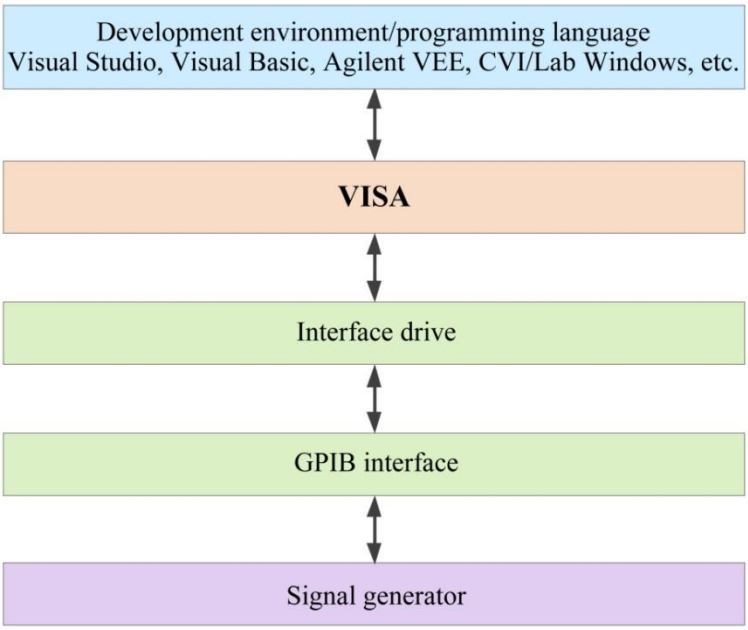


Figure 9.13 Programmable software/hardware layer

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
}
```

9.3.2.4 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_BUFLEN]; // #define VI_READ_BUFLEN 20
    char rd_Buf_LVL[VI_READ_BUFLEN];

    // 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 sweeping parameters of 3674 series vector network analyzer.

/**/

```
void SweepSettings()
{
    ViState state;
    long retCnt;
    //Select measurement
    status = viWrite(analyzer, ":CALC:PAR:SEL ‘CH1_WIN1_LINE1’ \n", 50,&retCnt);

    //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);

    //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 State 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_BUFLEN];

    //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
    sprint("Cw is %s", rd_Buf_CW);
    sprint("LEVel is %s", rd_Buf_ LVL);
}
```

9.3.5 Read Frequency Marker

```
/*****************************************************************************/
The following example shows how to read the frequency marker.
/*****************************************************************************/
void ReadMarker ()
{
    ViState state;
    long retCnt;
    char rd_Buf_Marker[VI_READ_BUFLEN]; // #define VI_READ_BUFLEN 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

```
/*****************************************************************************/
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 network analyzer hardware, it consists of the function library, utility program, tool kit, etc. It is a combination of a series of software code modules and corresponds to operation of a plan, such as configuring the network analyzer, reading from the network analyzer, writing to the network analyzer and trigger the network analyzer, etc. Residing in the computer, it is the bridge and link between the computer and the network analyzer. 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.

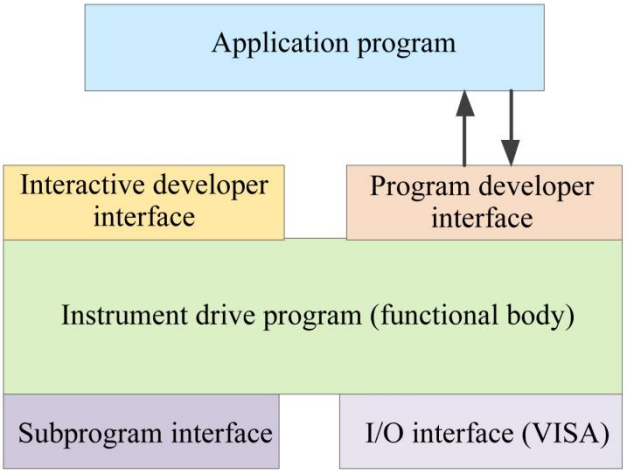


Figure 9.14 Structure model of instrument driver

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 (such as GPIB) 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, program 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/LabWindows, 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 program 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 3674 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 ____375
- Troubleshooting and debugging……377
- Errors……379
- Method to Obtain After-sales Services……400

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.

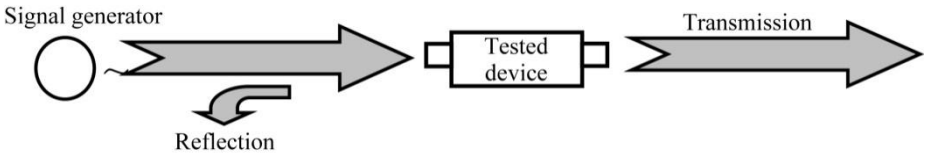


Figure 10.1 Response of DUT to the signal

10.1.2 Analyzer principles

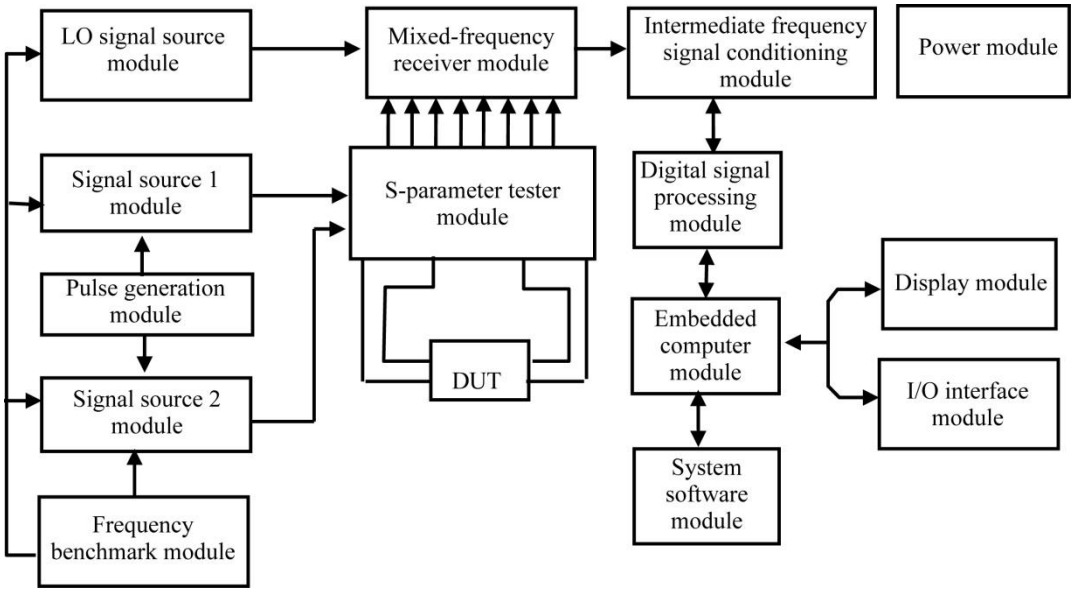


Figure 10.2 Block diagram of the analyzer principle

The 3674 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.

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 Troubleshooting

Tips

Troubleshooting and instructions

This section introduces the way on how to judge and handle failures (if any) of the 3674 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 area of the analyzer, please click "Help→ Error→Check Error Log" to understand the specific Error description.

- System problems.....377
- Abnormal curve display.....378
- Sweeping failure.....378
- Display problem.....379
- No response from front panel keys.....379
- Menu Operation Failure.....379

10.2.1 System problems

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.

10.2.1.2 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 marker appears.

10.2.3.2 Sweep stopped

Check whether the trigger state of the instrument is “Hold”. If yes, the trigger mode needs to 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 marker 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 marker will appear at the bottom of the screen when sweeping port 2. It is the normal instrument sweeping, no need to change the setting.

10.2.4 Display problem

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 GPIB state shown in the status bar at the bottom of the window is LOCAL. If the program is already in the program 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.

10.3.1 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:

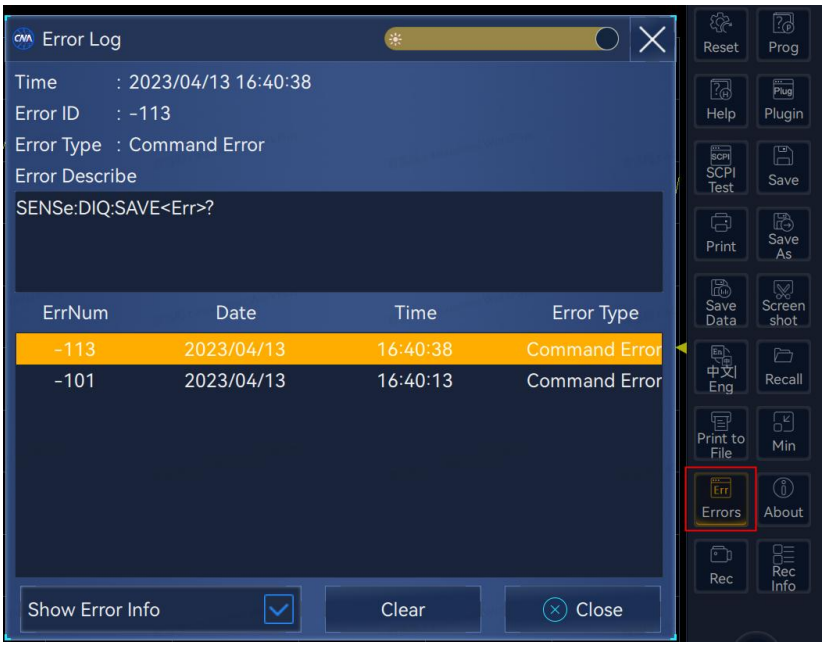


Figure 10.3 Error message display mode dialog box

Table 10.1 Font color distinction error level

Color	Error	Error Description
Red	Error	Errors occurred during measurement, such as data loss or incorrect settings that prevents the measurement from being completed
Blue	Warning	Irregular working conditions during measurement, such as mismatch between instrument settings and display results, or temporary
Black	Prompt	Step setting or measurement information.

10.3.2 Program 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 program 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.
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 arrray 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, 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 gpib32.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.
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.
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.** 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 unleveled.** The source cannot be properly leveled to the required power. It indicates that the power is not accurate.
856	Error	**LO unleveled.**
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 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.** 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 marker.** An attempt was made to create a △ marker function (for COM only) before creating a reference marker
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.
1111	Error	**Marker 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 area.**
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>.**
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. See 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 marker does not exist.**
1626	Prompt	**Specified window already exists.**
1792	Prompt	**Sweep completion**
1793	Prompt	**All trigger requests completed.**
1794	Prompt	**Marker 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 sweeping mode to segment sweeping mode, but there is no defined segment or there is no sweeping 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 remote control.
1873	Error	**Requested sweep segment not found.** A sweep segment that does not exist is specified under remote 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.** 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.**
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!**
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 3674 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 network analyzer 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 consign it as precision instruments.
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……403
- Appendix 2 Time domain measurement……409
- Annex 3 Frequency Offset Measurement……441
- Annex 4 Pulse Measurement……441
- Annex 5 Amplifier Gain Compression Measurement……468
- Annex 6 Mixer Measurement……485
- Annex 7 Intermodulation Distortion Measurement……485
- Annex 8 Noise Figure Measurement……485
- Annex 9 Advanced Time Domain Analysis……485
- Annex 10 Fast Continuous Point Measurement Mode……485

Appendix 1 Introduction to typical measurements

- After 30 minutes of preheating, reset the analyzer……404
- Set frequency and power……404
- Select a Measurement and Create a Trace……404
- Calibration……406
- Connect the device under test ……407
- Adjust scale and data analysis……407
- Record or Save data……408

The 3674 series vector network analyzer features various options and complicated measurement, so certain skills and practical experience are required for a poper 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 3674 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



The reset operation here is for system resetting, so as to ensure that the analyzer settings are known.

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

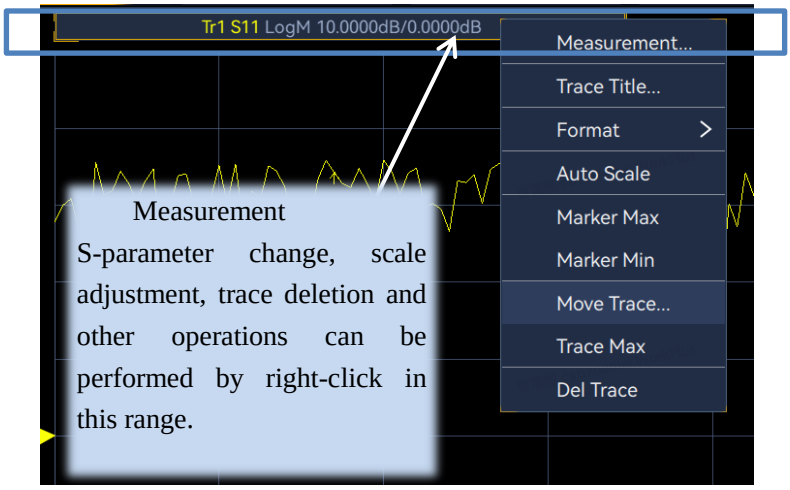
Set Frequency and Power

1) The default frequency range of system is 10MHz ~ 13.5GHz/26.5GHz/43.5GHz/50GHz/67GHz, 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 lever is changed.

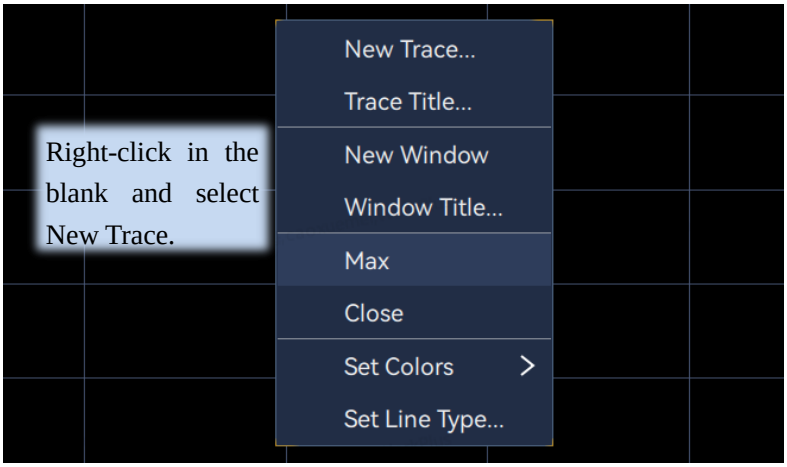
1.3 Select a Measurement and Create a Trace

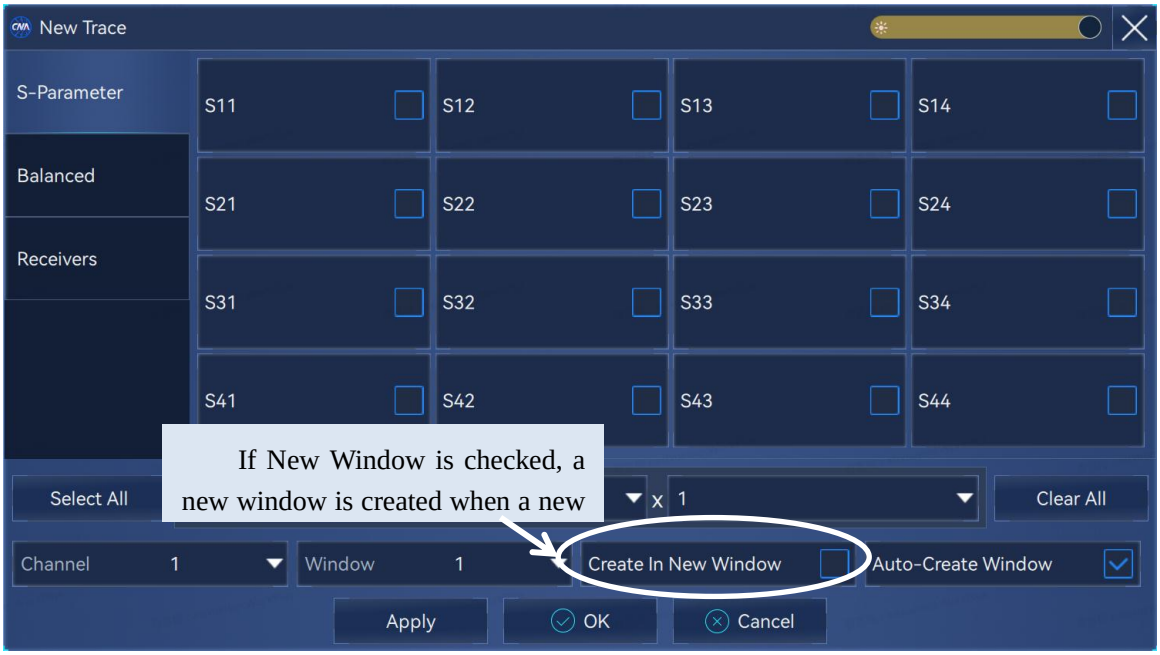
For more information, see "4.2 Select Measurement Parameters".

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.



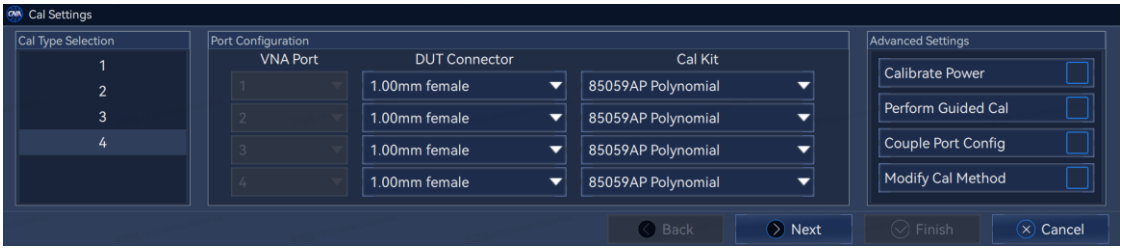


1.4 Calibration

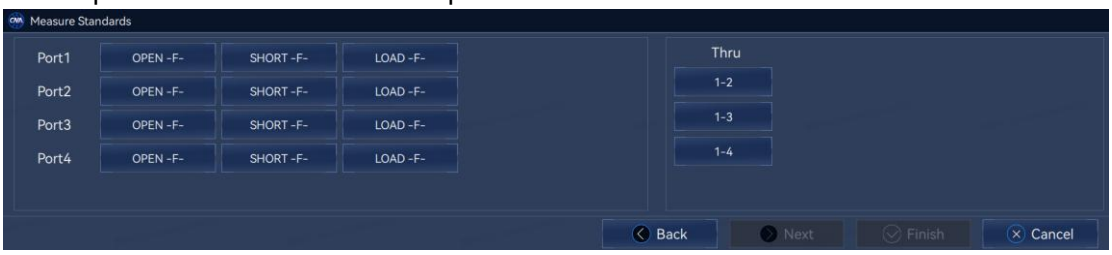
For more information, see "7 Calibration".

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.

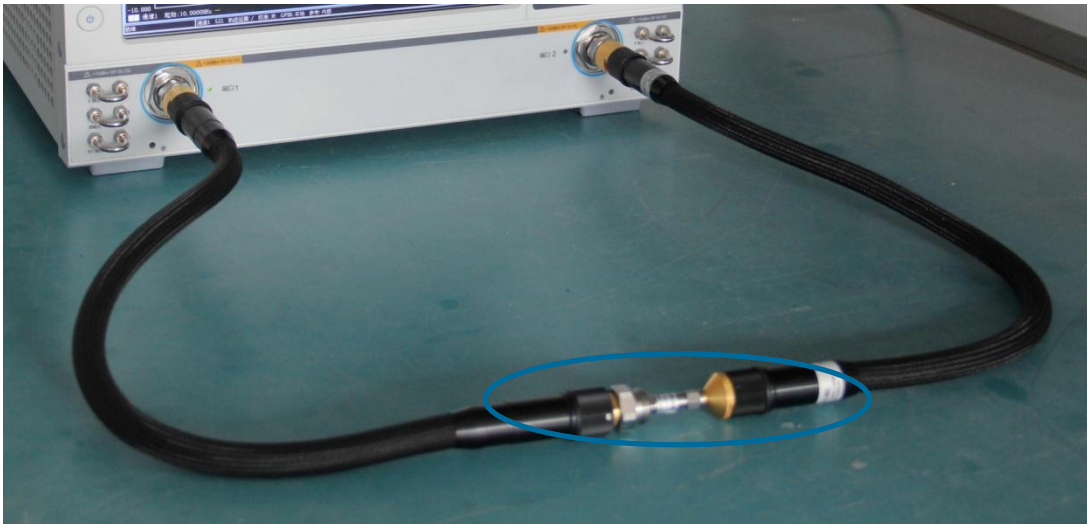


Connect the calibration kit, and then click Measure. The system will automatically proceed to the next step after this calibration is completed.



Click [Finish] after all calibrations have been completed.

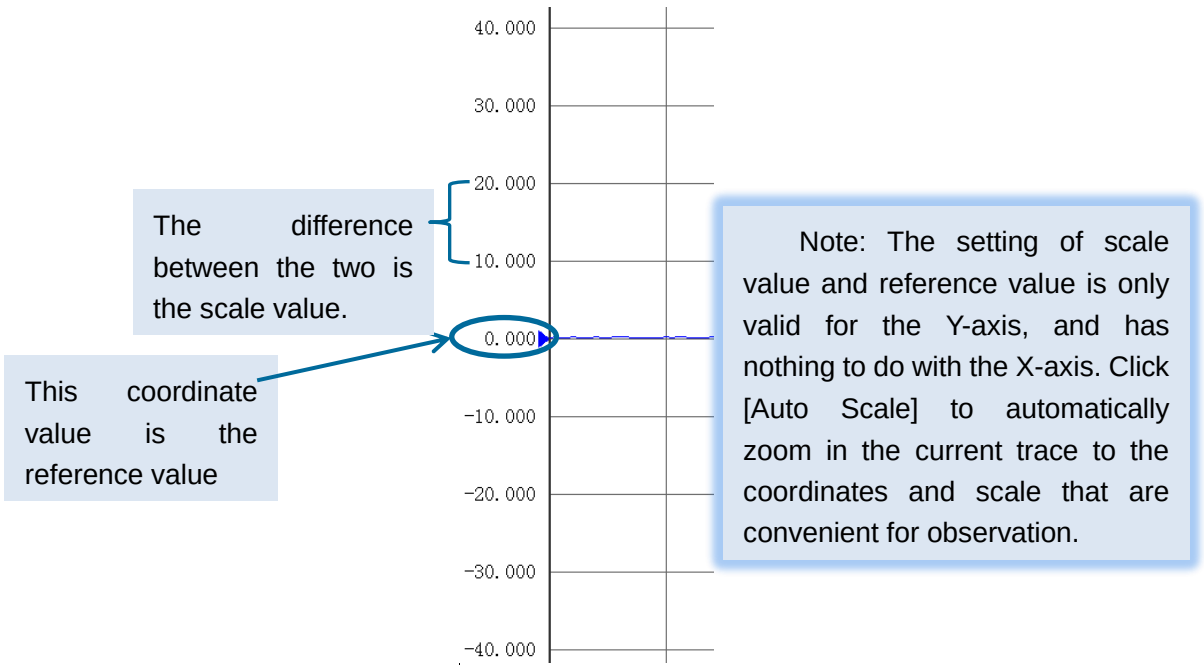
1.5 Connect the DUT



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.

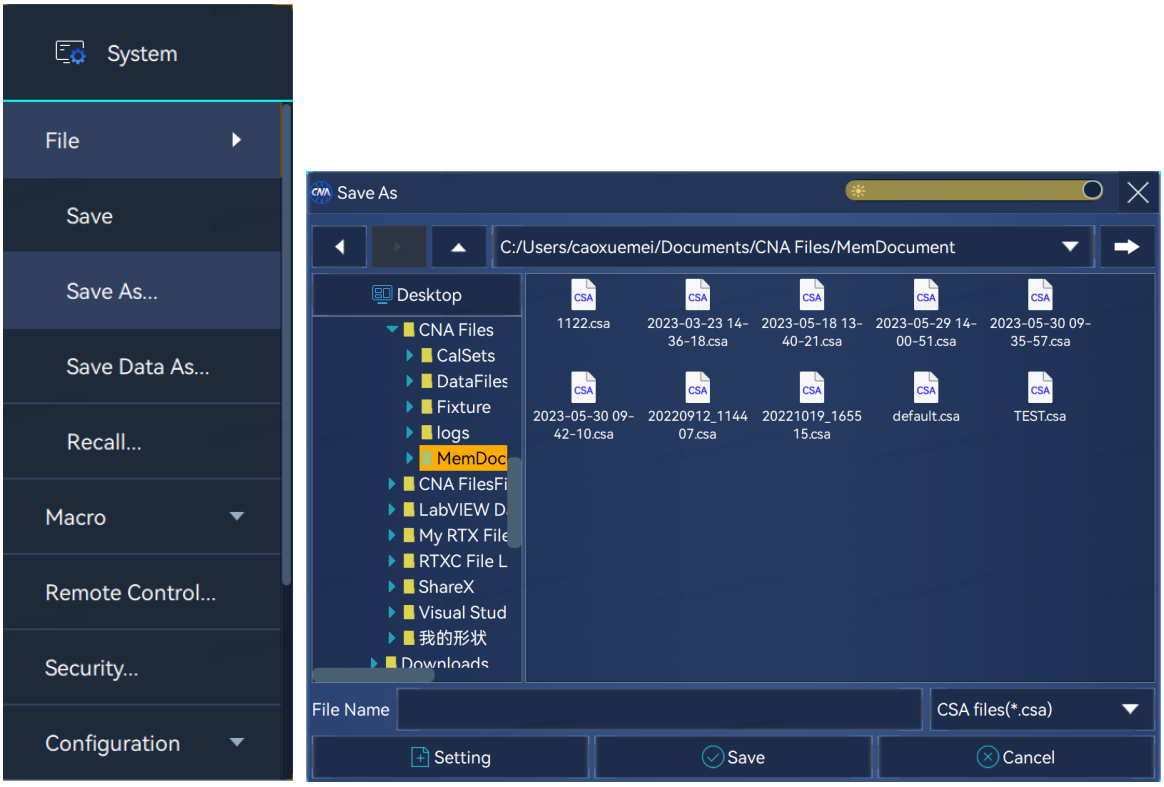


Click [Mrk] in the menubar to turn on the marker, and turn the adjustment knob to adjust the frequency corresponding to the marker. The marker reading is displayed in the upper right corner of the screen.

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.....409
- Time Domain Measurement Resolution and Range.....411
- Window Filtering.....417
- Time Gate Filtering.....420
- Time Domain Measurement Data.....425
- Bandpass and Lowpass Time Domain Modes.....431
- Setting of Time Domain Conversion Measurement.....434
- Time Domain Reflectometer (TDR) Impedance Test 434

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.

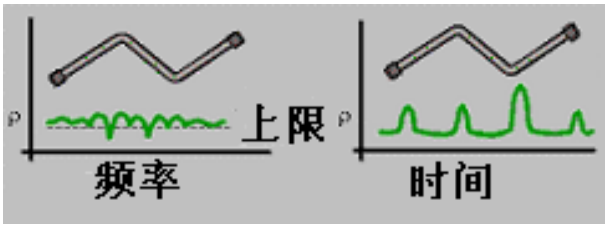


Figure 2.1 Measurement of frequency and time domains

频率	Frequency
上限	Upper limit
时间	Time

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.

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 S11 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, S11 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) Correct calibration error
4) Transform the frequency data into the time domain
5) Displaying measurement results

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.....411
- Display Resolution.....414
- Measuring Range.....415

2.2.1 Response resolution

- Concept of Response Resolution.....411
- Factors Affecting the Time Domain Response Resolution.....411
- Tips for Optimizing Response Resolution.....414

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:

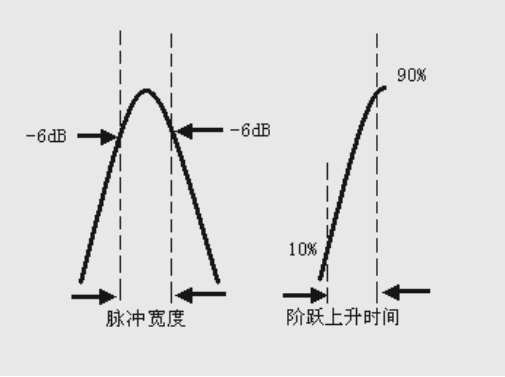


Figure 2.2 Time domain response resolution

脉冲宽度	Imp width
阶跃上升时间	Step rise time

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:

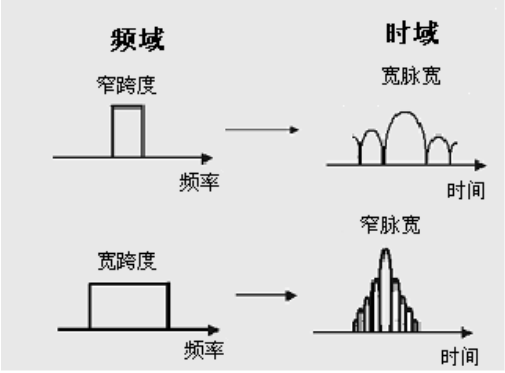


Figure 2.3 Effect of the frequency span on the response resolution

频域	Frequency domain
窄跨度	Narrow span
频率	Frequency
宽跨度	Wide span
频率	Frequency
时域	Time domain
宽脉宽	Width of broad pulse
时间	Time
窄脉宽	Width of narrow pulse
时间	Time

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

$$\text{响应分辨率} = \frac{0.6}{\text{频率跨度}} \times \begin{cases} 1.0 & \text{Minimum window} \\ 1.6 & \text{Standard window} \end{cases}$$

Bandpass impulse

$$\text{响应分辨率} = \frac{1.2}{\text{频率跨度}} \times \begin{cases} 1.0 & \text{Minimum window} \\ 1.6 & \text{Standard windows} \end{cases}$$

响应分辨率	Response resolution
频率跨度	Frequency span

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 conversion 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:

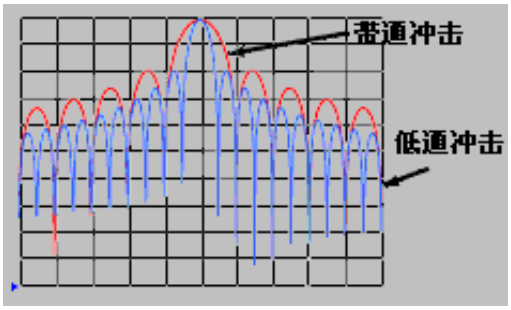


Figure 2.4 Influence of transform mode on resolution

带道冲击	Bandpass impulse
低道冲击	Lowpass impulse

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:

Display resolution = Time span/(sweep points -1)

The display resolution can be adjusted in the following two ways:

1) Reduce time span
2) Add the 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;
2) A resolution of 12.5 picoseconds can be achieved over a time span of 2.5 nanoseconds.

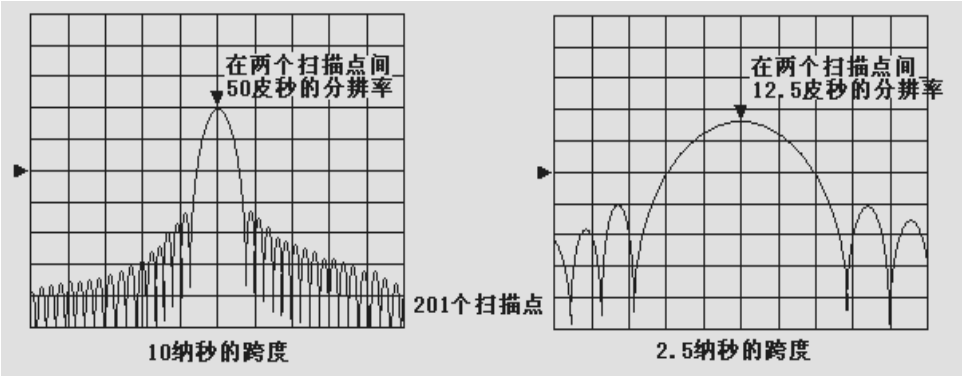


Figure 2.5 Resolution display

在两个扫描点间 50 皮秒的分辨率	50 nanosecond resolution between two sweep points
10 纳秒的跨度	10-nanosecond span
201 个扫描点	201 sweep points
在两个扫描点间 12.5 皮秒的分辨率	12.5 nanosecond resolution between two sweep points
2.5 纳秒的跨度	2.5-nanosecond span

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/sweep point frequency interval), so the measurement range is determined also by the frequency interval ΔF of the sweeping point:

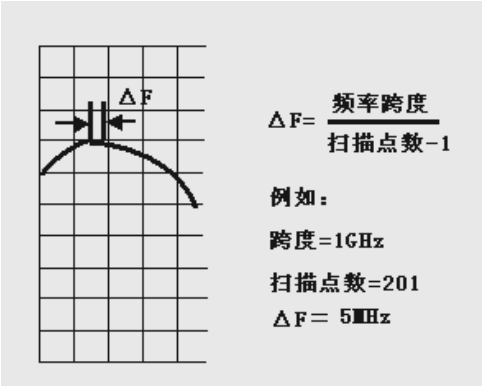


Figure 2.6 Definition of the sweeping point frequency space

频率跨度	Frequency span
扫描点数-1	Sweep point-1
例如:	For example:
跨度=1GHz	Span=1GHz
扫描点数=201	Sweep points=201

In the time domain measurements, the maximum measurable stop time is: 1/ΔF. 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) Add the 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 △F. 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

Physical length of the actual measurement range $= \frac{V}{\triangle F} \times 3 \times 10^8$ 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.....417
- Set Window Values Correctly.....418
- Window Properties.....419

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 overshot 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:

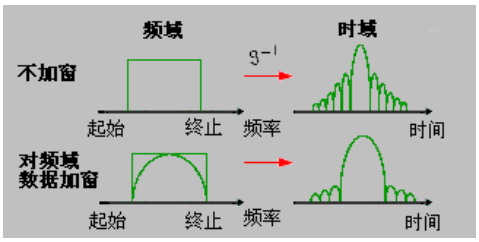


Figure 2.7 Using window filter to suppress side lobes

不加窗	No window
频域	Frequency Domain
时域	Time domain
起始	Start

终止	Stop
频率	Frequency
时间	Time
对频域数据加窗	Windowing frequency domain data
起始	Start
终止	Stop
频率	Frequency
时间	Time

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.

2.3.1.3 Effects of window filtering

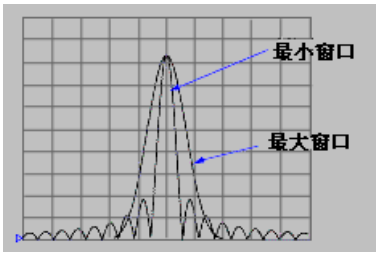


Figure 2.8 Influence of window on response signal

最小窗口	Minimum window
最大窗口	Maximum window

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.

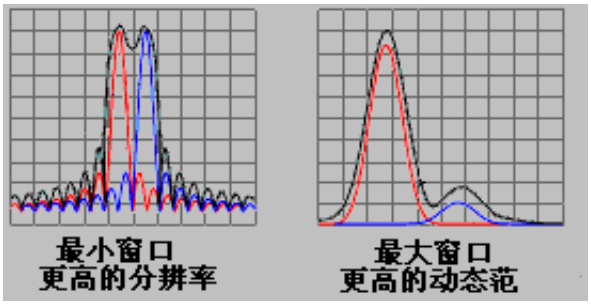


Figure 2.9 Correct Setting Window

最小窗口 更高的分辨率	Minimum window Maximum resolution
最大窗口 更高的动态范	Maximum window Higher dynamic scope

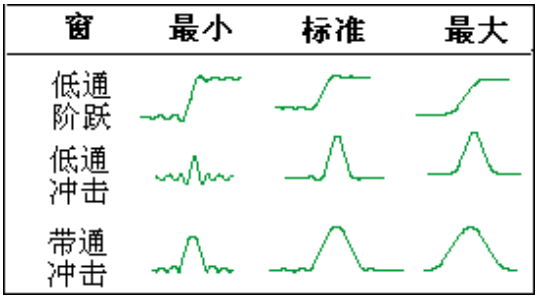
2.3.3 Window Properties

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 level
		Step rise time	Imp Width	
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



窗	Window
最小	Min
标准	Standard
最大	Max
低通阶跃	Lowpass step
低通冲击	Lowpass impulse
带通冲击	Bandpass impulse

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.....420
- Gate Setting.....421
- Gate Shape.....422

2.4.1 How 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.
2) Enable the time domain conversion function, and the analyzer calculates the time domain response of the DUT.
3) Select the gate shape:

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:

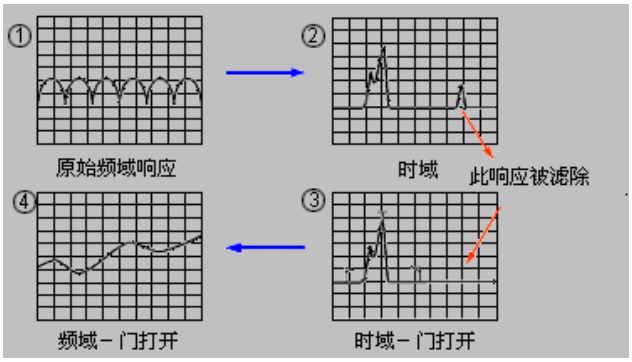


Figure 2.10 Measurement application of bandpass time gate

原始频域响应	Original frequency response
时域	Time domain
此响应被滤除	This response is filtered
频域-门打开	Frequency domain-door open
时域-门打开	Time domain-door open

2.4.2 Door setting

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) Door 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:

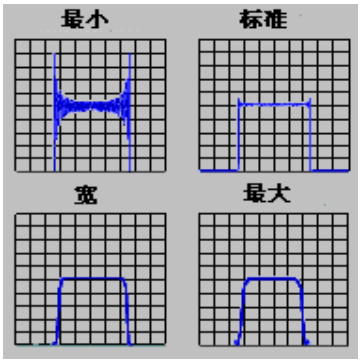


Figure 2.11 Passband ripple of different gate filters

最小	Min
标准	Standard
宽	Width
最大	Max

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.

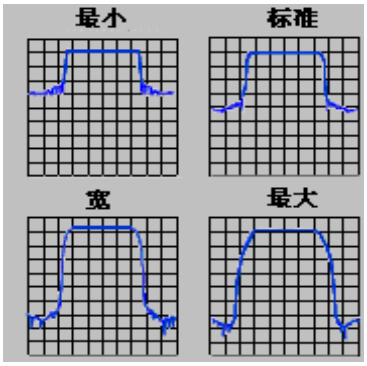


Figure 2.12 Sidelobe levels of different gates

最小	Min
标准	Standard
宽	Width
最大	Max

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: T**he 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	±0.10dB	-48 dB	1.4/frequency span	2.8/frequency span
Standard	±0.01dB	-68 dB	2.8/frequency	5.6/frequency

			span	span
Wide	±0.01dB	-57 dB	4.4/frequency span	8.8/frequency span
Max	±0.01dB	-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.

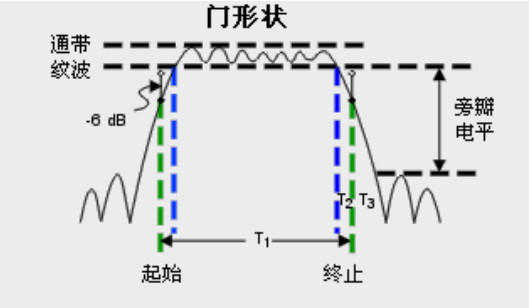


Figure 2.13 Gate shape

通带纹波	Passband ripple
门形状	Gate shape
旁瓣电平	Sidelobe level
起始	Start
终止	Stop

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 following formula, where the passband of the filter is 0.

$$T_{1MIN}=2\times T_2$$

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.

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.....425
- Reflection Measurement in Bandpass Mode.....426
- Transmission Measurement in Bandpass Mode.....427
- Error Location in Lowpass Mode.....428
- Reflection Measurement in Lowpass Mode.....429
- Transmission Measurement in Lowpass Mode.....430

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 △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.

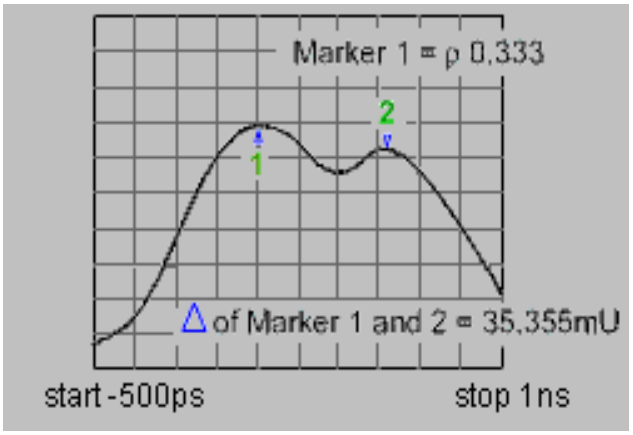


Figure 2.14 Influence of masking caused by energy reflection on measurement

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.

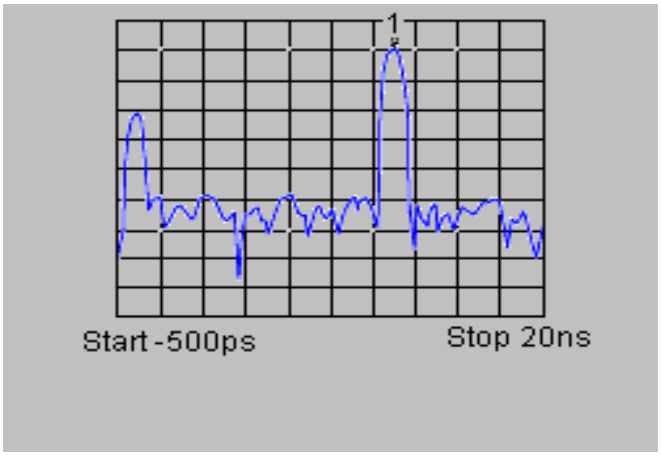


Figure 2.15 Influence of masking caused by energy absorption on measurement

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.

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.

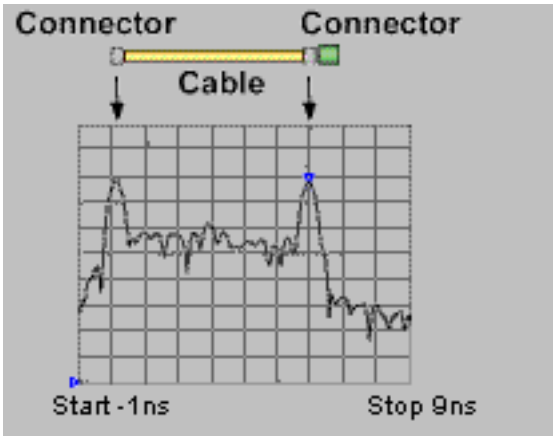


Figure 2.16 Reflection measurement in bandpass mode

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.

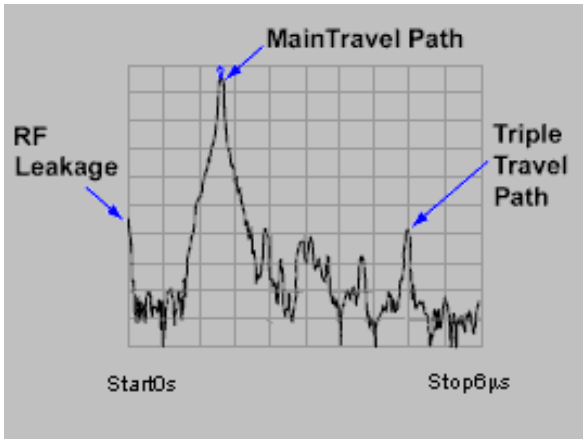


Figure 2.17 Transmission measurement in bandpass mode

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.

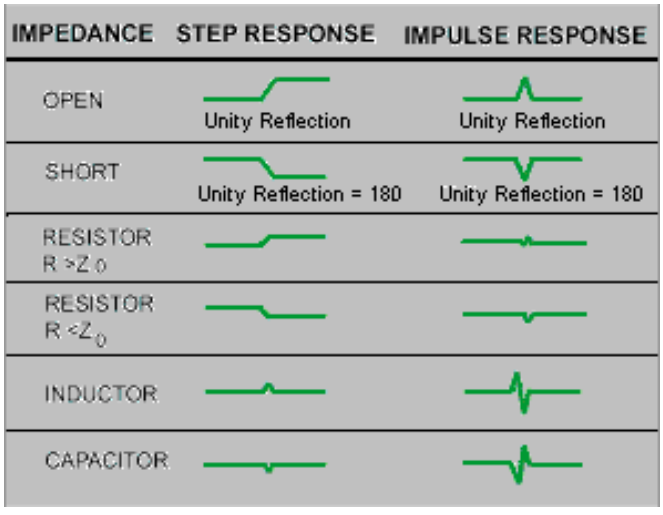


Figure 2.18 Lowpass time domain response waveform of discontinuous circuit

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.

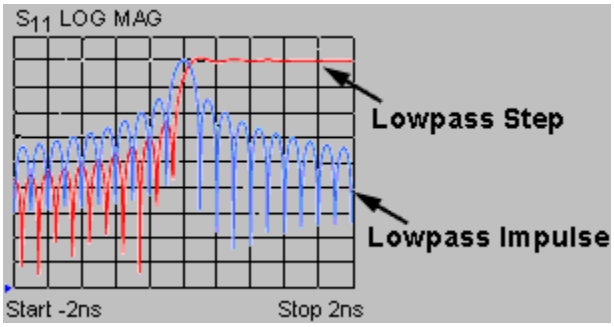


Figure 2.19 Reflection measurement in lowpass mode

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.

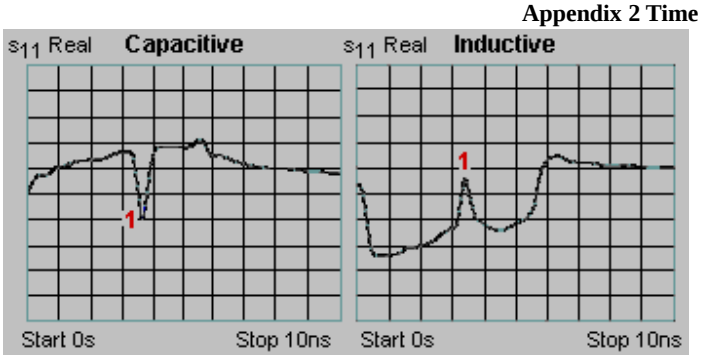


Figure 2.20 Lowpass step measurement results of different cables

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.

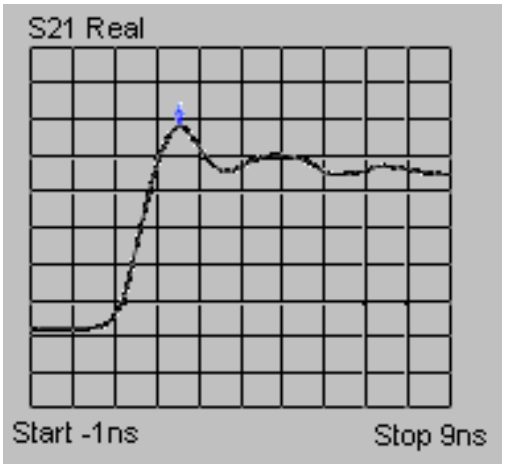


Figure 2.21 Lowpass step measurement results of the amplifier

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.

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.....431
- Mode Selection Based on Test Device.....432
- Frequency Range and Data Points.....432
- Pulse and Step Response.....433
- Use Data Formats.....434

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) Allowing for measurement of some frequencies with start and stop frequencies already set.
4) Only applicable for 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) Allowing for pulse response measurement and step response measurement as the

conversion data includes 0Hz.

5) Allowing for 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.

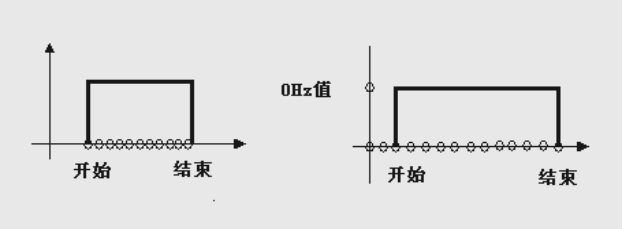


Figure 2.22 Bandpass mode

开始	Start
结束	End
0Hz 值	0Hz value
开始	Start
结束	End

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_{cutoff}=N×F_{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 Nx300kHz at least.

2.6.4 Pulse Response and Step Response

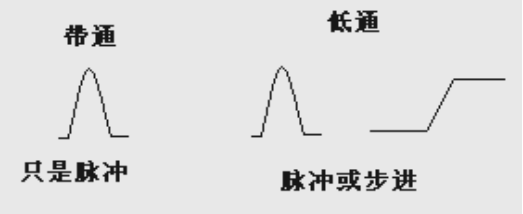


Table 2.23 Lowpass mode

带通	Bandpass
低通	Low-pass
只是脉冲	Only pulse
脉冲或步进	Pulse or step

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].



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.
- Select the gate type in the [Gate Type] box: bandpass, bandstop;
- In the [Gate Shape] box, select the door shape: Min, Nor, Wide, Max.

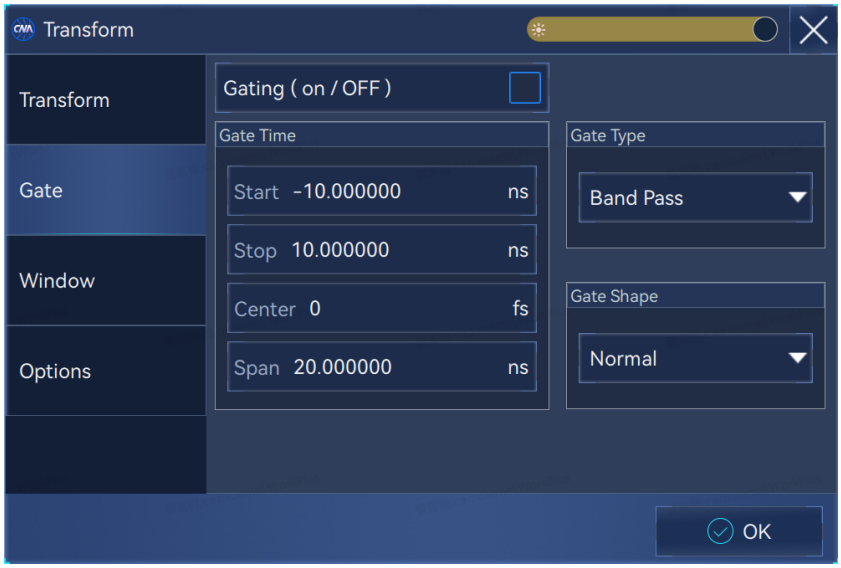


Figure 2.26. Setting of time domain conversion measurement - Set door

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. 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.

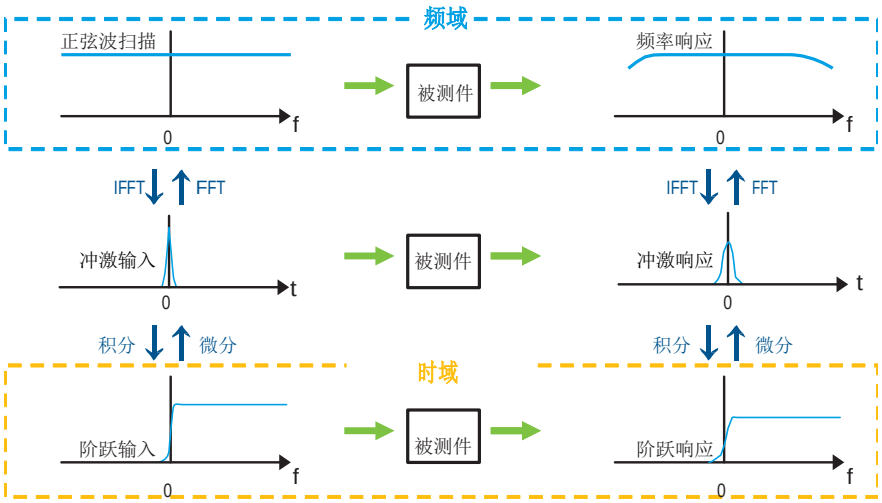


Table 2.27 Comparison of principle of two TDR impedance test methods

正弦波扫描	Sine wave sweep
频域	Frequency domain
被测件	DUT
频率响应	Frequency response
冲激输入	Impulse input
被测件	DUT
冲激响应	Impulse response
积分	Integral
微分	Differential
时域	Time domain
被测件	DUT
积分	Integral
微分	Differential

阶跃输入	Step input
阶跃响应	Step response

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 Setting

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.

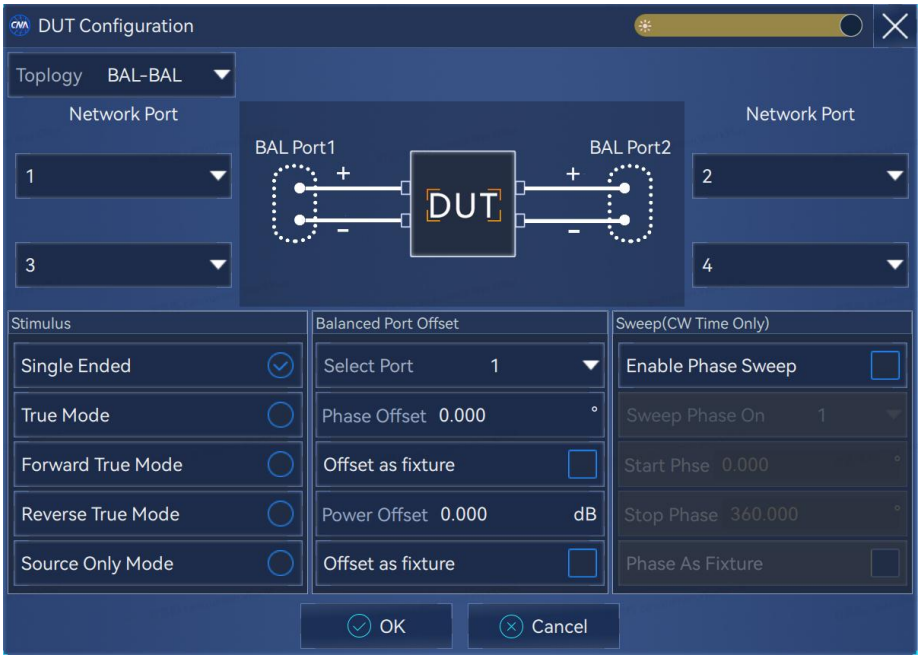


Figure 2.28 Create a test trace

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** area: **[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

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.

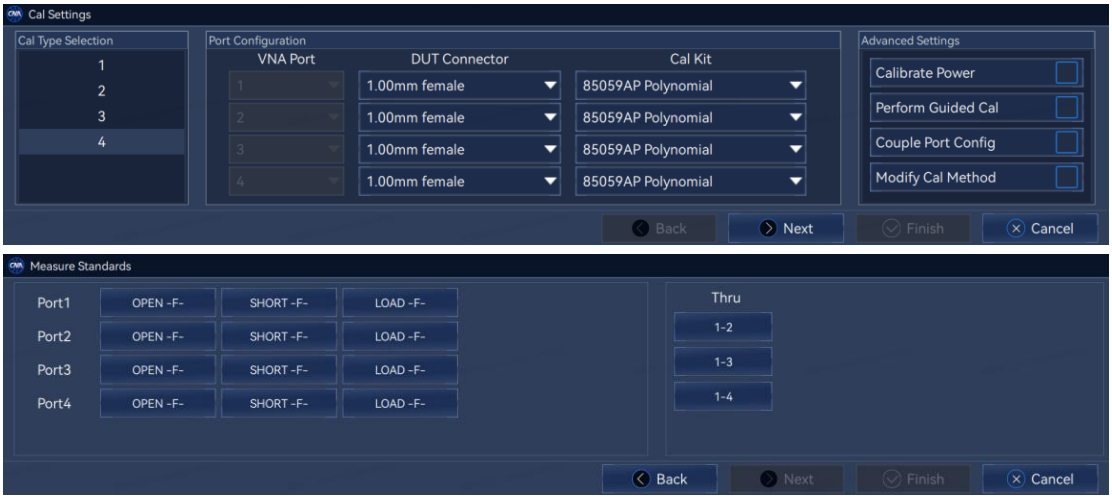


Figure 2.30 Calibration process

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.

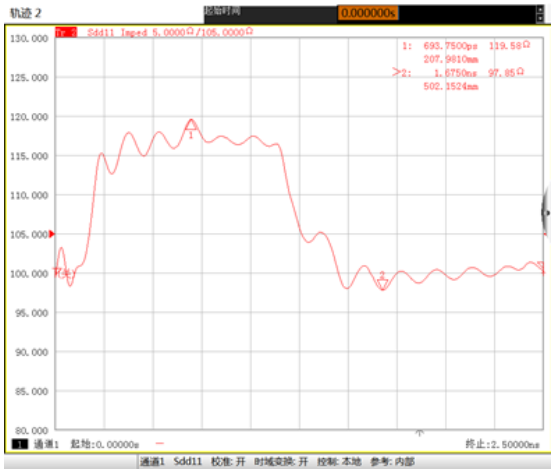


Figure 2.31 Time domain reflectometer (TDR) impedance display

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 Frequency Offset Measurement

- Frequency Offset Measurement Setting.....441
- Coupled/Uncoupled Mode Setting.....443
- Frequency Offset Measurement Calibration.....445
- Typical Applications of Frequency Offset Measurement.....446

3.1 Frequency Offset Measurement Setting

The frequency offset measurement mode allows for adjusting the output frequency of the vector network analyzer source to a value different from the received frequency. The 3674 series vector network analyzer is provided with hardware and software for the frequency offset measurement function. Annex3 mainly introduces the basic settings of frequency offset measurement.

1) Open the Settings dialog box:

- Menu path: click **[Freq]** → **[Frequency Offset...]**.

2) After performing the above operations, the Settings dialog box will pop up, as shown in the figure below. The stimulus and response in frequency offset measurement correspond to the source and receiver respectively, and two methods are available for settings of source and receiver, as described below. The source2 provides signal power for port 3 and port 4, and the source power can be set in the **[Power and Attenuators]** dialog box. Such operation has been described in 4.4, and is not repeated herein.

- Couple main setting method;
- Uncoupled method (which allows for independent setting of values of source and receiver).

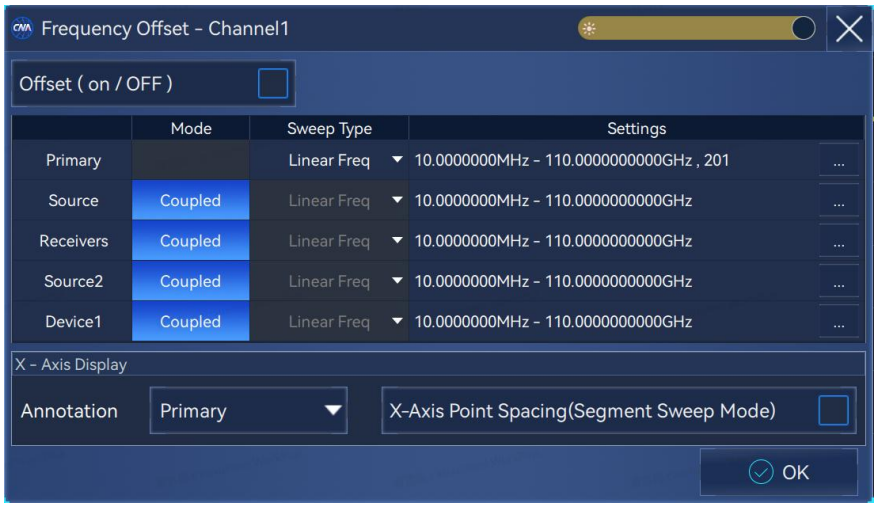


Figure 3.1 Frequency Offset Measurement dialog box

Frequency Offset (on/OFF): turn on the frequency offset measurement of all current measurement channels. When the frequency offset is not enabled, a same signal frequency is applied to all measurement channels;

Main Setting: set the current test channel. In the coupled mode, the source or receiver has the same sweep type as the main setting, and the coupled frequency setting range can be calculated according to the main setting through certain mathematical formula based on the set multiplier and divisor (the setting method of multiplier and divisor will be explained in detail in the next section). Based on this method, when you want to change the source and receiver parameters, it is only required to change the main setting. The main setting parameter can be changed only when the frequency offset is ON.

The main setting is applicable only when both the source and the receiver are in the coupled mode, and is more often applied in non-coupled mode, under which the source and receiver parameters can be set independently. If source1 and source2 are used, please click **[Power]** → **[Power and Attenuators...]** to perform setting.

Receiver: All receivers used in the channel, including the reference receiver, can be tuned to a specific frequency by Settings menu.

Sweep Type: the sweep type set for each item. It should be noted that the following situations cannot be applied at the same time:

- Power sweep and segment sweep cannot be applied at the same time;
- The uncoupled logarithmic frequency sweep mode will produce invalid data in the case of source and receiver offset;
- The coupled logarithmic frequency sweep mode can be applied in the following two cases:

  1) Frequency offset = 0, multiplier = 1, divisor = 1.

  2) Multiplier = 0.

X-axis display selection: select X-axis display mode in the Display window.

X-axis equidistant display: it is only applicable for the segment sweep mode.

Notice

Frequency offset function

When the frequency offset function is enabled, the power of all receivers in the channel, including the reference receiver, will be adjusted to the set value. At this time, the frequencies of source and receiver are different, resulting in inaccurate S-parameter measurement data of all receivers including the reference receiver. In order to measure at the source and receiver frequencies and display accurate measurement results, two-channel measurements must be used. The frequency conversion loss can be calibrated through the formula editor, as described below.

3.2 Coupled and Uncoupled Mode Setting

3.2.1 Coupling Mode Setting

Click **[Freq]** → **[Frequency Offset]** to open the Frequency Offset dialog box, left click the Settings bar, and click the **Edit** menu to pop up the Source Coupling Settings dialog box as shown in the figure below:



Figure 3.2 Source 1 Coupling Setting dialog box

- **Coupling formula:**

  Start frequency = [Start frequency of main setting * (Multiplier/Divisor)] + Frequency Offset

  Stop frequency = [Stop frequency of main setting * (Multiplier/Divisor)] + Frequency Offset

- **Frequency Offset:** it is for setting the frequency offset value in Hz. For mixer measurement, the offset is the LO frequency. The frequency offset range is ±1000THz.
- **Multiplier:** it is for setting the multiplier value in the range of ±1000.
  a) A negative multiplier indicates that the source sweep frequency changes from high to low, and for down-conversion mixer, the input frequency setting should be lower than the LO frequency.
  b) The multiplier is set to 0, and the start frequency is equal to frequency offset.
- **Divisor:** it is for setting the divisor value in the range of 1~1000.

3.2.2 Uncoupled Mode Setting

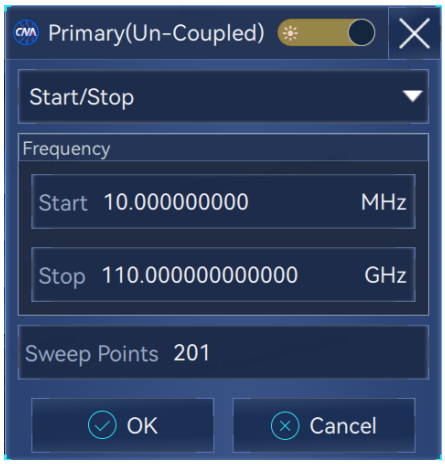


Figure 3.3 Uncoupled Setting dialog box

Each sweep type is provided with individual Setting window, and the setting options of each sweep frequency is described below.

- **Linear and logarithmic sweep:** it is for setting the start frequency or center frequency and span, and also the setting of sweep points.
- **Power sweep:** it is for setting center frequency and sweep points;
- **Spot frequency:** it is for setting point frequency and sweep time;
- **Segment sweep:** Flexible measurement configuration function is provided for segment sweep in uncoupled mode, and in segment sweep setting, attention should be paid to the following:



Figure 3.4 Segment Sweep Setting dialog box

1) Only when the number of total sweep points of effective segment is equal to the number of total points of the channel can the OK button be clicked;
2) Independent IF bandwidth and independent sweep time are valid in the main setting and uncoupled receiver mode only, and are invalid for source mode;
3) Independent power is only valid in the main setting and the uncoupled source mode, and is invalid for receiver mode.

3.3 Frequency Offset Measurement Calibration

In the traditional calibration process, the reference channel is measured to correct the error term. But, the frequency offset measurement stimulus and response have different frequencies, so in this case, the traditional standard calibration such as full dual-port SOLT method is not applicable for error correction. For frequency offset measurement, the source and receiver power calibration shall be applied for error correction.

To open the power calibration option, click the auxiliary button **[Cal]→[Power Calibration]**. For specific operation, refer to "7 Calibration".

1) Source power calibration

- It is for setting an accurate power value for the stimulus frequency without considering the accuracy of the receiver;
- But inter-channel copy function is not available.

2) Receiver calibration

- A calibrated source is required;

- But inter-channel copy function is not available.

3) Mismatch error

A mismatch error will occur when connectors with different impedances are connected between the measurement ports. In S-parameter measurement process, the full two-port calibration can be used to clear this mismatch error. But in frequency offset measurement, such mismatch error is hard to be cleared by frequency offset measurement, and the simplest method is to connect a high-quality attenuator at the input port and the mixer output port, and in this way, the port match is more effective to improve the attenuator value by more than two times, and the larger the attenuator value, the better the improvement effect, but the smaller the dynamic range of the measurement.

4) LO accuracy and stability

In the frequency offset mode, if the LO signal is not accurate enough or unstable, the output signal will fall outside the edge or bandpass of the IF filter, and besides, the LO power value is also critical in the mixer measurement, so it is necessary to ensure the accuracy and stability of the LO power.

3.4 Typical Applications of Frequency Offset Measurement

- Measurement of Frequency Conversion Loss (Gain)……446
- Frequency Conversion Compression Measurement ……406
- Isolation Measurement ……406
- Harmonic Distortion Measurement……408
- Return Loss and VSWR Measurement……408

3.4.1 Measurement of Frequency Conversion Loss (Gain)

Frequency conversion loss can be defined as the ratio of the power at the output frequency to the power at the input frequency for a given LO power. Different LO powers will affect the value of frequency conversion loss. The magnitude of the frequency conversion loss reflects the energy conversion efficiency of the mixer to convert the input signal from the current frequency to the output frequency. If the frequency conversion loss is not flat in the measured frequency range, the output signal will lose some valuable information.

The frequency conversion loss is a transmission measurement, which measures the input signal power value by inputting a specific stimulus and LO signal power. Since the output frequency is different from the input frequency, the measurement must be performed in frequency offset mode.

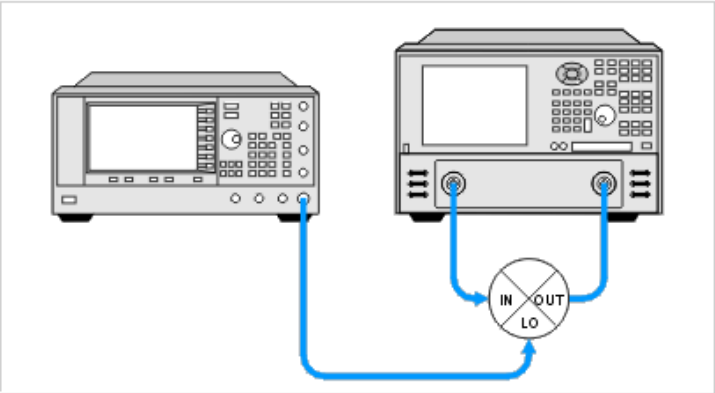


Figure 3.5 Connection device for frequency conversion loss measurement

The following is a simple description of the measurement settings process through an example, on the premise of assuming that the RF signal frequency range is 1-2GHz, the fixed LO is 500MHz, and the output frequency is 1.5GHz ~ 2.5GHz. First, set the sweep frequency range to 1GHz ~ 2.5GHz for source power and receiver calibration. Enable the frequency offset mode, set the source and receiver sweep frequency range according to the measurement requirements, turn on the frequency offset, and click OK. Click **[Measurement] → [Measurement...] → [Receiver],** and select the measurement item to get numerator B and denominator R1 to get the measurement curve. This curve indicates the frequency conversion loss of the mixer.

3.4.2 Frequency Conversion Compression Measurement

Frequency conversion compression indicates the maximum input signal power value in the linear operating region of the mixer, which is very similar to the gain compression in the amplifier. In order to better understand the frequency conversion compression, one should first understand the physical meaning of the frequency conversion loss, i.e., the ratio of the output signal power to the input signal power, which remains constant over a certain input power range. When the input signal power increases to a threshold value, this constant ratio begins to change. The input signal power value whose variation ratio is less than the constant ratio 1 dB is called 1 dB compression power value.

The frequency conversion compression value represents the dynamic operating range of a device; therefore, the frequency conversion compression measurement of the mixer is also essential. The measurement method and connection method are the same as the frequency conversion loss, except that the sweep mode needs to be set to power sweep, and the 1dB or 3dB compression point can be found in the measurement curve by the marker.

3.4.3 Isolation Measurement

Isolation is a measurement of the amount of signal leakage between channels. The greater the isolation provided by the mixer, the less signal leakage to other ports. The frequency of the isolation measurement signal is the same as the stimulus, not the converted frequency. Therefore,

frequency offset mode is not necessary for this indicator measurement. The main measurement terms include: from LO to IF isolation, from IF to RF isolation, and from RF to IF isolation.

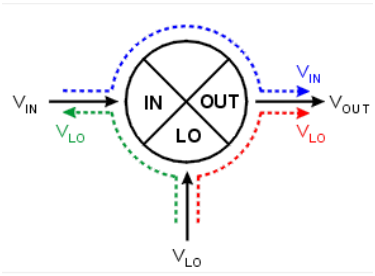
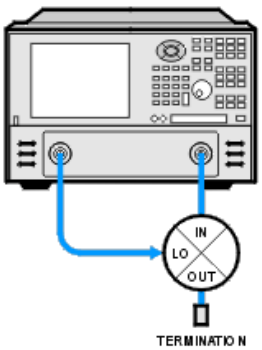


Figure 3.6 Signal leakage representation of the mixer

Any signal leaking into the DUT and the desired signal will generate a new signal, resulting in intermodulation distortion, and this non-desired signal is often difficult to filter out by filtering. The measurement connection is shown in the figure below, and the following two points need to be noted during the measurement process:

1) The size of the isolation from the input port (IN) to the output port (OUT) is closely related to the value of the LO (LO) power, and therefore a properly operating LO power needs to be provided during the measurement;

2) In addition to the two ports under test, another port should be connected to a terminal, and the terminal does not have to choose a specific impedance load (50/75 ohms). For example, if the output port of the mixer is intended to be connected to a filter, then the



filter can be used as a terminal in the process of measurement isolation.

Figure 3.7: LO to RF isolation test

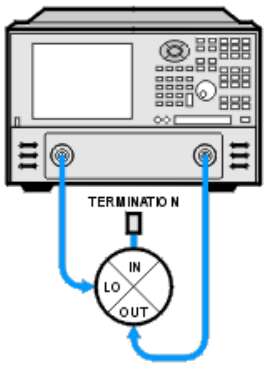


Figure 3.8 LO to IF isolation test

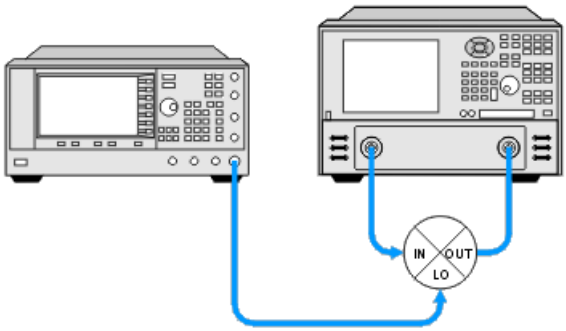


Figure 3.9 RF to IF isolation test

3) The RF-IF leakage test method of converter is the same as that of the mixer, but the converter output often has a filter, so the vector network needs to have a larger dynamic range. Some converters have built-in LO, so different methods are required to measure the leakage from LO to RF or IF:

- Adjust the frequency of the vector network to be the same frequency as the LO, and connect the RF or IF port of the converter to the receiving end of the vector network, with the source not connected to any port;
- You can also use a frequency meter to measure the leakage of the LO, just by adjusting the spectrum analyzer to the frequency of the LO.

3.4.4 Harmonic Distortion Measurement

Harmonic signal is a multiple of the mixer RF input signal or the LO signal, and such harmonics will produce harmonic distortion in the mixer output port. This harmonic distortion is generated mainly due to the non-linear characteristics of the device. Harmonic distortion is different from intermodulation distortion generated by multiple signals. Intermodulation distortion is mainly the distortion generated by intermodulation signals that are produced by two or more signals in the mixer.

The harmonics can be measured using the frequency offset option in the vector network, where the LO frequency of the mixer is set to zero and the harmonic output is obtained by setting the multiplier of the RF input. Since the harmonics unit is dBc, the difference between the RF signal and the harmonics needs to be calculated. Before the measurement, it is necessary to calibrate the source and receiver in the full frequency range and turn on the frequency offset mode. To measure the second harmonics, the multiplier needs to be set to 2 and the third harmonics 3.

3.4.5 Return Loss and VSWR Measurement

Return loss and VSWR are both linear reflection measurements, and the reflection frequency of the converter does not change, so it is also a reflection measurement for the converter. A large return loss reduces the energy of the transmitted signal and weakens its ability to carry information, so this indicator needs to be measured strictly in practice.

During the measurement, it is only necessary to set the measurement mode to return loss or VSWR. The connection is shown in the figure below.

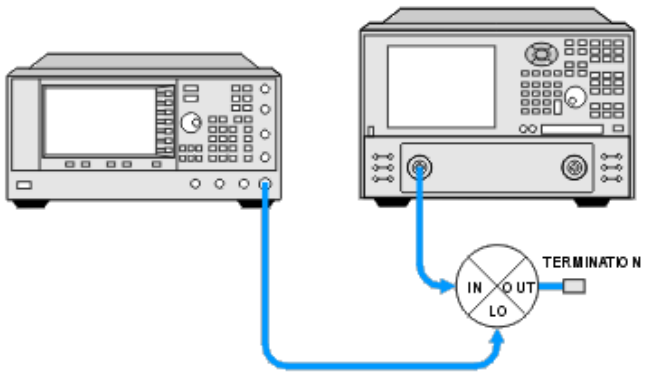


Figure 3.10 Mixer input port return loss test

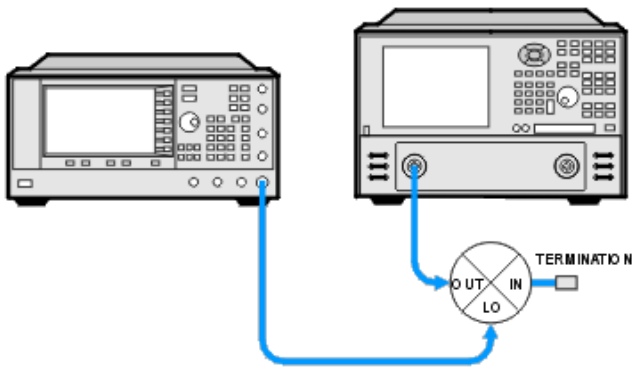


Figure 3.11 Mixer output port return loss test

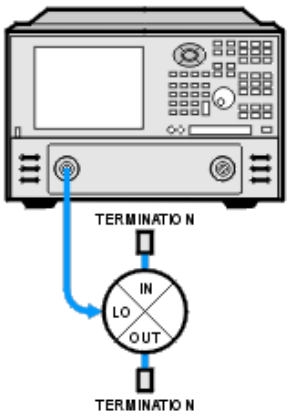


Figure 3.12 Return loss test at the mixer's LO port

Annex 4 Pulse Measurement

- Pulse Measurement Principle 453
- Overview of Pulse Measurement 454
- Pulse Measurement Steps 462

4.1 Pulse Measurement Principle

The 3674 vector network analyzer can use continuous wave signal (CW) as stimulus, but also can output pulse modulated stimulus signal. The characteristics of many components under continuous wave and pulse stimulus conditions are different, and under pulse stimulus, the bias of the amplifier may change and overshoot and ringing may occur. There are also devices for which a continuous wave stimulus signal cannot be applied for testing. For example, many power amplifiers cannot withstand the heat produced in continuous operation; when the devices are tested on a chip, many devices do not have sufficient heat dissipation capability for continuous wave signal stimulus. The use of pulsed signals for testing can make the test state of these devices coincide with the actual operating state.characteristics of the device. In this way, the true characteristics of the device can be obtained while avoiding damage to the part under test due to overheating.

When the RF continuous wave signal is controlled by a baseband pulse period signal, which is then constantly turned on or off, a pulse modulated signal is formed, as shown in Figure 4.1. The frequency of the continuous wave is called the carrier frequency, the continuity time is the pulse width (PW), the pulse repeatability is expressed by the repetition frequency (PRF) or repetition period (PRP), and the duty ratio represents the ratio of pulse width to pulse period.

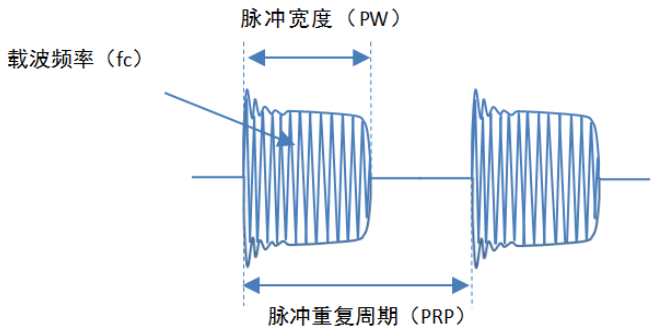


Figure 4.1 Time domain waveform of pulse signal

载波频率(fc)	Carrier frequency (FC)
脉冲宽度(PW)	Pulse width (PW)
脉冲重复周期(PRP)	Pulse repetition period (PRP)

The pulse modulated signal shows a response of sin(x)/x in the frequency domain, as shown in Figure 4.2. The width of each flap in the frequency domain is inversely proportional to the pulse

width (PW), and the interval between the spectral components is equal to the pulse repetition frequency.

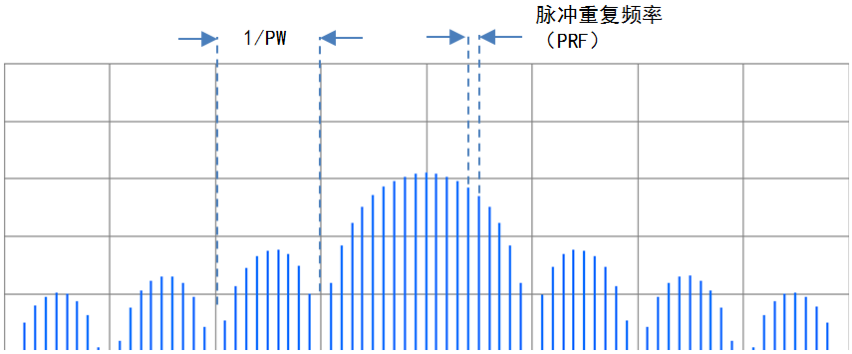


Figure 4.2 Pulse signal spectrum

脉冲重复频率(PRF)	Pulse repetition frequency (PRF)

4.2 Overview of Pulse Measurement

4.2.1 Pulse Measurement Path

The hardware path for pulse testing is shown in the attached Figure 4.3. The network analyzer has a built-in pulse baseband signal generator, which can generate a total of 5 pulse baseband signals, namely P0, P1, P2, P3, and P4. These five signals have the same frequency. P0 is specifically provided to the receiver ADC to control the data acquisition time. In the two-port model, P1-P4 can provide pulse modulated signals for stimulus source port 1; in the four-port model, P1-P4 can provide pulse modulated signals for stimulus source port 1 and port 3, which will modulate and then output the continuous wave. P1-P4 can also provide switching signals for the strobing gate on the IF receiving path, allowing IF signals of a specific time and width to pass through. P1-P4 can be output through the rear panel to the external modulator. 4-channel pulse signals can also be input from external sources through the rear panel to modulate the stimulus source and control the IF strobing gate.

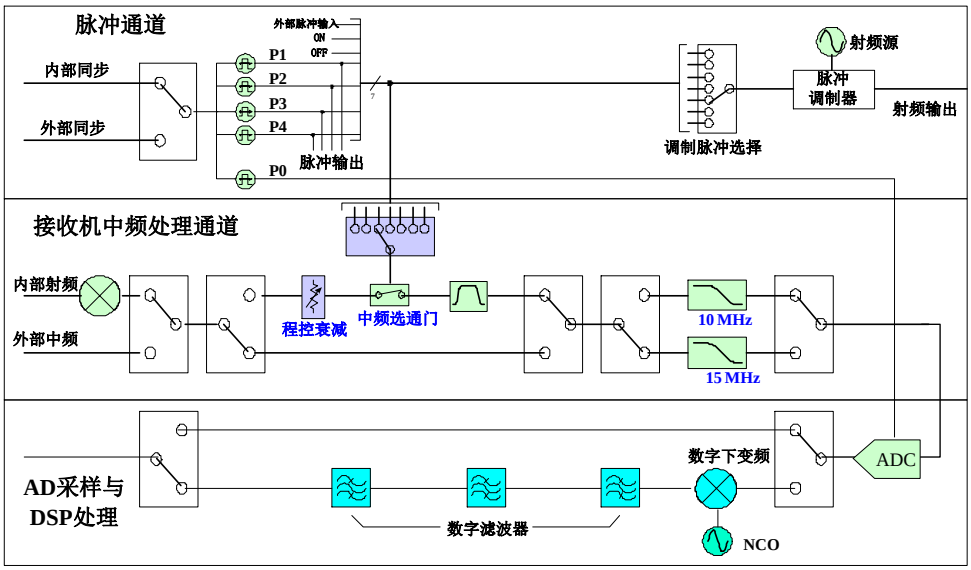


Figure 4.3 Pulse measurement signal path

脉冲通道	**Pulse channel**
内部同步	Internal synchronization
外部同步	External synchronization
外部脉冲输入	External pulse input
脉冲输出	Pulse output
调制脉冲选择	Modulation pulse selection
射频源	RF source
脉冲调制器	Pulse modulator
射频输出	RF output
接收机中频处理通道	Receiver IF processing channel
内部射频	Internal RF
外部中频	External IF
程控衰减	Programmable attenuator
中频选通门	IF strobing gate
AD 采样与 DSP 处理	AD sampling and DSP processing
数字滤波器	Digital filter
数字下变频	Digital downconverter

4.2.2 Pulse Measurement Curve

In the pulse measurement curve:

1) The horizontal axis displays the frequency value during pulse measurement, with each frequency point representing a measurement at that carrier frequency; the time value is showed during envelope measurement, with each point representing a measurement at that time offset; and the time value is showed during pulse point measurement, with each point representing a point in a successive pulse group.

2) The vertical axis shows the S-parameter values under pulse modulation.

4.2.3 Pulse Measurement Method

Pulse measurement is mainly divided into broadband measurement mode and narrowband measurement mode.

(i) Broadband measurement mode:

In the bandwidth measurement mode, the vector network analyzer must be synchronized with the pulse signal, and the data acquisition is performed only when the pulse signal appears. Therefore, the vector network analyzer must have an acquisition trigger signal with the same frequency as the PRF, as shown in Figure 4.4. The delay relationship of this acquisition trigger signal with respect to the pulse synchronization signal needs to be set correctly. Therefore, the bandwidth detection mode is also called synchronous data acquisition mode.

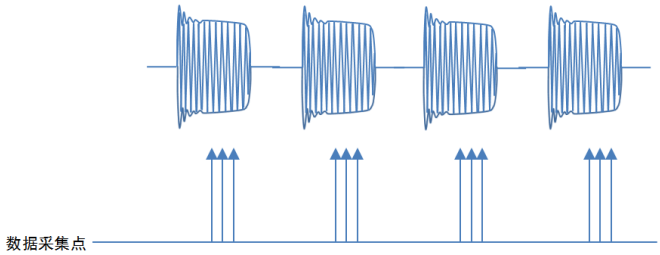


Figure 4.4 Broadband measurement data acquisition

数据采集点	Data acquisition point

The advantages of bandwidth measurement are fast test speed, simple test steps, and no loss of dynamic range when the duty ratio of the pulse signal is low. The smaller the duty ratio of the pulse signal, the longer the test takes. Since the network analyzer always samples the pulse when it is present, the signal-to-noise ratio is essentially constant with respect to the duty ratio signal.

There are two disadvantages to using bandwidth measurements. Compared to narrowband measurements, the noise entering the instrument can be higher because of the wider IF bandwidth used during the test. Another disadvantage is that the measurable pulse width cannot be particularly narrow, and when the pulse duration is less than the acquisition time of the receiver, the pulse cannot be measured correctly. Usually, if the pulse width of the signal needed to test the DUT is wide enough, then bandwidth detection is the preferred method for testing the pulse S-parameters.

There are 3 types of measurements in wideband mode:

1) In-pulse fixed-point test: Data acquisition is performed only within a defined window in the pulse. The data acquisition window is determined by the data sampling period (determined by the IF bandwidth) and the set delay time.

(2) Pulse point test: It is used to characterize how the characteristics of a string of pulses change with time after passing through the device under test. This test needs to be performed at point frequency. The result of the test is a change in amplitude or phase over time. The test points are fixed in relation to a pulse trigger signal in time, and the points on the entire test curve trace are obtained on a successive pulse group, without any missed pulses, as shown in Figure 4.5.

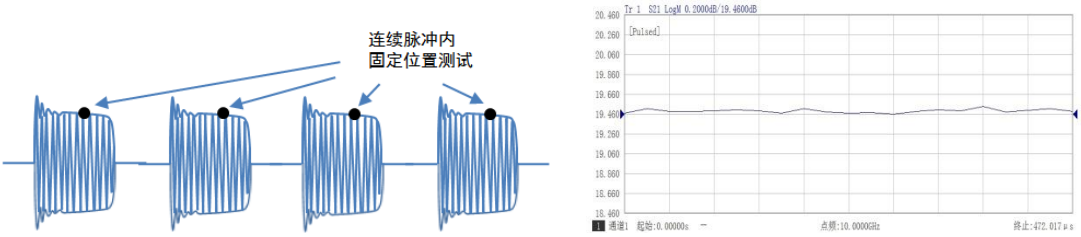


Figure 4.5 Pulse point test diagram

连续脉冲内固定位置测试	Fixing position test within continuous pulse
通道 1	Channel 1
起始	Start
点频	Point frequency
终止	Stop

(3) Pulse envelope test: It is used to test the time domain variation of pulse amplitude and phase. As shown in Figure 4.6, the test frequency is fixed at a certain point frequency, the pulse signal is divided into a number of evenly spaced points in time, and the test points move within the pulse cycle with the increase in delay time. After testing a point, add a delay to the next point until the entire pulse cycle is covered.

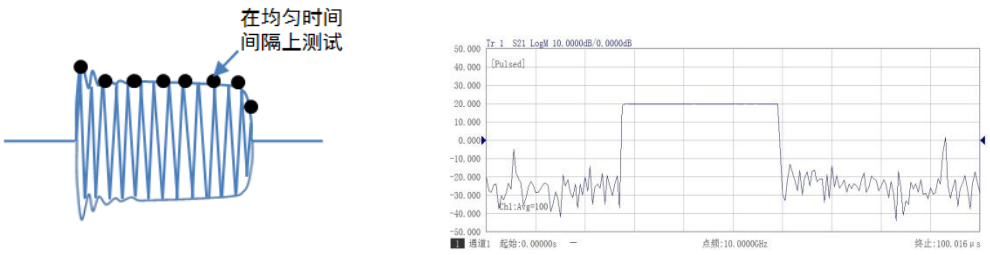


Figure 4.6 Pulse envelope test schematic

在均匀时间间隔上测试	Test within even time interval
通道 1	Channel 1
起始	Start

点频	Point frequency
终止	Stop

(ii) Narrow-band measurement mode:

In the narrowband measurement mode, the receiver IF filter filters out all spectral components except the center spectral component, as shown in Figure 4.7. The frequency of the center spectrum component left after filtering is the RF carrier frequency. After filtering, the RF pulse signal becomes a continuous wave signal, so the data sampling and the input pulse signal do not need to be synchronized, and this technique is also known as asynchronous data acquisition mode.

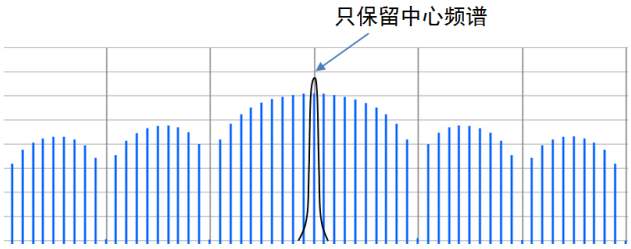


Figure 4.7 Narrowband Measurement

只保留中心频谱	Retain center spectrum only

Narrowband test mode sometimes requires strobe on the IF signal, so that the time window for the IF signal to pass through is controlled by a strobe switch on the IF path. The strobe switch itself needs a synchronous trigger signal to ensure that the strobe switch can be ON within a period of time when the pulse signal appears.

The main advantage of narrowband measurements is that the minimum width of the testable pulse is virtually unlimited, because no matter how wide the pulse spectrum is distributed, most of the spectral components are filtered out, leaving only the center frequency component. The minimum pulse width that can be tested with the 3674 in narrowband mode is 33ns. Another advantage of narrowband detection is that its dynamic range is typically greater than that of the wideband mode because most of the out-of-band noise is filtered out due to the use of narrowband IF filters.

The disadvantage of narrowband measurements is that as the duty ratio decreases, the average power of the pulse decreases, resulting in a lower signal-to-noise ratio and a reduced dynamic range of the test. In the frequency domain, the amplitude of each frequency component decreases as the duty ratio decreases. The loss of dynamic range (dB) due to duty ratio reduction can be expressed as 20*log(duty ratio).

There are 3 types of measurements in narrowband mode:

1) Average pulse test: The data acquisition point is not fixed at a specific position in the pulse signal, but the average value of the whole pulse is tested to obtain the S-parameter. The IF strobing gate is always ON.

2) In-pulse fixed-point test: Data acquisition is performed only within a specified window in the pulse, and the width of the window and the position in the pulse (delay time) are set by the IF strobing gate. For all carrier frequencies, the position of the strobing gate in the pulse is fixed.

3) Pulse envelope test: test the time domain variation of pulse amplitude and phase. The test frequency is fixed at a certain point frequency, the pulse signal is divided into several evenly spaced points in time through the IF strobing gate, the test points move within the pulse cycle, and the strobing gate is delayed to the next point after testing one point until the whole pulse cycle is covered.

4.2.4 Measurement Settings

(I) Broadband measurement mode

In bandwidth measurement, the data acquisition point should be within the pulse, so the pulse width, delay time, and IF bandwidth should be set to meet the test requirements. The time required for data acquisition is determined by the IF bandwidth, and the larger the IF bandwidth, the fewer the number of acquisition points required, and thus the shorter the acquisition time. Therefore, a larger IF bandwidth can be used to test narrower pulse widths, while a smaller IF bandwidth requires larger pulse widths. The relationship between IF bandwidth and data acquisition time is shown in Table 4.1.

Table 4.1

IFBW	Minimum number of sampling points	Required acquisition time per data point
5MHz	15	250ns
1MHz	85	1.4μs
100kHz	639	10.65μs
10kHz	7242	120.7μs
1kHz	55380	923μs

During the in-pulse fixed-point testing, a minimum width of one data acquisition time must be guaranteed. An appropriate delay time also needs to be set, as shown in Table 4.8. If the acquisition moment is not in the right place in the pulse, noise can be introduced into the test results, leading to increased errors.

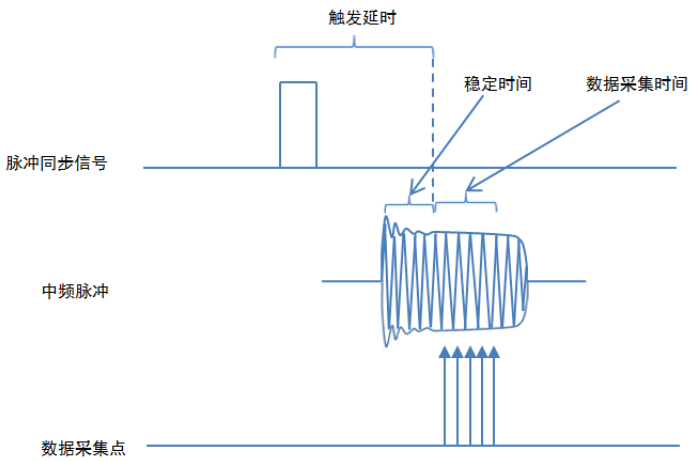


Figure 4.8 Broadband measurement

脉冲同步信号	Pulse synchronization signal
中频脉冲	IF pulse
数据采集点	Data collection point
触发延时	Trigger delay
稳定时间	Stabilization time
数据采集时间	Data collection time

In pulse point testing, each test point represents adjacent pulse data and it must be ensured that the test does not lose any pulse signal. The pulse repetition period is limited by the data acquisition time, and the previous data must be processed before the next pulse arrives. The relationship between the maximum pulse repetition period (PRF) and the IF bandwidth is shown in Table 4.2.

Table 4.2

IFBW	Minimum PRP	Maximum PRF
5MHz	250ns	4MHz
1MHz	1.4us	714.3kHz
100kHz	10.65us	93.9kHz
10kHz	120.7us	8.3kHz
1kHz	923us	1kHz

(II) Narrow-band measurement mode

In the narrowband measurement mode, only the center frequency component is selected for the pulse IF spectrum and all other components are filtered out. In order to obtain better accuracy and dynamic range, the rejection of the IF filter for other spectral components needs to be as large as possible. The correspondence between the IF filter and the pulse spectrum is shown in Figure 4.9. As can be seen from the figure, the magnitude of the bandstop rejection is related to the IF bandwidth and the frequency interval between the pulse spectra, and the closer the spectral line is to the center frequency, the smaller the bandstop rejection is. The interval between the pulse spectrum components is determined by the pulse repetition frequency (PRF), so the IF bandwidth cannot be too large at a certain PRF. After the PRF is set, the IF bandwidth is automatically set to the maximum bandwidth that can be set, and the user can reduce the bandwidth from there. The maximum value of PRF that can be set for 3674 in narrowband mode is 30MHz, and the maximum value of IF bandwidth that can be set is 10kHz.

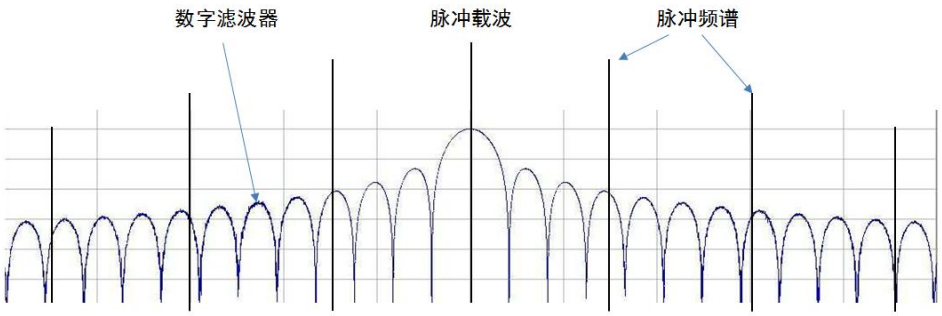


Figure 4.9 IF filtering

数字滤波器	Digital filter
脉冲载波	Pulse carrier
脉冲频谱	Pulse spectrum

In narrowband mode, the IF strobing gate is used to set the location of the test point within the pulse. The strobing gate intercepts the pulse signal from the time domain and keeps only the part of the signal that needs to be tested, thus forming the data points required for in-pulse fixed-point testing or envelope testing, as shown in Figure 4.10. The delay time of the strobing gate and the width of the strobing gate can be set, and must be set to within the pulse signal; otherwise, the signal will not be measured. The strobing gate can also be closed, so that the entire pulse signal will be tested.

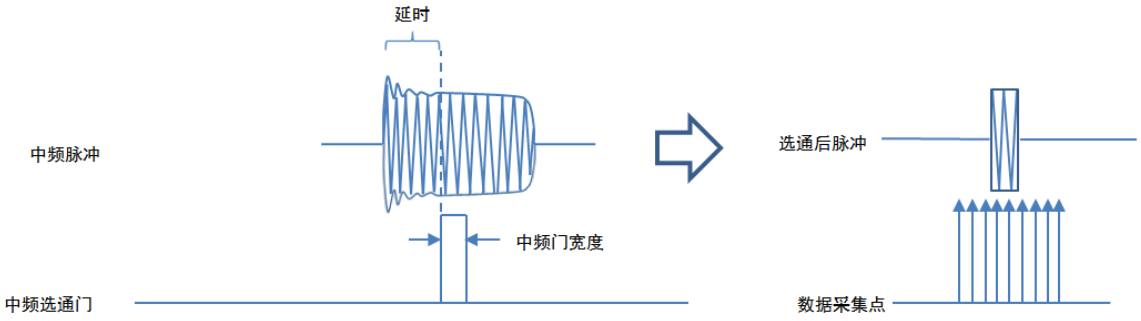


Figure 4.10 Narrowband Measurement

中频脉冲	IF pulse
中频选通门	IF strobing gate
延时	Delay
中频门宽度	IF gate width
选通后脉冲	Pulse after strobing
数据采集点	Data collection point

4.2.5 Dynamic Range of Pulse Measurement

In BW testing, since the data sampling points are inside the pulse, the dynamic range of the test is hardly affected by the duty ratio of the pulse, but only by the IF bandwidth. Reducing the IF bandwidth decreases the noise base and increases the dynamic range, but requires more sampling points and therefore a larger pulse width.

The narrowband test is a test of the average pulse response, and thus the dynamic range decreases as the duty ratio decreases, as shown in Figure 4.11. In Figure a, the pulse width is 10μs, and the bandstop rejection is about -80dB. In Figure b, the duty ratio drops by half, and the dynamic range decreases by about 6dB. In Figure c, the duty ratio drops to 1/10 of the original, and the dynamic range decreases by about 20dB. By increasing the averaging process, the noise base can be reduced to some extent and the dynamic range can be improved, as shown in Figure d.

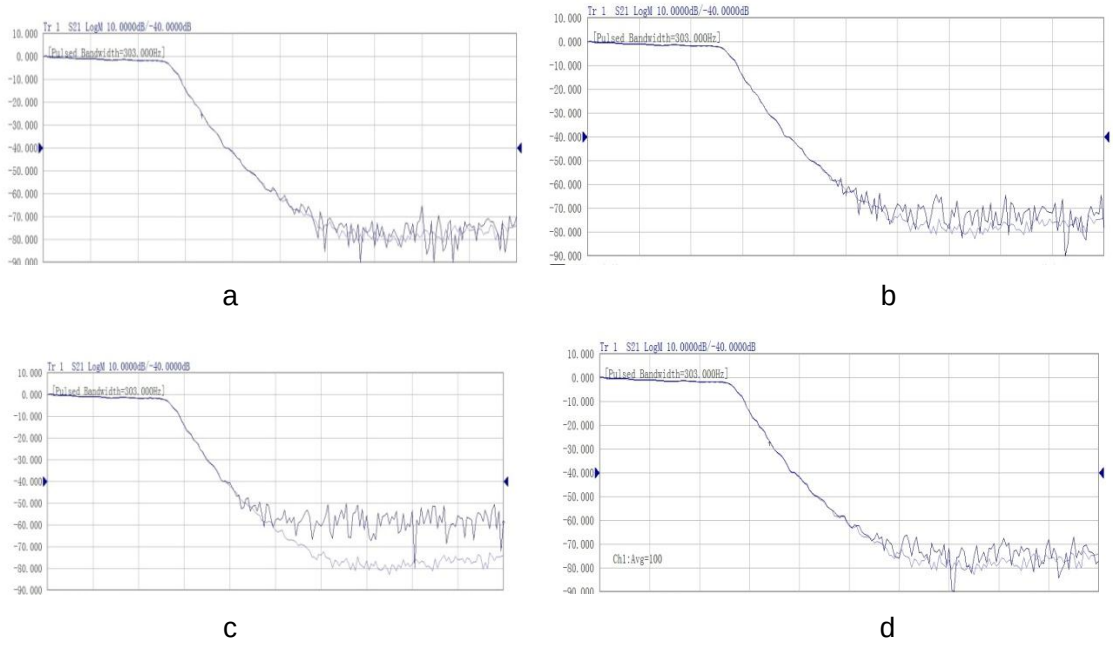


Figure 4.11 Effect of duty ratio on dynamic range

4.3 Pulse Measurement Steps

4.3.1 Reset analyzer

Menu path: **[SYS]→[RST]**.

4.3.2 Pulse Measurement Settings

Menu path: **[Swp] → [Pulse Settings...]**.Then, the Pulse Settings dialog box pops up.

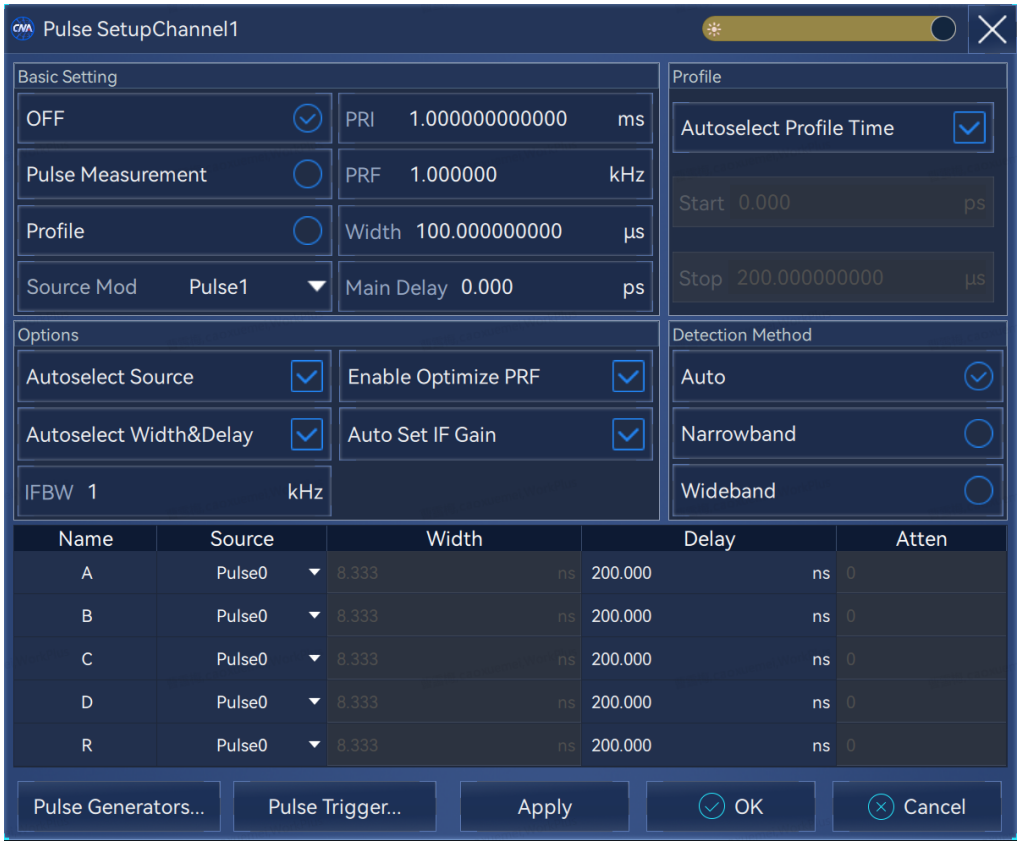


Figure 4.12 Dialog box of starting pulse settings

Basic setting:

- **OFF (continuous wave): The** source and receiver pulse modulation is turned off and in continuous wave operation mode.
- **Pulse measurement**: Turn on pulse modulation S-parameter measurement mode.
- **Envelope (time domain measurement)**: Turn on the pulse time domain measurement mode to display the time domain waveform of the pulse envelope.
- **Source modulation**: Select the source of the pulse signal used to modulate the RF stimulus source from the drop-down menu. There are 7 options: pulse 1, pulse 2, pulse 3, pulse 4, external, on, off.
- **Period:** The repetition period of the pulse signal.
- **Main repetition frequency**: **the repetition frequency** of the pulse signal.
- **Main pulse width**: the width of the pulse signal.
- Main delay time: the delay time of the pulse.

Notice

Pulse parameter settings

The repetition frequency and the period are in inverse relationship. If one of them is changed, the other one will be changed accordingly. The pulse width should be less than or equal to the pulse period.

Options:

- **Auto Select Modulation Source**: Automatically select the pulse source for the IF channel. In broadband mode, pulse P0 is selected as the pulse source for IF channel by default; if not selected, pulse P0 or CW (continuous wave) can be selected. In narrowband mode, by default, the pulse P1 is used to drive the source modulator and pulse P2 is used to drive the IF strobing gates of all A, B, C, D, and R channels when checked; if unchecked, the source modulation and the pulse source of each IF strobing gate can be set manually. Select from seven options: pulse 1, pulse 2, pulse 3, pulse 4, external, on, and off.
- **Automatic selection of pulse width and delay time**: The receiver IF channel pulse width and delay time are set automatically. In wideband mode, the receiver pulse width cannot be set, and the delay time is set to 5% of the source pulse width by default when selected, which will leave about 5% of the pulse enabling time margin for the receiver acquisition; you can set the delay time manually if not selected. In the narrowband mode, when it is checked, IF strobing pulse width is set to 75% of the source pulse and the delay time is set to 20% of the source pulse width by default. You can set it manually if it is not checked.
- **Allow to change the pulse period**: When it is checked, the software automatically sets the period according to the IF bandwidth in narrowband mode to achieve the best filtering effect.
- **Set IF gain automatically**: IF gain cannot be set in wideband mode. In narrowband mode, the software automatically sets the IF gain of the receiver when it is selected; you can set it manually if it is not selected.
- **IFBW: Set** the receiver IF bandwidth.

Envelope:

- **Automatically set the envelope measurement time**: when it is checked, the start time is automatically adjusted to 0 and the end time to twice the pulse width. It can ensure that a complete pulse waveform is displayed. When it is unchecked, set the start and end time manually (these two parameters determine the window size of the displayed pulse envelope. In order to display a complete pulse, the width between start time and end time

must be no less than the source modulation width plus the delay time).

- **Start:** the start time of the pulse envelope.
- End: the end time of the pulse envelope.

Detection mode:

- **Automatic selection of detection mode**: automatically select narrowband or wideband measurement mode according to the pulse width.
- Narrowband asynchronous **detection:** set to narrowband asynchronous measurement mode.
- Wideband **synchronous detection:** set to wideband synchronous measurement mode.

Receiver IF channel pulse setting

It is used to set the pulse parameters of IF channels A, B, C, D and R. Because the data acquisition width is determined by the IF bandwidth in wideband mode, the width cannot be set, only the delay time needs to be set, and the pulse source can only select pulse P0; in narrowband mode, set the width and delay time of the IF strobing gate, and select the pulse source.

- **Source:** the pulse source used to drive the data acquisition or IF gate if CW is selected as continuous wave.
- Width: The width of the IF strobing gate pulse.
- Delay: The data acquisition moment or IF strobing gate pulse **delay**.
- Attenuation: the amount of IF path attenuation.

Pulse generator settings

After clicking [Pulse Generator...], the Generator dialog box pops up for modulating the pulse generator of the signal source. Pulses P1-P4 are used for modulating the source, and pulse P0 is used for synchronous analyzer data processing and pulse generator signal generation. Wideband pulses enable pulse generator P0 by default. Disabling pulse generator P0 will cause distortion of data processed by analyzer.

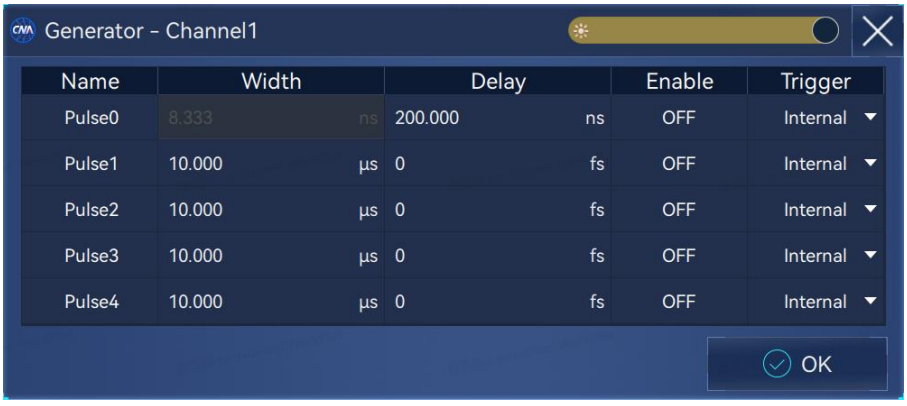


Figure 4.13 Pulse generator settings dialog box

- Width: The pulse width of each pulse generator.

- Delay: The pulse **delay time** of each pulse generator.
- Enable: **Enable the** corresponding pulse generator when it is selected.
- **Trigger: The source of pulse** synchronization signal. When internal is selected, pulse P0 is used; when external is selected, pulse synchronization source is input from the rear panel.
- **Enable:**Used to enable or disable the output of the analyzer's internal 5-way pulse generator.

Click **[OK]** button to make the settings effective after all settings are finished.

Pulse Trigger Setting

After clicking **[Pulse Trigger...]** , the Trigger dialog box pops up, which is used to set the pulse synchronization signal source, or you can set it by [Trigger] - [Pulse Trigger].

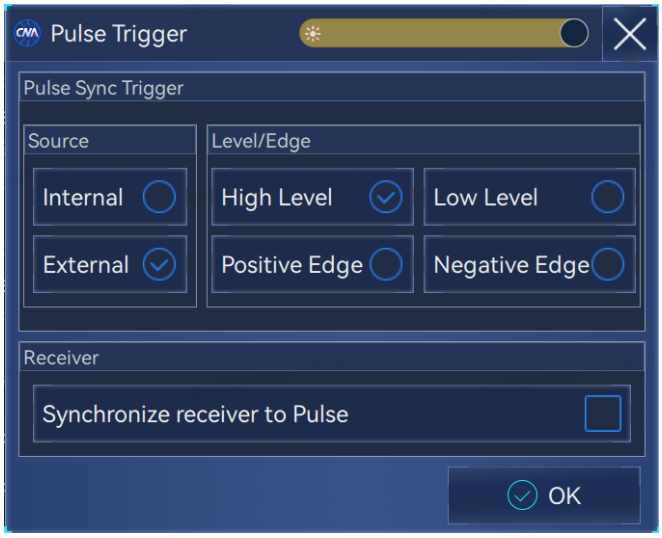


Figure 4.14 Pulse generator settings dialog box

- Synchronization source: When internal is selected, pulse P0 is used, and when external is selected, pulse synchronization source is input from the rear panel. When the triggering mode of one pulse source is changed, all pulse sources follow to become the same triggering mode and cannot be set individually.
- Level/Edge: Set the level and edge of pulse signal.
- **Receiver and pulse synchronization**: When it is checked, the receiver triggers synchronously with the pulse.

Click **[OK]** button to make the settings effective after all settings are finished.

4.3.3 Calibration

After setting is finished, perform pulse measurement calibration **by clicking [Calibration]** → **[Calibration...]**. Follow the wizard calibration to finish all calibrations.

1) Select the port and perform single-port, 2-port or multi-port measurement.

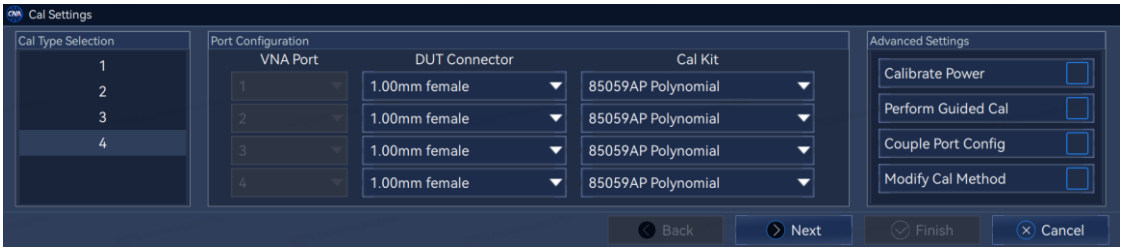


Figure 4.15 Calibration setting dialog box

2) Select the connection type and calibration kit of the DUT according to the connection type of the DUT, because this will affect the subsequent calibration.

Setting instructions:

Port 1: The type of DUT connector connected to port 1.

Port 2: The type of DUT connector connected to port 2.

Calibration kit: The selected calibration kit type. If the electronic calibration kit has been inserted, vector network analyzer will automatically detect it and give the option here.

3) Connect according to the diagram, and click “Measure” after the connection is completed. It will automatically jump to the next step after the measurement is completed. Click “Finish” after all measurements are done to complete all calibrations.



Figure 4.16 Measurement and calibration steps

4.3.4 Print or save data

Click [**File**] → **[Save As]/[Save Data As],** and select the data format and path to **save.**

Annex 5 Amplifier Gain Compression Measurement

This chapter introduces the basic principle and measurement method of the two-dimensional sweep option for amplifier gain compression measurement.

- Amplifier Gain Compression Principle 468
- Overview of Gain Compression Measurements 469
- Gain Compression Measurement Steps 475

5.1 Amplifier Gain Compression Principle

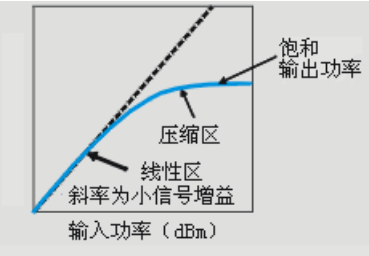


Figure 5.1 Input and output characteristics of the amplifier

饱和输出功率	Saturated output power
压缩区	Compression zone
线性区	Linear zone
斜率为小信号增益	A slope of small signal gain
输入功率（dBm）	Input power (dBm)

The amplifier has a linear gain region in which the input power level of the amplifier is linearly related to the output power level, and the gain in this region is called small signal gain (also called linear gain). As the input power increases and reaches a certain level, the gain decreases, causing the amplifier to approach saturation. We define the input power level that causes the amplifier gain to drop by 1 dB relative to the linear gain as the input power of 1 dB gain compression.

1) The gain of the amplifier at a certain power level is the slope of the curve at this point.

2) When the amplifier works in the linear region, the gain is constant and independent of the input power level, and the gain in this region is usually called small signal gain.

3) As the input power increases, the amplifier gain starts to decrease and the amplifier enters the compression zone.

4) The most common measurement of amplifier compression is the 1dB compression point, which is defined as the input power (sometimes the output power) when the amplifier gain drops by 1dB (relative to the amplifier's small signal gain).

When driving an amplifier with a sinusoidal signal, the output of the amplifier is no longer

sinusoidal at the time of compression. Instead of having only the fundamental wave component of the input signal, the output appears as a harmonic component. As the input power increases further, the amplifier saturates and the output power remains constant. Further increasing the input power of the amplifier at this point does not change the output power. In some cases, regarding the traveling wave tube (TWT) amplifier, the output power will actually decrease as the input power continues to increase after saturation. This indicates that the amplifier has negative gain. Because the amplifier is expected to operate in the linear region, it is important to understand the magnitude of the input signal that causes gain compression.

5.2 Overview of Gain Compression Measurements

- Amplifier Gain Compression Terminology 469
- Gain Compression Parameter Curves 470
- Gain Compression Parameters 470
- Compression Methods 470
- Data Acquisition Mode 472
- Compression Method for Smart Sweep 474
- Factors Related to Measurement Accuracy 475

5.2.1 Amplifier Gain Compression Terminology

Linear Power Level

The input power that puts the amplifier in the linear gain (also called "small signal gain") region.

Reference Gain

A gain that provides a reference for calculating the compression level. If the chosen compression method is different, the specific meaning is also different.

Compression Level

The specific amount of compression that is reduced relative to the reference gain. It is usually 1 dB.

Target gain

The gain at a given compression level (e.g., at 1 dB compression). This is not included in the amplifier gain compression measurement parameters, but is important for understanding the various compression factors.

For example, when compressing from linear gain, the following settings are made:

Linear gain (the gain of the amplifier in the linear region) = 10.2dB

Compression level (fixed value) = 1dB

Target gain = 9.2dB

This is called target gain because the amplifier gain compression will search for the gain closest to 9.2dB. It may not always be 9.2dB.

Compression Point

The gain measured at this point is the closest to the target gain. All compression parameter

data come from this point, including the input power of the compression point, the output power of the compression point, the gain of the compression point, etc.

5.2.2 Gain Compression Parameter Curve

In the amplifier gain compression curve:

The horizontal axis always shows the frequency value. Each frequency point means that the specified compression point is found by measurement and calculation of the power sweep curve at that frequency.

The vertical axis always shows the value of the specific parameter at the compression point. The linear S-parameter displayed for the amplifier gain compression channel is used to measure the transmission and reflection parameters of the amplifier in the linear region.

5.2.3 Gain Compression Parameter

The following parameters assume that DUT input = port 1 and DUT output = port 2

Table 5.1 Compression parameters

	Parameter	Description
Linear S-parameters	S11	Input matching
	S21	Gain
	S12	Reserve isolation
	S22	Output matching
Compression parameters	CompIn21	Input power of compression point
	CompOut21	Output power of compression point
	CompGain21	Compression point gain
	CompS11	Input matching of compression point
	RefS21	It is used to calculate the linear gain value of the compression level. The exact meaning is related to the selected compression method.
	DeltaGain21	Compression point gain - linear gain (logarithmic format)

5.2.4 Compression Method

Compression from linear gain

The reference gain is the gain of the linear region. The target gain is obtained by subtracting a specific compression level (1 dB) from the linear gain. For example: 8.3dB-1dB=7.3dB.

Compression from maximum gain

The linear region of an amplifier gain is not necessarily perfectly linear. The highest gain value measured at each frequency is used as the reference gain (S21). The target gain is obtained by subtracting a specific compression level (1dB) from the maximum gain.

Saturation State Compression

This method gives a better indication of the compression point when measuring the amplifier in the nonlinear region, as shown in the following figure:

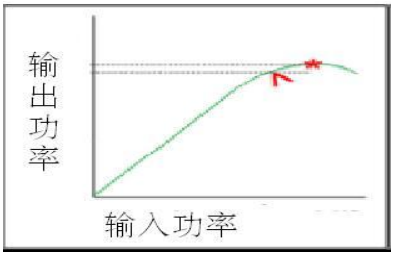


Figure 5.2 Saturation state compression calculation

输出功率	Output power
输入功率	Input power

The maximum value of output power can be found at each frequency*. Then the input power is reduced until the output power is reduced by a specific value (1 dB). This point is called the compression point^.

Compression from fallback points and X/Y compression

These two methods are very similar.

Both methods take advantage of the difference between the linear region and the compression point on the input power (X-axis).

For the Y-axis:

The gain is shown on the Y-axis of the compression method from the fallback point.

The output power is shown on the Y-axis for the X/Y compression method.

The search method of the compression point is different for 2D sweep and smart sweep.

The following figure shows the calculation of the fallback method and the X/Y method at one frequency.

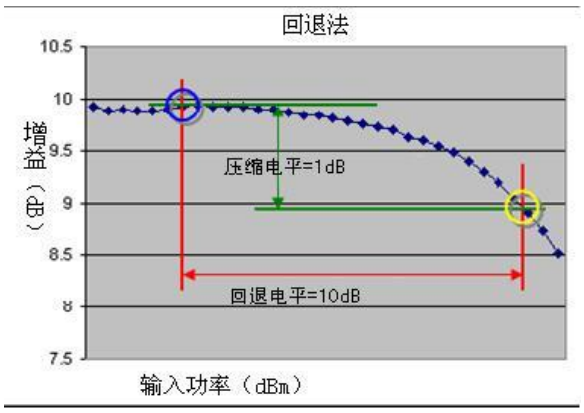


Figure 5.3 Calculation from the fallback point compression method

回退法	Fallback
压缩电平=1dB	Compression level=1dB

回退电平=10dB	Fallback level=10dB
增益（dB）	Gain (dB)
输入功率（dBm）	Input power (dBm)

Note: The compression point (yellow circle) is the point where the input power is 10dB higher and the gain is 1dB lower relative to the reference point (blue circle).

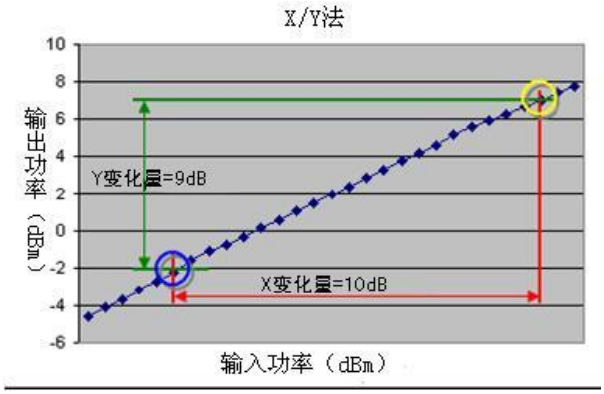


Figure 5.4 Compression calculation by X/Y method

输出功率（dBm）	Output power (dBm)
X/Y 法	X/Y method
X 变化量=10dB	X amount of change=10dB
Y 变化量=10dB	Y amount of change=10dB
输入功率（dBm）	Input power (dBm)

Note: The compression point (yellow circle) is the point where the input power is 10dB higher and the output power is 9dB higher relative to the reference point (blue circle).

5.2.5 Data Acquisition Modes

The amplifier gain compression provides three data acquisition modes: two two-dimensional sweep and one intelligent sweep.

The two sweep modes work in the following ways:

(1) All amplifier gain compression measurements begin with the measurement of the S-parameters at linear power levels. Reflection parameter measurements can only be measured after full port calibration is completed.

(2) Gain measurements are derived from all measured frequency and power values.

(3) After the data measurement is completed, a search is performed to find the compression point. This is usually the insertion point between the two measurement points closest to the target gain.

(4) After each sweep is executed, the current sweep progress is displayed in the lower left corner of the window.

Tips

Compression points of the fallback and X/Y methods

For the fallback compression method and the X/Y method, the amplifier gain compression does not determine whether the start-end range of the power sweep is greater than the fallback or X value. The closest compression point is always displayed.

Two-dimensional sweep mode

(1) Sweep **power at each frequency point -** At each frequency point, it is swept sequentially from the starting power to the ending power.

The example below shows that the device reaches its highest power level (p3=power3) at the first frequency (f1=frequency1). This will cause the device to heat up earlier in the measurement and affect the compression results.

The following example shows the values of three frequency points and three power points, giving a total of nine summary results:

Table 5.2 Sweep power data at frequency points

1	2	3	4	5	6	7	8	9
f1,p1	f1,p2	f1, p3	f2,p1	f2,p2	f2,p3	f3,p1	f3,p2	f3,p3

(2) Sweep frequency at each power point - at each power level the frequency sweep is from the start frequency to the stop frequency in sequence

Figure 5.3 Sweep frequency data at power points

1	2	3	4	5	6	7	8	9
f1,p1	f2,p1	f3, p1	f1,p2	f2,p2	f3,p2	f1,p3	f2,p3	f3,p3

The vector network analyzer does not display all two-dimensional measurement data. It is possible to save the data as a *.csv file and read it in Excel.

Alternatively, you can use the compression analysis function to select a frequency to observe all power sweep information.

➢ Intelligent sweep mode

Intelligent sweep is usually the fastest and most accurate way to measure gain compression. Unlike the two-dimensional sweep acquisition method, instead of measuring all specific frequencies/power points, the intelligent sweep performs a series of power constant value searches. For each frequency, a "intelligent assumption" of the input power is made within the power sweep range to find the input power at the compression point.

The intelligent sweep continues until one of the following conditions occurs:

(1) All data points are within the limits. When the compression level of the data points reaches a specific limit, the input power is changed to continue the measurement to improve the measurement accuracy.

(2) A specific compression level cannot be obtained within the sweep range. Generally, the reason is that the starting power is too high or the ending power is too low.

(3) The maximum number of repetitions has been reached. Before reaching the maximum number of repetitions, if the measured compression point is not within a specific limit, then the last power sweep value will be used as the compression point.

5.2.6 Compression Methods for Intelligent Sweep

The intelligent assumption procedure works differently according to the compression method. When utilizing the fallback and X/Y compression methods, there will be large variations in the input power provided to the DUT during repeated sweeps. This will affect the DUT and the measurement results.

Compression from fallback points and X/Y compression

Because these both sweep methods explicitly separate the linear and compression regions, each repetition requires the same frequency range of signals to be swept at two very different power levels. The first half of the sweep measures the DUT falloff or X power level. The second half of the sweep measures the corresponding start-to-end power range of the DUT at the compressed power level. At the beginning of the second half, the power level increases as the fallback or X value increases. Set a specific time for the DUT to respond to changes in power level. The safety sweep is not to minimize the input power. Rather, it is to prevent the DUT from being affected too much by changes in input power.

Compression from linear gain

After the reference gain is measured in the linear input power region, the next step is to repeat the measurement of the DUT at a high power level where the DUT can be pushed into compression. During subsequent measurements, depending on the power level of the DUT, the power is increased or decreased to achieve the desired compression level. Usually, by the third repeat sweep, a suitable algorithm is available to draw the curve and pinpoint the compression point.

Notice

Effect of power sweep on measurement results

The DUT may change significantly when the power is swept from one time to the next. In this case, the impact can be reduced by using security sweep or by carefully selecting the right settings.

Compression from maximum gain

At each frequency, a maximum gain is saved to calculate the compression point. The intelligent sweep does not perform additional iterative sweeps at each frequency to search for the maximum possible gain of the amplifier.

Saturation State Compression

At each frequency, a maximum output power is saved to calculate the compression point. The intelligent sweep does not perform additional iterative sweeps at each frequency to search for the maximum possible output power of the amplifier.

5.2.7 Factors related to measurement accuracy

The analyzer must provide enough power to drive the amplifier into the saturation zone. If the power level of the analyzer signal source cannot meet requirements, a preamplifier needs to be placed before the amplifier under test to boost the power level. If a preamplifier is utilized, the preamplifier must be included in the calibration. If necessary, the output of the amplifier should be adequately attenuated; too much output power may result in:

(1)The input compression level of the analyzer receiver is exceeded, resulting in inaccurate measurement results;
(2) damage to the analyzer's receiver;

Attenuation of the output power of the amplifier can be accomplished with an attenuator or coupler, as they are an integral part of the test, the impact of the attenuator and coupler must be considered during calibration, and the correct error correction method can reduce the impact of these accessories.

(1) Frequency response is the main source of error in gain compression measurements, and through response calibration can significantly reduce this error;

(2)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.

(3)The accuracy can be improved by reducing the IF bandwidth or utilizing the measurement average, but at the expense of the measurement speed.

5.3 Gain Compression Measurement Steps

- Reset Analyzer.....476
- Open Gain Compression Measurement 476
- Make Gain Compression Settings.....476
- Calibration.....481
- Connect the DUT and Provide Bias 483
- Adding Traces.....483

- Print or Save Data.....484

5.3.1 Reset analyzer

5.3.2 Open Gain Compression Measurement

Menu path: **[Measurement]** → **[Measurement Class]**, select **[Gain Compression],** click **[OK]**. Here, the default is channel 1. If you want to make gain compression measurements in a new channel, you can check the [Create in New Channel] button. If only the [Create in New Window] button is checked, the gain compression measurement will be performed in a new window with the default channel.



Figure 5.5 Creation of measurement dialog box

5.3.3 Make Gain Compression Settings

5.3.3.1 Frequency Settings

Menu path: **[Measurement]** → **[Gain Compression Settings]**. Click **Sweep** to set the frequency, and click **[OK]** to close the dialog box after the settings are completed.

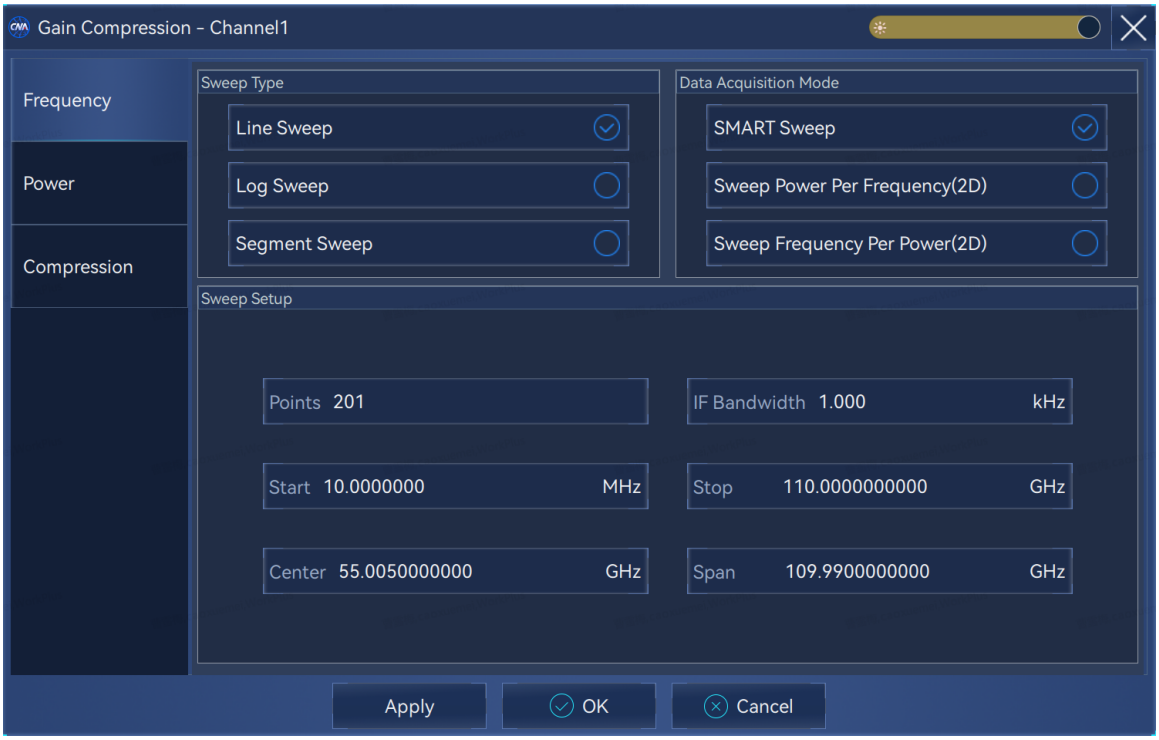


Figure 5.6 Frequency setting dialog box

Sweep setting description:

Number of points: the frequency points to be measured. Frequency points may be limited by the number of power points. See the table below.

IF bandwidth: The size of **the IF bandwidth** affects the trace noise when measured at linear power levels. The trace noise is directly related to the accuracy of the measurement of the compression point. Narrow IF bandwidth improves accuracy, but reduces measurement speed.

Start/End, Center/Span: Set the frequency range to measure gain compression.

Data point limit: The maximum number of data points depends on the data acquisition method and the compression method, as shown below:

Table 5.4 Data point limits at both sweep modes

Compression method	**Intelligent sweep** (reduce the number of frequency points to ensure that the total number of data does not exceed the limit)	**Two-dimensional sweep** (adjust the number of power points to ensure that the total number of data does not exceed the limit)
Linear gain compression	Data points = frequency points Maximum value = 20, 001	Data points = frequency points * power points Power Max = 2, 001 Data Max = 20, 000
Maximum gain	Data points = frequency points Maximum value = 20, 001	

compression		
X/Y or fallback	Data point = 2*frequency point Maximum value 20, 001	

Notice

Frequency and power point settings

The dialog box allows you to enter any frequency and power points, and these values are checked when you press Enter or OK. If the limits are exceeded, the corresponding data will be modified to the maximum value and no error message will appear.

5.3.3.2 Base Config

Menu path: **[Response]** → **[Measurement]** → **[Gain Compression Setting]**. After clicking **Power**, you can set the power. Click **[OK]** to close the dialog box after the setting is completed.



Figure 5.7 Power setting dialog box

DUT input port:

The vector network analyzer port connected to the DUT input.

Linear power level: An input power that ensures the DUT is in the linear region. The linear gain is the reference value from which the linear gain compression is calculated. Input matching is also measured at this power level.

Source attenuator: A vector network analyzer port attenuator connected to the DUT input. The input power range of the DUT is adjusted by this attenuator. All vector network analyzer channels have the same attenuation value for continuous sweep.

Receiver attenuator: The vector network analyzer port attenuator connected to the DUT input. When the power entering the receiver test port is about +10dBm, the vector network analyzer receiver may be compressed. By setting this attenuator, you can make the receiver work in the linear region and get the correct measurement results. However, with the receiver attenuation, the noise of the measured signal will be increased. At this point, the IF bandwidth can be reduced as needed to reduce noise.

Source fixed amplitude mode: The internal fixed amplitude is used by default. Open loop can only be used when making broadband pulse measurements.

DUT output port:

The vector network analyzer port connected to the DUT output.

Reverse Power: The output power of the vector network analyzer port connected to the DUT output port when doing a reverse sweep. This setting is only valid when reverse isolation and output matching are measured or when the vector network analyzer completes a full dual-port calibration for correction. The setting value is not automatically coupled with the input port power.

Source Attenuator: A vector network analyzer port attenuator connected to the DUT output. The range of the reverse output power is adjusted by this attenuator.

Receiver attenuator: The **receiver attenuator** connected to the DUT output port.

Source fixed amplitude mode: Specify the internal fixed amplitude mode

Power sweep:

Power points: The number of power points needed for measurement in a 2D sweep. The power points and the power sweep range together determine the sweep step (which is automatically calculated). Power points may be limited by the number of frequency data points. This setting does not appear in the intelligent sweep, where enough power points are needed to find the specified compression level.

Start and End Power

2D sweep: In the falloff, X/Y and maximum gain compression modes, the reference gain and compression points are measured within this power range. Therefore, it is necessary to ensure that this range is wide enough to include the reference gain and compression points. For example, if the fallback level is 10dB, then the power range must be higher than 10dB. Otherwise, amplifier gain compression will select the closest reference gain and compression point to determine the compression value, making the results inaccurate. For linear gain compression, the reference gain is measured at the linear power level, so the start and end power levels are only used to find the compression point.

Intelligent sweep: Set the power range for searching the compression point. The reference gain of the compression point can be the linear power level, the fallback and the X-value, depending on the compression method. To reduce the number of repetitions needed to find the compression point and to narrow the power sweep, it is only necessary to ensure that the compression point is included, but no linear region should be included.

Tips

Low-band high power mode

If you need more input power below 3.2GHz, click Path Configuration and then click Low Band High Power.

Power step is calculated from the current start, end and point settings, and cannot be changed directly.

Path configuration: Click to open the Path Configuration dialog box.

5.3.3.3 Compression Settings

Menu path: **[Measurement]** → **[Gain Compression Setting]**. Click **Compression**, and click **[OK]** to close the dialog box after the setting is finished.

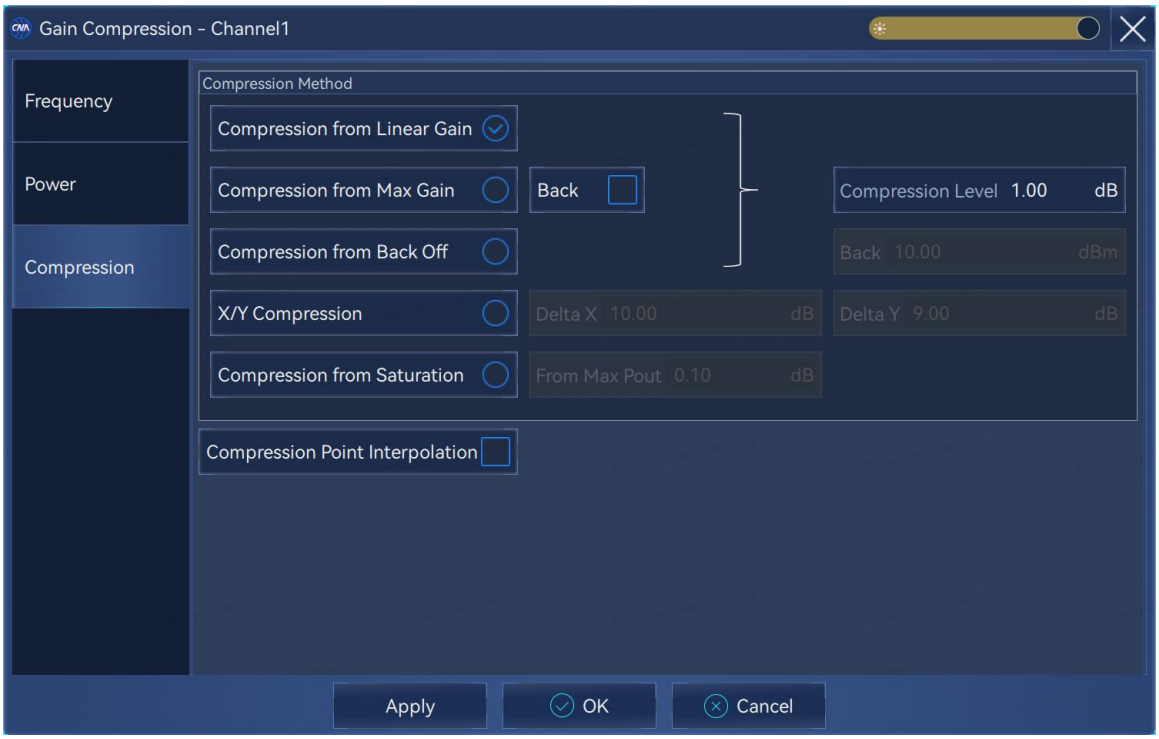


Figure 5.8 Compression setting dialog box

Compression method

Compression from linear gain: The compression level is measured based on linear compression. The linear compression is measured using the linear power level.

Compression from max. gain: The compression level is measured based on the maximum compression level. The maximum compression value is updated with each iteration in the intelligent sweep.

Compression from fallback point: This compression method uses the compression level

and the fallback point to find the compression point.

X/Y compression: This compression method uses specific parameters (X and Y) as criteria to find the compression point.

Saturation compression: Similar to maximum gain compression, except for specific cases, the compression level is measured from the maximum power beyond the level. When measuring amplifiers with non-monotonic gain, this method can be used to better identify the compression point. In the intelligent sweep, the maximum power is updated with each iteration. To increase the chance of measuring the amplifier's true maximum, the safety sweep needs to invoke low accurate roughness increments.

5.3.4 Calibration

After setting is finished, perform amplifier gain compression calibration **by clicking [Calibration] → [Calibration...]**. Follow the wizard calibration to finish all calibrations.

Select the connection type and calibration kit of the DUT according to the connection type of the DUT, because this will affect the subsequent calibration.

Setting instructions:

Port 1: The type of DUT connector connected to port 1.

Port 2: The type of DUT connector connected to port 2.

Power meter: same connection type as port 1.

Calibration kit: The selected calibration kit type. If the electronic calibration kit has been inserted, VectorNet will automatically detect it and give the option here.

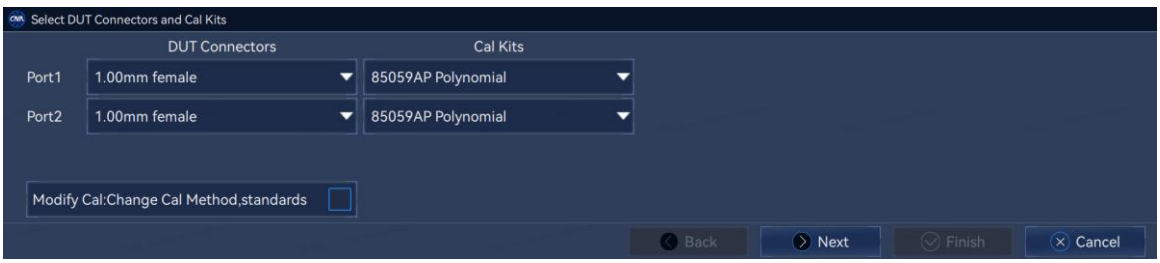
Power calibration port: Select the input port of the amplifier as **the power calibration port**.

Power offset: The deviation value between the amplifier input power and the calibration power, which is usually caused by the adapter or attenuator between the calibration plane and the power meter.If not, this setting can be skipped.

Tolerance: the error tolerance of power calibration.

The maximum number of readings: If the power calibration fails, it will automatically repeat the sweep. The maximum number of readings is the number of repeated sweeps. If the power calibration still fails when the value is exceeded, the sweep will not be repeated and the calibration will proceed directly to the next step.

Power meter configuration: select the **power meter** model; if the USB power probe has been inserted, it will automatically detect and give options.



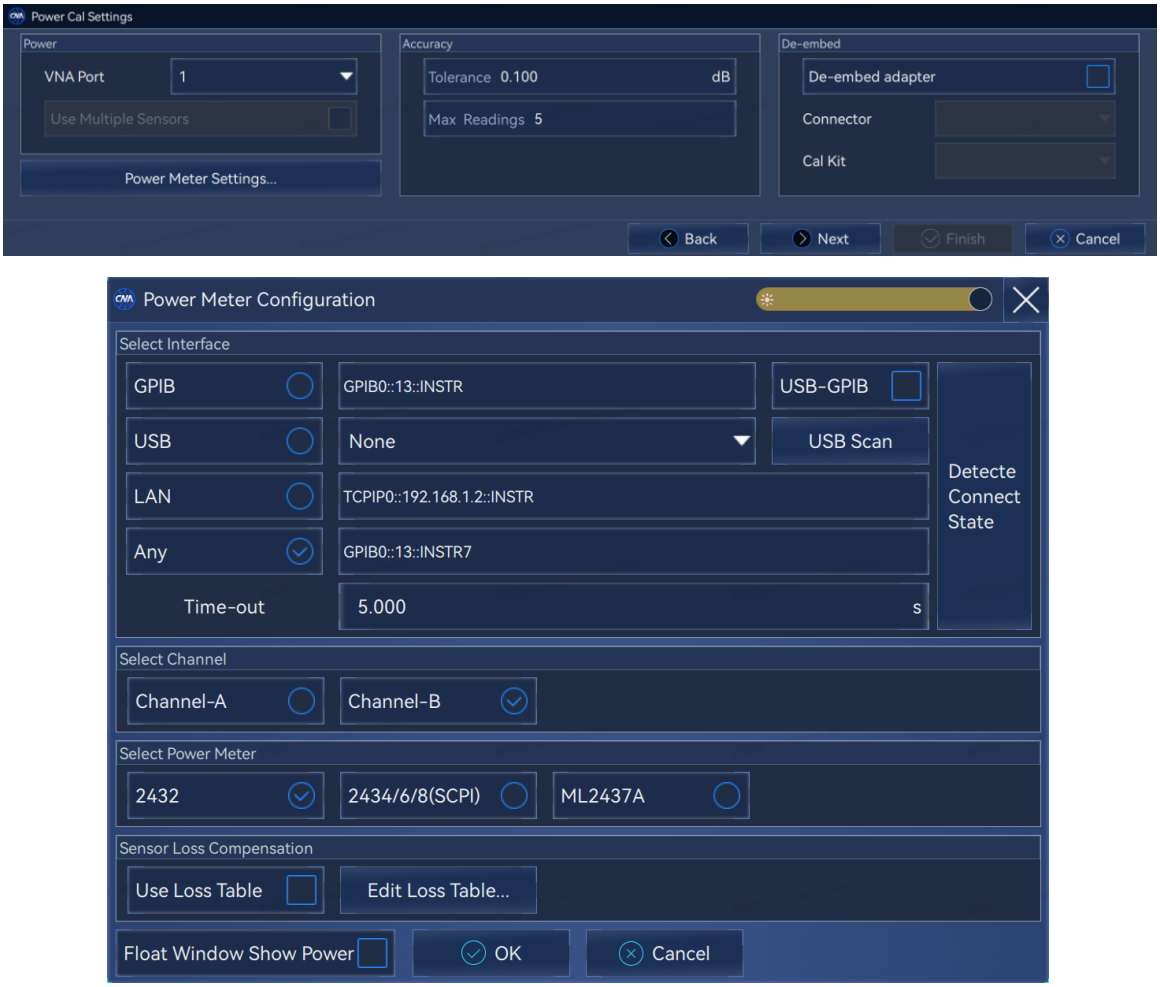


Figure 5.9 Selection of DUT connection type and calibration kit

Connect according to the diagram, and click “Measure” after the connection is completed. It will automatically jump to the next step after the measurement is completed. Click “Finish” after all measurements are done to complete all calibrations.



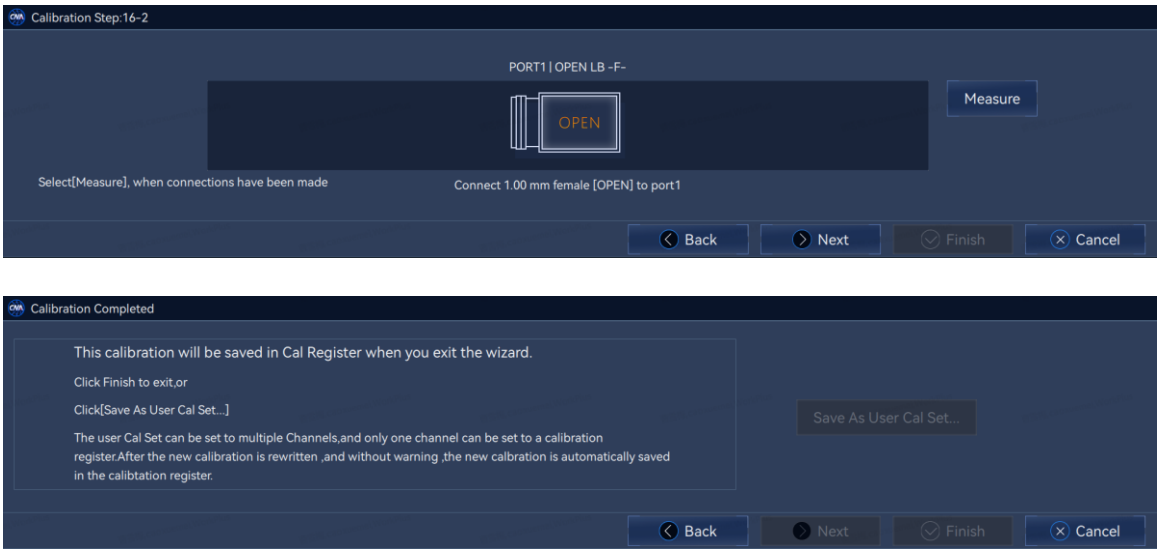


Figure 5.10 Illustration of calibration steps

5.3.5 Connect the DUT and provide bias

Connect the amplifier and provide DC bias as shown in Figure 5.11.



Figure 5.11 Gain-compression measurement connection

5.3.6 Add Traces

The vector network analyzer displays linear gain S21 traces by default; if you want to get more measurement parameters, you need to add traces by clicking [Traces**]** → **[New** Traces]. Check the new trace you want to create, click **[OK]**, and close the dialog box.

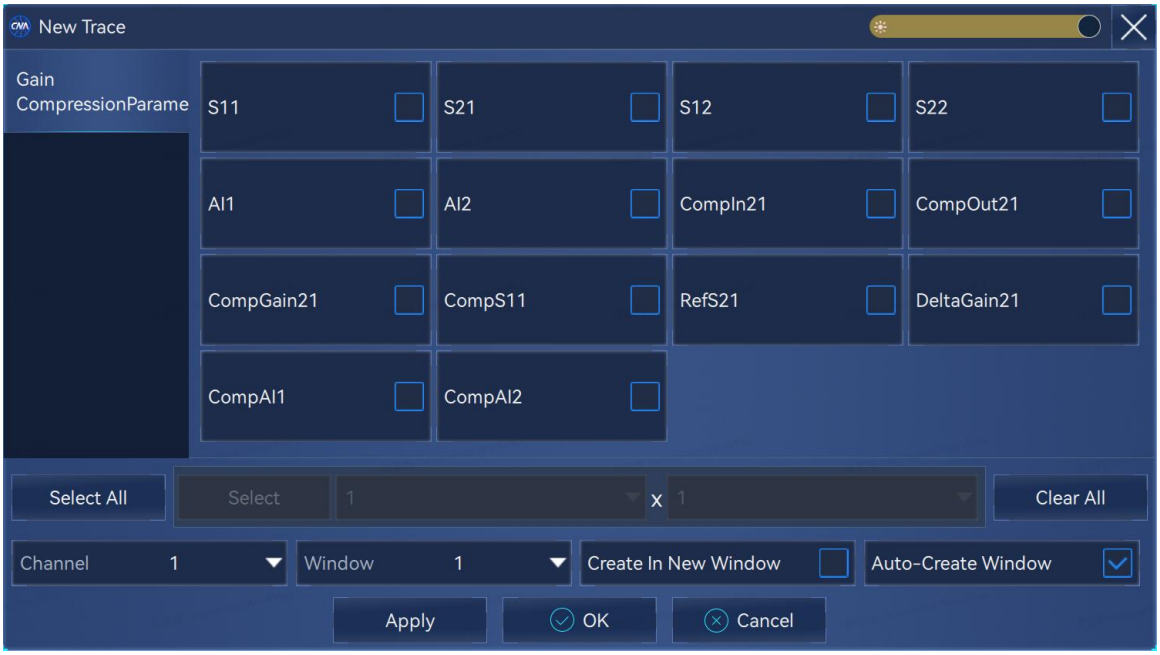


Figure 5.12 New Trace dialog box

5.3.7 Print or save data

Click **[System]→[File]→[Save As]/[Save Data As],** and select the data format and path to **save.**

Annex 6 Mixer Measurement

- Overview of Mixer Measurements.....485
- Scalar Mixer Measurements.....486
- Calibration of Scalar Mixer Measurements 496
- Vector Mixer Measurement 501
- Vector Mixer Measurement Calibration 504
- Embedded LO Mixer Measurement 504

6.1 Overview of Mixer Measurements

Mixer measurements mainly include scalar measurements and vector measurements.

6.1.1 Comparison of scalar/vector mixer measurements

Table 6.1 Comparison of scalar/vector mixer measurements

	Scalar measurement	Vector measurements
Overall comparison	➢ High-accuracy measurement of frequency conversion gain and loss ➢ SOLT and power calibration ➢ Relatively easier setting than vector measurement	➢ High-accuracy measurement of phase and group delay ➢ Calibration requires a combination of SOLT calibration and reciprocal mixer/filter pairs ➢ Relatively complex setting and calibration procedures ➢ Easy to measure converter after calibration is complete
Measure	➢ Supports both forward and backward measurements ➢ DUT can be connected to any port of the vector network analyzer	➢ Amplitude response VC21 ➢ Phase response ➢ Group Delay ➢ The input of the DUT must be connected to the vector network analyzer port 1 ➢ The output of the DUT can be connected to any port of the vector network analyzer
Equipment requirements	➢ Power meter	➢ Reference mixer ➢ Calibration mixer/filter pair (need to meet S21=S12)
	Mechanical calibrator or electronic calibrator	

6.1.2 Measurement Limitations

Tips

Mixer measurements are not available in the following cases:

- Analog frequency sweep
- Logarithmic frequency sweep
- Non-proportional RX measurements
- Electronic calibration as described by the user himself
- Time domain
- Balanced measurements
- Port Extensions
- Some fixture characteristics
- External test setting

6.2 Scalar Mixer Measurements

6.2.1 Perform Hardware Configuration

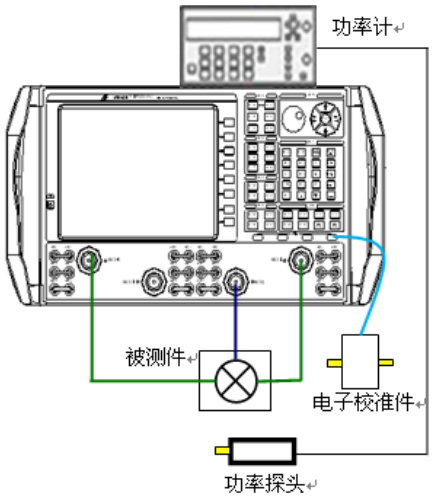


Figure 6.1 Hardware setting

功率计	Power meter
被测件	DUT
电子校准件	Electronic calibration kit
功率探头	Power probe

A power meter, two signal sources and a set of calibration kits are required for calibration of scalar mixer measurements. Among them, there are two kinds of calibration kits, mechanical

calibration kits and electronic calibration kits. When the electronic calibration kits are inserted in the USB port (as shown in the above figure), the whole machine will be automatically recognized. The DUT can be connected to any port of the instrument. The four-port vector network analyzer can use the instrument's internal source2 as the LO input, so there is no need to use an external source.

6.2.2 Setting the Measurement Type

Menu path: **[Measurement]** → **[Measurement Class]**. Select [**Scalar Mixer/Converter], and** click **[OK]**. Here, the default is channel 1. If you want to perform mixer measurement in a new channel, you can click **the [Create in New Channel]** button.



Figure 6.2 Measurement class selection

6.2.3 Make Mixer Measurement Settings

Menu path: [Measurement] → [Mixer Settings], and the pop-up dialog box contains functions such as **[Settings], [Sweep], [Frequency], [Power], and [LO Power]**.

1) Setting dialog box

Click [**Settings**], and the following dialog box pops up.

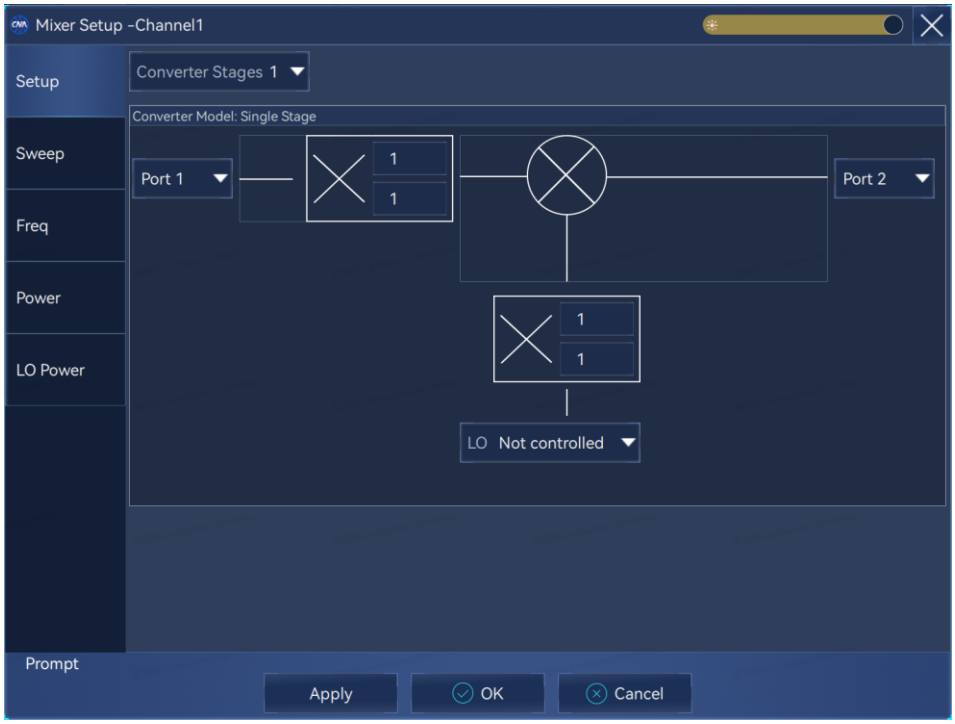


Figure 6.3 Setting dialog box

Conversion order: Click the drop-down box to select the order of conversion, and the default is 1st order. If you select 2nd order, 2 mixing conversions will be performed, and the corresponding conversion model will be the model of 2nd order.

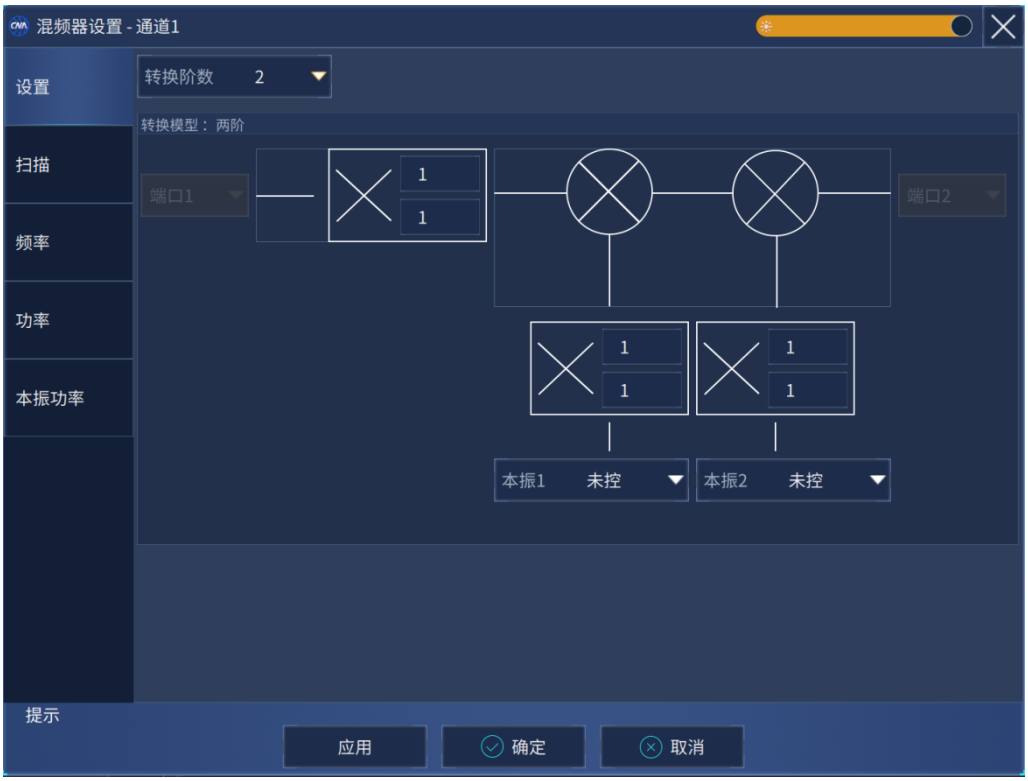
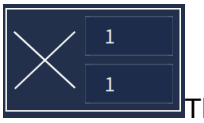


Figure 6.4 Mixer model setting dialog box

Conversion model: Set the **[Input Port]**, **[LO Port]** and **[Output Port]** of the mixer in the Model dialog box. When LO and LO2 are uncontrolled, an external source can be used to provide a fixed LO; otherwise, the instrument port must be used for configuration.

Fractional multiplier:



The combination of (numerator/denominator) forms a fractional value and is multiplied with the input and output frequencies. The coefficients of both the numerator and denominator on the multiplication base are 1 by default and are adjustable from 1 to 10, with no negative values allowed. When the coefficients on the multiplication are changed, the **[Output Frequency Range]** in **[Frequency]** is then changed. The receiver frequency can be tuned to the harmonics of the mixer using the fractional multiplier.

2) Sweep dialog box

Click **[Sweep]** and the following dialog box pops up.



Figure 6.6 Mixer model setting dialog box

a) Sweep Type: See the introduction of the sweep type setting in section 4.5.1 of the manual for details. The default is [**Linear Frequency]**, and [**Point Frequency]**, [**Segment Sweep]** and [**Power Sweep]** can be selected through the dialog box.

b) **[X-axis Equal Interval Display]** checkbox: the default is gray unavailable state. Only when the segment sweep is selected, the checkbox can be checked.

c) **[Spurious** Suppression] checkbox: When checked, the host starts the spurious signal suppression function.

d) **[Sweep Points]** input box: the frequency points to be measured.

e) **[IF Bandwidth]** input box: IF bandwidth used

The IF bandwidth is set to obtain an acceptable trace noise when measuring at linear power levels. The trace noise is directly related to the measurement accuracy of the compression point. A smaller IF bandwidth (narrow IF bandwidth) will improve the accuracy, but the measurement speed will be slow.

3) Frequency dialog box

After clicking [**Frequency],** the following dialog box pops up.

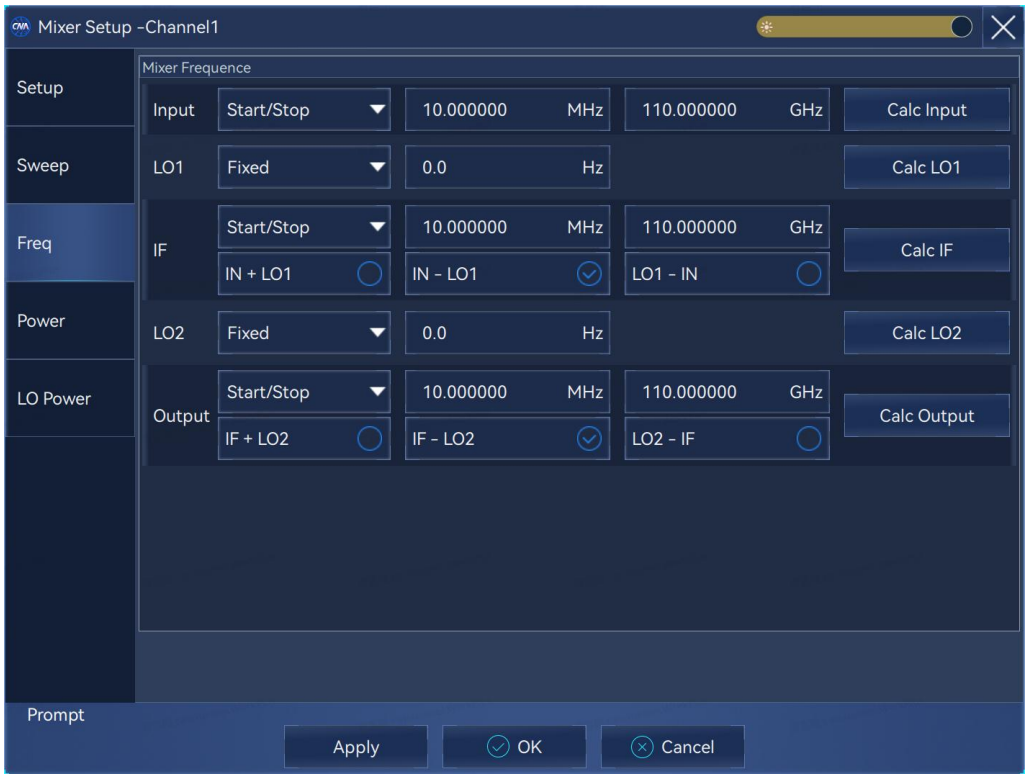


Figure 6.7 Mixer frequency setting dialog box

a) Frequency setting method

- **[Start/Stop]**
  After selecting from the drop-down box, enter the start frequency and stop frequency in the following bands.
- **[Fix]**
  When selected from the drop-down box, enter a fixed frequency in the following band
- **[Center/Span]**
  When selected from the drop-down box, enter the center frequency and frequency span respectively in the following band

b) Frequency calculation method

- **[input + LO]**
  When selected, the calculation will be performed according to the relationship of output = LO + input
- **[Input - LO]**
  When selected, the calculation will be performed in the relationship of output=input-LO
- **[LO-input]**
  When selected, the calculation is performed in the relationship of output = LO- input
- **[Calculate input]**
  After setting the LO and output, click **[Calculate Input]** to calculate the input

frequency by itself.

- **[Calculate LO]**
  When the input and output are set, clicking **[Calculate LO]** to calculate the frequency of the LO by itself.
- **[Calculate Output]**
  After setting the LO and input, click **[Calculate Output]** to calculate the frequency of the output by itself.

Tips

Frequency setting

Select **[Start/End]**, **[Fixed]** and **[Center/Span]** modes by the drop-down box, and one of them must be **[Fixed]**. If the sweep mode in the sweep dialog box is **[Point Frequency]** or **[Power Sweep]**, all of the mixer frequencies are **[Fixed].**

It is not necessary to input all the frequencies. You can set two of them first, and then click **Calculate** to get the third frequency. For example, set the input frequency range and output frequency range, and then click **[Calculate LO] to calculate**frequency range of the LO automatically.

4) Power dialog box

Power setting: Click **[Power]** in the dialog box, and then the following dialog box will pop up.

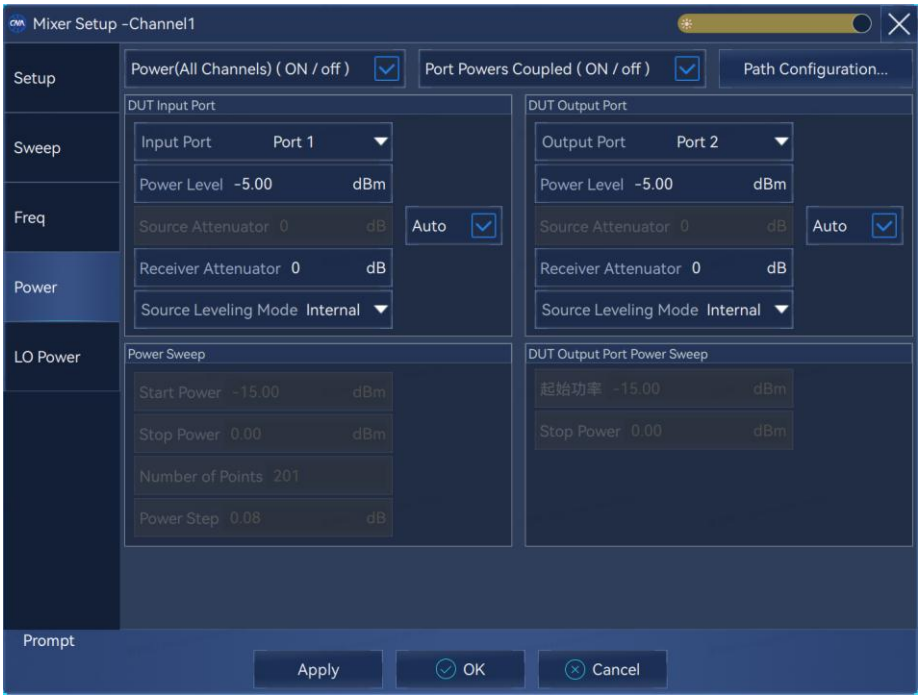


Figure 6.8 Power setting dialog box

a) [Turn Power On (All Channels)]

Control ON/OFF of all channels RF power. It is ON when checked and OFF when unchecked.

b) **[Port Power Coupling]**

Make the power level settings of the DUT input and output ports correlate with each other. When it is unchecked, the power settings of the two ports are relatively independent.

c) **DUT input/output port**

- **[Input/Output Port]**
  Configure the measurement ports for the DUT input and output. Under the vector mixer measurement mode condition, a reference mixer needs to be connected at port 1, so the DUT input can only be connected to port 1 of the test instrument.
- **[Power Level]**
  Set the level of the DUT input port and output port.
- **[Source Attenuation]**
  After checking the auto option, the test instrument can choose the attenuation amount according to the actual measurement needs to meet the test requirements, or you can customize the attenuation value to meet the required input powe. All ports must have the same attenuation value under continuous sweep mode.
- **[Receiver Attenuation]**
  Set the RX attenuation of the DUT port.
- **[Source Fixed Amplitude Mode]**
  The selection options include internal fixed amplitude, open loop (only used in wide pulse measurement mode) and receiver modes.

d) **DUT input and output port power sweep** (available only when power sweep mode is selected in the sweep field).

- **[Start Power and Stop Power]**
  Set the start and stop power of the input/output ports.
- **[Sweep Points]**
  The number of power points to be swept.
- **[Power Step]**
  The step value can be calculated from the current start and stop frequencies, and the number of points swept.

e) **[Path Configuration]**

After clicking, the following path configuration dialog box pops up to set the configuration of the signal path to each port in the channel.

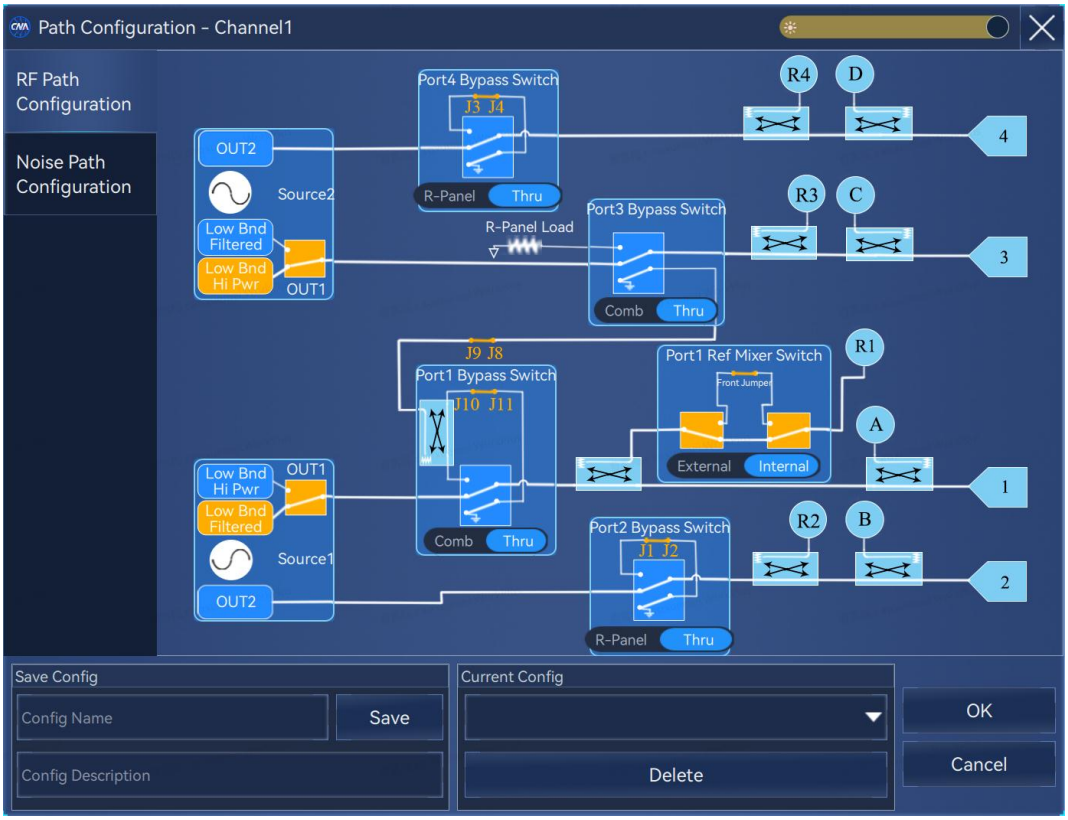


Figure 6.9 Source path configuration dialog box

5) LO Power dialog box

After clicking on [LO Power], the following dialog box will pop up.



Figure 6.10 Mixer LO power setting dialog box

a) Power ON/OFF (all channels)

Click the checkbox to turn on the internal source of all channels of the vector network analyzer, and remove it to turn off the internal source of all channels

b) LO 1

The default is uncontrolled. When the port of LO 1 is set in the Mixer Settings dialog box, the corresponding port will be displayed.

- **Power of [LO 1]**

  Set the power of the LO 1

- **[Source Fixed Amplitude Mode]**

  According to the actual measurement requirements, you can choose **[Internal Fixed Amplitude]**, **[Open Loop]** (only used in wide pulse measurement mode) and **[Receiver]** modes, and **the default is [Internal Fixed Amplitude]** mode.

c) LO 2

The default is uncontrolled. When the port of LO 2 is set in the Mixer Settings dialog box, the corresponding port will be displayed.

- **Power of [LO 2]**

  Set the power of the LO 2

- **[Source Fixed Amplitude Mode]**

  According to the actual measurement requirements, you can choose **[Internal Fixed Amplitude]**, **[Open Loop]** (only used in wide pulse measurement mode) and **[Receiver]** modes, and **the default is [Internal Fixed Amplitude]** mode.

d) Port settings

- **[Source Attenuator Setting]**
  Set the amount of attenuation of the LO
- **[Receiver Attenuation Control]**
  Set the amount of receiver attenuation

e) Sweep power setting

The default is gray. When the sweep mode is **[Power Sweep]**, start the sweep power setting.

f) Path configuration

After clicking, the path configuration dialog box pops up to configure the path.

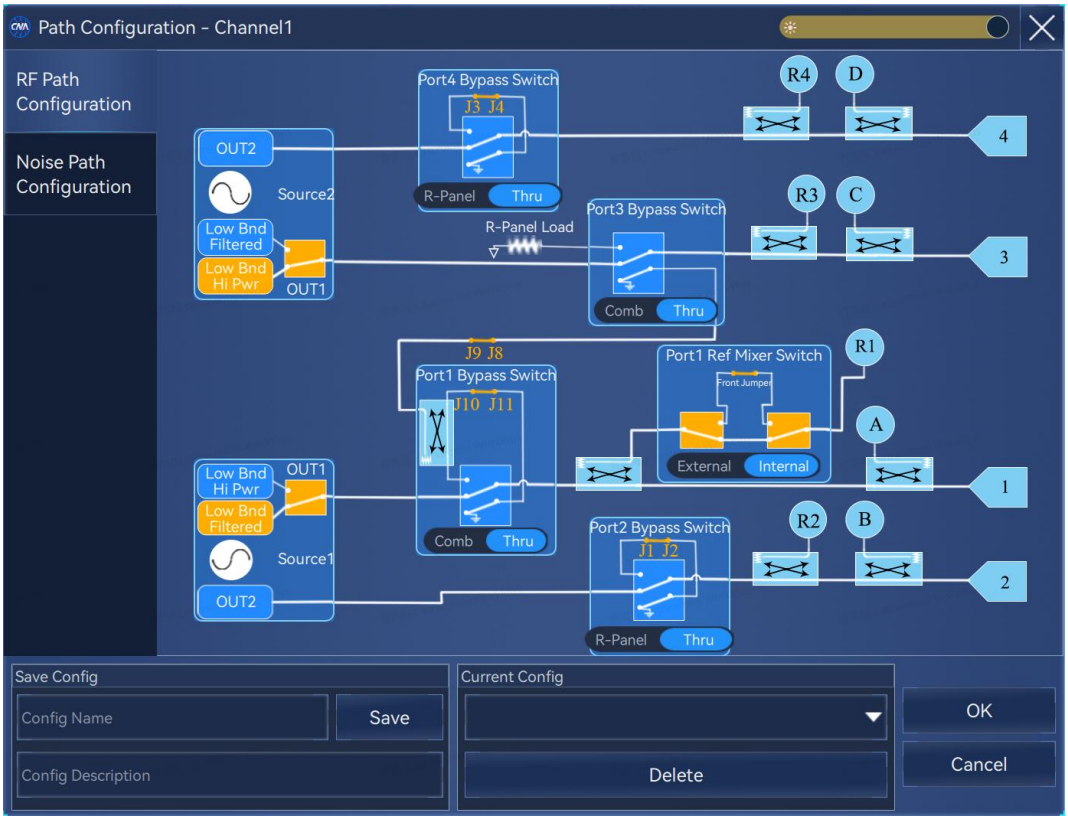


Figure 6.11 Path configuration dialog box for LO

6.3 Scalar Mixer Measurement Calibration

After the mixer measurement setting is completed, calibrate it. By clicking **[Calibrate]** → **[Calibrate...]**, follow the calibration wizard to complete all calibrations.

1) Cal setting

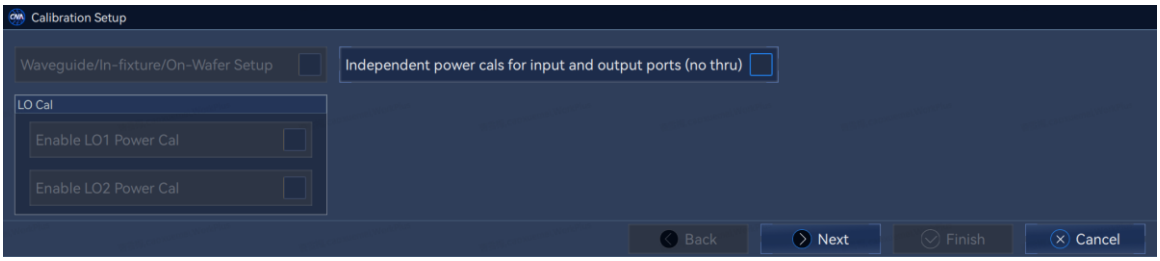


Figure 6.12 Calibration Settings

a) Edit Mixer: You can click this menu to modify the mixer settings

b) Power meter calibration (no through) of input and output: When through calibration is not available, check this option, and then connect the power meter to the input and output ports for power calibration in the subsequent calibration process.

c) Waveguide/fixture/on chip setting: Grey is not available by default.

2) Select DUT connection type and calibration kit

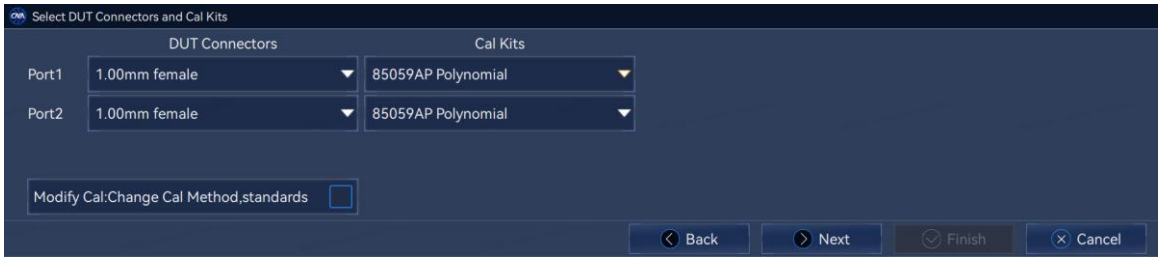


Figure 6.13 Connectors and calibration kits

Select the specific DUT connection type and calibration kit type for different ports.

a) **DUT connection type**:

- **[Port 1]**: The type of DUT connector connected to port 1.
- **[Port 2]**: The type of DUT connector connected to port 2.
- **[Input Power Sensor]**: Same connection type as port 1.

b) Calibration kit: The selected calibration kit type. If the electronic calibration kit has been inserted, vector network analyzer will automatically detect it and give the option herein.

c) **[Power Sensor Input Adapter De-embedding]**

When the connector of the power sensor and the connector of the DUT are not of the same type, in order to make the measurement accurate, it is necessary to measure and correct the adapter located in the reference plane and connected to the power sensor. After it is checked, the measurement and correction of the adapter will be performed in the next calibration steps.

d) **Modify calibration**: After it is checked, click Next. Then, the “Modify Through Connection” dialog box will appear as shown in the figure below.

3) Modify Through Connection

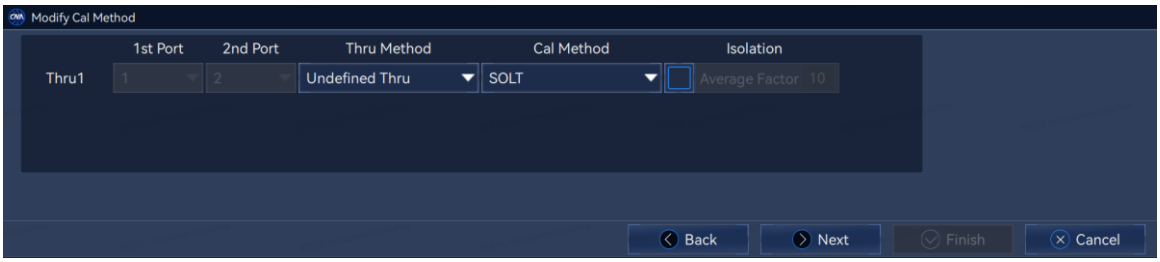


Figure 6.14 Modify the through connection method

a) **Through calibration method**: The through calibration method includes **[Zero Length Through], [Defined Through] and [Unknown Through**]. Select one of them by the drop-down box.

b) **[Calibration Type/Calibration Standard]**: After clicking, the calibration type dialog box in section 6.6.5 will pop up.

c) **[Add Through or Remove Through]**: After clicking **[Add Through],** a through connection method will be added. Click **[Remove Through]** to remove the unwanted through connection method.

4) Port power calibration setting dialog

a) Power

- **Power calibration port**: Select the input port of the amplifier as **the power calibration port**.
- **Power offset**: The deviation value between the amplifier input power and the calibration power, which is usually caused by the adapter or attenuator between the calibration plane and the power meter. If not, this setting can be skipped.

b) Accuracy

- Tolerance: the error tolerance of power calibration.
- **The maximum number of readings**: If the power calibration fails, it will automatically repeat the sweep. The maximum number of readings is the number of repeated sweeps. If the power calibration still fails when the value is exceeded, the sweep will not be repeated and the calibration will proceed directly to the next step.

c) Power meter configuration: select the power meter model. If the USB power probe has been inserted, then click USB detection and check it.

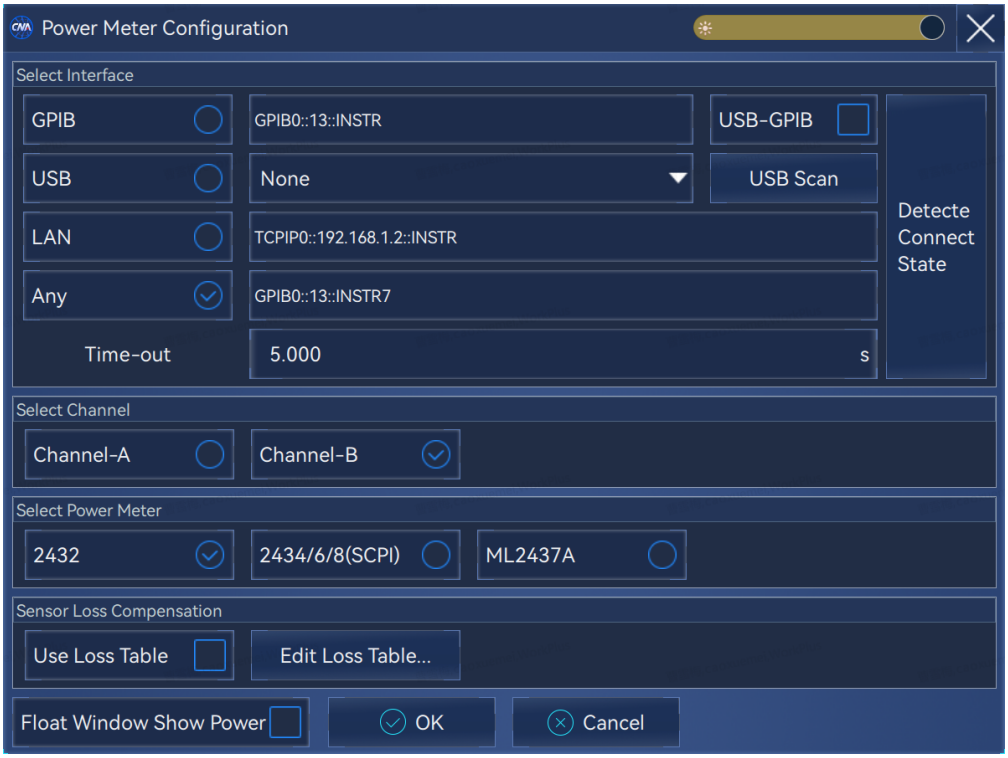


Figure 6.15 Port power calibration setting dialog box

Power calibration level is usually selected as -5dBm. A low power level may produce a very low and noisy calibration. If other components are added between the test port and the calibration reference plane, the power level of the power sensor can be adjusted appropriately to make it slightly higher than 0dBm.

5) Calibration steps

Connect according to the diagram, and click “Measure” after the connection is completed. It will automatically jump to the next step after the measurement is completed. Click “Finish” after all measurements are done to complete all calibrations.



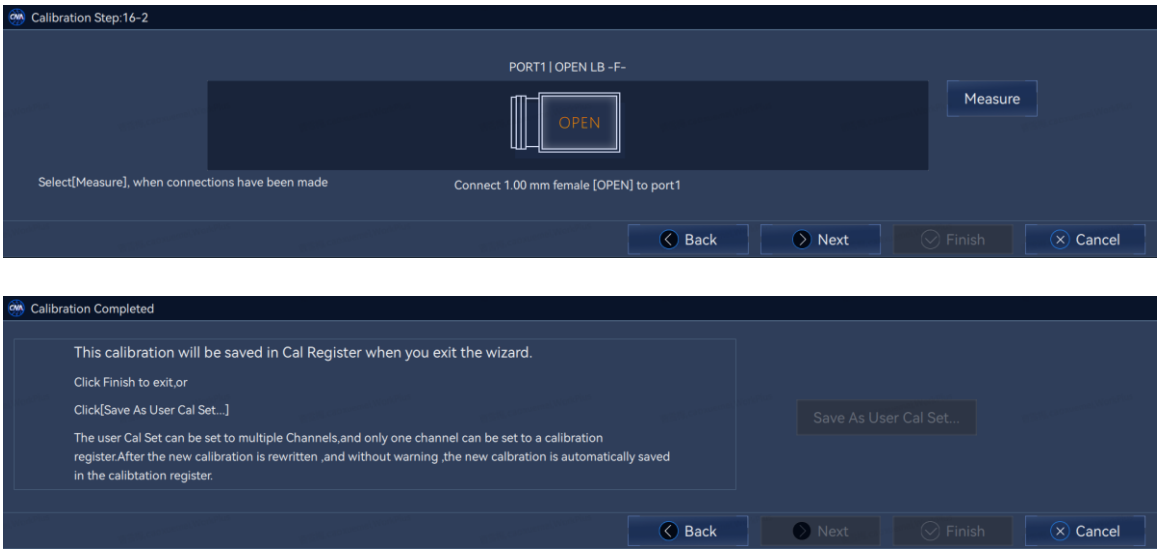


Figure 6.16 Calibration steps dialog box

6) Adjust trace

After calibration, measure the DUT. In order to display more measurement parameters, click **[Trace]** → **[New Trace]**, and select a parameter from the list.

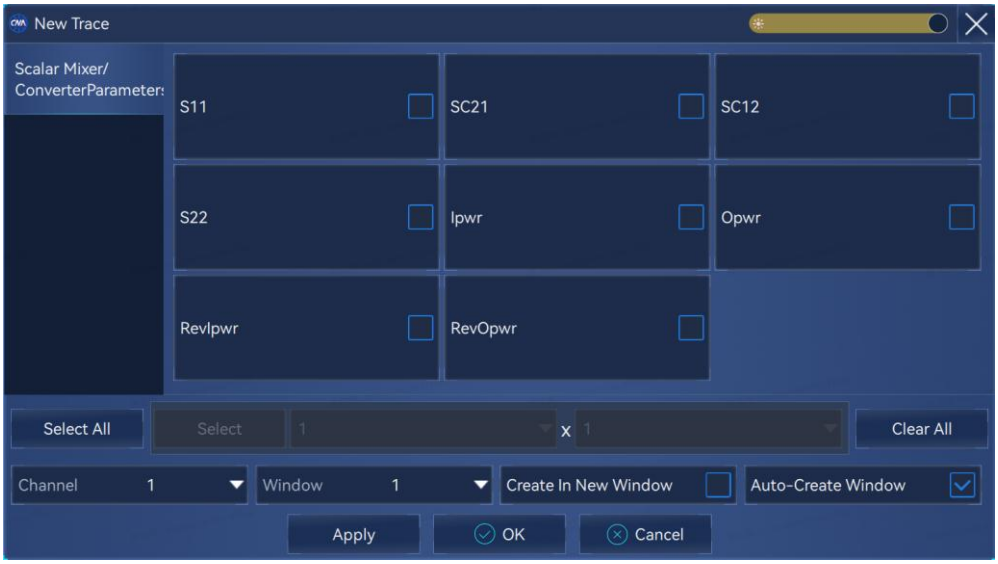


Figure 6.17 New Trace dialog box

The input port is the frequency signal before frequency conversion, and the output port is the frequency signal after frequency conversion. The DUT input and output ports can be connected to any port of the instrument.

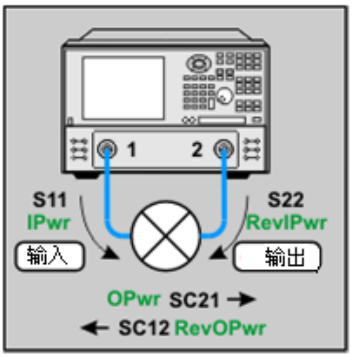


Figure 6.18 Measurement parameters

输入	Input
输出	Output

The mixer parameters are defined as follows:

Table 6.2 Scalar mixer measurement parameters

	Parameter	Description
Proportional parameters	S11	Input matching, indicating stimulus and response at the input port (A/R1)
	S22	Output matching, indicating stimulus and response at the output port (B/R2)
	SC21	Variable frequency loss, indicating stimulus at input port and response at the output port (B/R1)
	SC12	Reverse isolation, indicating stimulus at the output port and response at input port (A/R2)
Non-proportional parameters	IPwr	Input power, indicating stimulus and response at the input port
	RevIPwr	Reverse input power, indicating stimulus and response at the output port
	OPwr	Output power, indicating stimulus at the input port and response at the output port
	RevOPwr	Reverse output power, indicating stimulus at the output port and response at the input port

6.4 Vector Mixer Measurement

1) Vector mixer measurement hardware setting

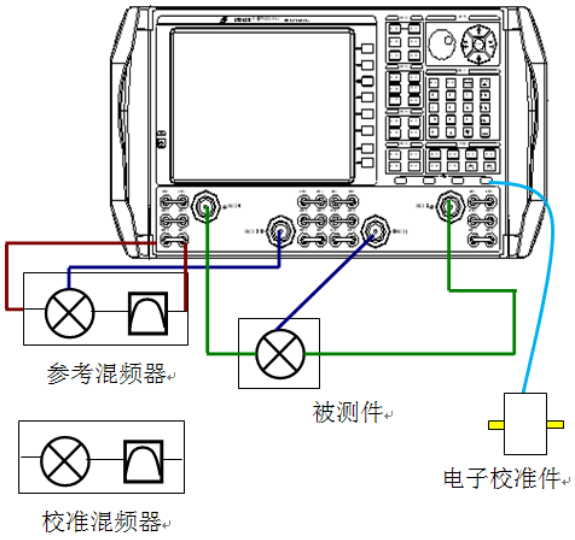


Figure 6.19 Measurement hardware equipment

参考混频器	Reference mixer
被测件	DUT
校准混频器	Calibration mixer
电子校准件	Electronic calibration kit

a) Reference mixer

The reference mixer, which provides the phase reference for the measurement, is connected between the R1 input and R1 output of the reference channel of the vector network analyzer.

The reference mixer, as part of the test system, should remain stationary throughout the calibration and measurement process. The mixer can be controlled by a switch to select whether the reference mixer is controlled manually in the measurement path of the instrument.

The reference mixer does not need to have the same performance as the calibration mixer or the test mixer, and it only needs to have the same frequency coverage as the test mixer. In short, it is necessary to choose a reference mixer that is still available in a variety of different settings. For example, a wideband mixer can be used to replace a narrowband mixer solution.

A low-pass filter is added to the output of the reference mixer, so as to eliminate the LO signal leakage from the mixer and ensure that the instrument does not have any source unlocking and error caused by the LO leakage. Strictly speaking, the filter is not necessary device.

The reference mixer is connected to the instrument in the following way:

i. Connect the reference mixer input to the instrument reference source R1 output;

ii. Connect the reference mixer output to the instrument reference source R1 input.

b) Calibration mixer/filter

The calibration mixer/filter is used as a through standard in the vector mixer measurement calibration and must meet the following requirements:

i. The mixer should cover the entire range of the test mixer in both directions, meaning that it has the same amplitude and phase response in both up-conversion and down-conversion directions, as shown in the following figure:

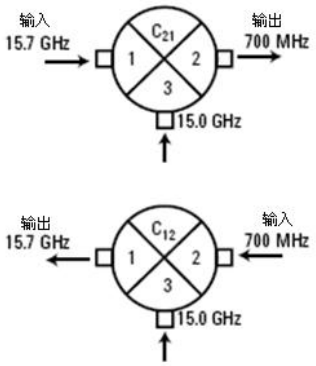


Figure 6.20 Amplitude and phase bi-directional response

输入	Input
输出	Output

ii. If the input and output frequency ranges overlap, the input and output isolation of the mixer should be at least 10 dB greater than the frequency conversion loss of the overlapping portion of the frequency.

iii. At the output of the calibration mixer, the filter must be able to filter out the unwanted frequency mixing products and pass through the desired frequency mixing products.

c) Regarding the calibration mixer/filter and the reference mixer direction

The reference mixer direction is always the same as the mixer under test to ensure that the output frequency is the same as the DUT. The calibration mixer/filter can upconvert and downconvert to characterize the LO source and can use internal source 2 as the LO signal.

2) Establishment of vector mixer measurement

a) Using the auxiliary keys, press [**Measurement]** and then [**Measurement Class]**;

b) Select [Vector Mixer **Measurement]**, then click [**Confirm to Delete the Current Measurement]** and click **[New Channel]** to create a measurement in the new channel;

c) A VC21 measurement is set up and displayed as complete;

d) If you want to choose to add additional parameters, click [**Trace]** and then [**New Trace]**,

and select any one parameter from the list.

3) Vector mixer measurement parameters

The DUT input is usually connected to port 1, and the DUT output can be connected to any port.

a) VC21,VC31,VC41 - frequency conversion loss or gain by default, stimulus at the input, response at the output;

b) S11 - stimulus and response at the input;

c) S22, S33, S44 - stimulus and response at the output;

d) R1 or R - stimulus at the input, absolute power measured at the receiving end of R1 (uncorrected);

e) B, C, D - stimulus at the input, absolute power measured at the output (non-corrected).

4) Vector mixer measurement setting

Menu path:

[Measurement] → [Mixer Setting]

The following dialog box will appear on the screen, and the mixer settings are the same as the scalar mixer settings.

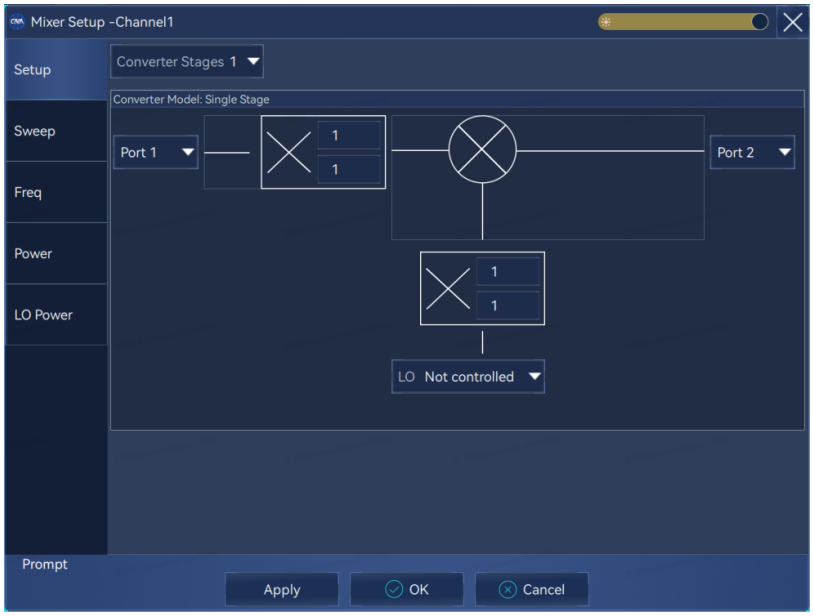


Figure 6.21 Vector mixer measurement calibration dialog box

6.5 Vector Mixer Measurement Calibration

The calibration wizard guides the user to complete the calibration process, where the first three steps describe the calibration mixer, which is used as a through standard in the calibration process.

1) Perfrom the dual 2-port SOLT calibration over the entire covered input frequency range of the DUT and the full 2-port SOLT calibration over the entire output frequency range of the DUT using either mechanical or electronic calibration kits.
2) Add a LO and describe the input-output matching of the calibration mixer/filter with the output bing open, shorted and loaded. Save the S2P file after calibration so that the data can be recalled for vector mixer measurement calibration when the same stimulus settings are made;
3) Connect the reference mixer to port 1 and then to the front panel reference channel between R input and output, and connect the calibration mixer/filter output port to port 2.
4) Measure the calibration mixer/filter as a through standard;
5) Calculate the error term and perform the phase measurement correction of the test mixer/converter.

Start calibration: [Calibration] → [Calibration]

1) Calibration mixer characteristics help dialog box

The following dialog box appears by clicking Next:

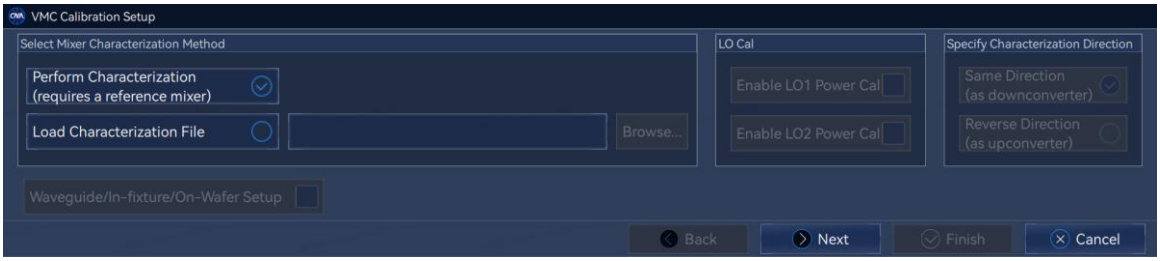


Figure 6.22 Calibration mixer characterization

- Complete characterization (**reference mixer required**): it indicates that a mixer characterization is implemented in addition to the vector mixer measurement calibration. The mixer characterization file will be saved after the calibration is completed. If you do not have a characterization file to load, select it.
- Load a characterization **from a file**: Load a S2P calibration mixer characterization file and click Browse to load the file.

Tips

Mixer characterization

- For segmented vector mixer measurements, a *.S2PX file needs to be loaded.
- The frequency range of the S2P file must be the same or greater than the frequency range of the frequency conversion measurement. If the S2P file has a larger frequency range, an interpolation method is used to match the measurement.
- The connection direction of the calibration mixer must be the same as the direction in the characterization file. Since the direction is not characterized in the file, careful connection

is required.

- The characterization calibration of a mixer may also be performed separately.

2) DUT connection type

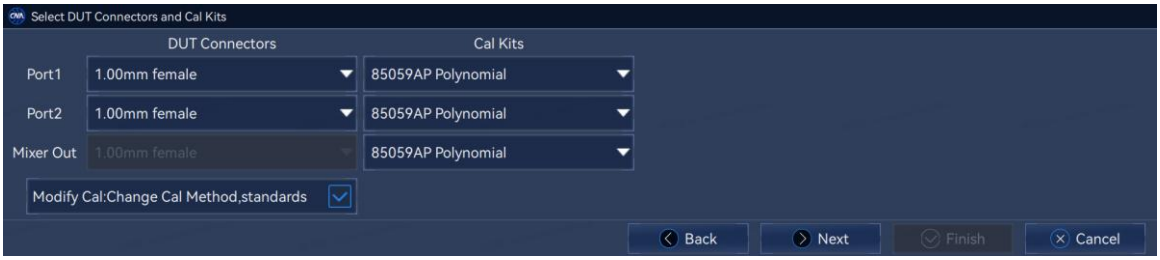


Figure 6.23 DUT connection type and calibration

A specific type of DUT connection and calibration kit can be selected for each port.

[Mixer Output]: Connected to the filter output of the calibration mixer, specify the calibration kit/standard to measure the calibration mixer/filter.

[Modify Calibration: Change Calibration Method, Standard]: Check and click Next to start the Modify Calibration dialog box.

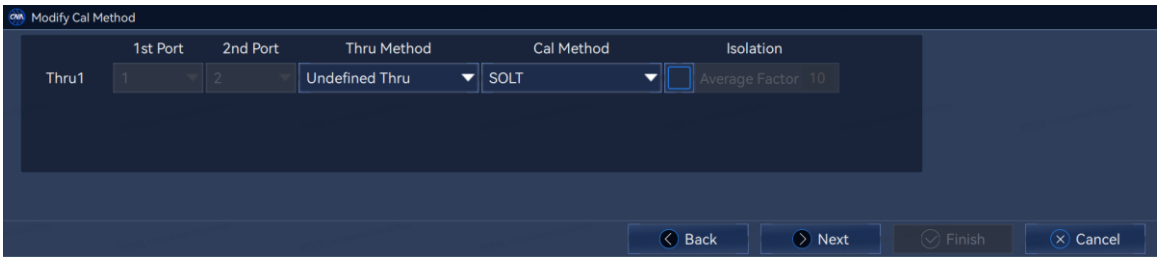


Figure 6.24 Modify the through connection method

Through calibration method: The through calibration method includes **[Zero Length Through], [Defined Through] and [Unknown Through**]. Select one of them by the drop-down box.

[Add Through or Remove Through]: After clicking **[Add Through],** a through connection method will be added. Click **[Remove Through]** to remove the unwanted through connection method.

3) Vector mixer calibration steps dialog box

According to the prompts, connect the open circuit, short circuit breaker, and load to port 1 and port 2 respectively to complete the measurement. Add a suitable adapter between port 1 and port 2 for through measurement. Connect open circuit, short circuit and load at the output end of the image filter to complete the measurement.



Figure 6.25 Vector mixer calibration dialog box

4) Save mixer characteristics dialog box

The vector network analyzer provides the function of saving calibration mixer characteristic data. After calibration is completed, the mixer characteristic calibration file can be saved to the designated path. When the same calibration mixer is used for vector mixer measurement

calibration, the saved S2P file can be directly called.

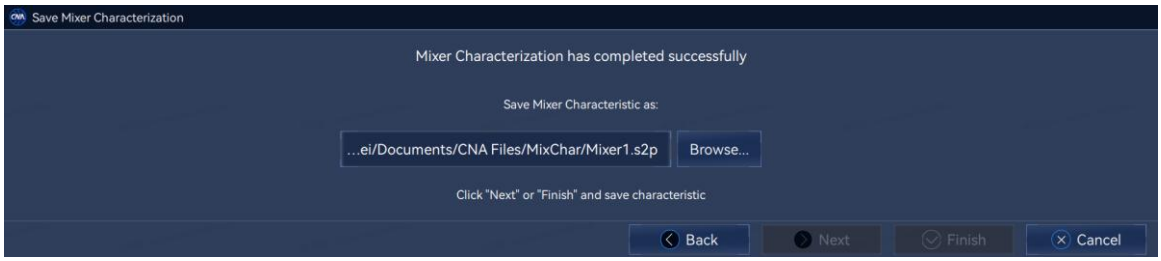


Figure 6.26 Save mixer characteristics dialog box

Browse: Find a path to store calibration mixer characteristics data, which can use a default name or a custom name.

Next step: After the calibration mixer characteristics are saved, executing the next step will complete the system calibration.

6.6 Embedded LO Mixer Measurement

A fixed LO is embedded inside the mixer under test, and the embedded LO characteristics allow for vector mixer measurement and scalar mixer measurement of the mixer.

But measuring these devices is challenging due to two reasons

Vector mixer measurement requires the use of a reference mixer, which has the same LO frequency as the DUT. An independent internal source or external source must be used as the LO of the reference mixer. At the same time, this LO should also be controlled by the instrument. When internal source 2 is used, the instrument measures faster.

1) All embedded LO measurements require the instrument receiver to be able to adjust to the exact frequency to measure the mixer output, which requires a very accurate LO frequency.

How to measure embedded LO

Enter the nominal frequency of the embedded LO in the Mixer Settings dialog box, which will serve as the starting point for measurement.

Before each measurement sweep of the DUT, start the background sweep to determine the frequency of the embedded LO and obtain a configurable accuracy.

Background sweep:

Broadband sweep: Preliminarily measure the embedded LO frequency, and selectively add a frequency near a selected data point. The input signal of the DUT can be selectively adjusted to a point frequency. The B receiver sweeps the selected frequency band near the preset output frequency. Obtain the difference between the desired output signal and the found frequency, and apply the obtained result as an adjustment.

Accurate sweep: The B receiver measures at the selected data points. Calculate the exact frequency offset until the allowable residual value or maximum repeatability is reached.

For vector mixer measurements, when the LO frequency is determined, the reference mixer automatically upgrades.

How to use the embedded LO to measure the DUT

1) Establish a vector mixer measurement or a scalar mixer measurement
2) In the Mixer Settings dialog box, enter the nominal frequency of the embedded LO as the LO frequency
3) Implement calibration
4) Start and complete the Embedded LO Model dialog box
5) Measure the vector mixer (frequency conversion amplifier):

a) LO source of reference mixer:

1) Internal source
2) An external source: must be controlled by the instrument and must use a 10MHz reference to lock the instrument

b) Precautions for calibration process

Calibration process - The reference mixer and calibration mixer share a common LO source. When an external source is used, an additional power distributor is required, as shown in the following figure:

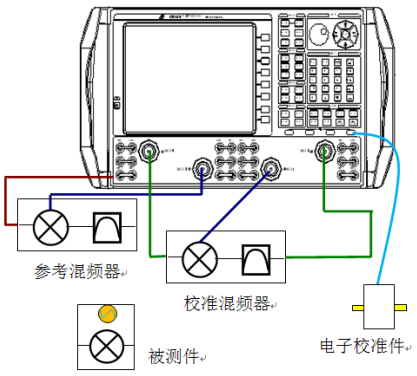


Figure 6.27 Embedded LO calibration

参考混频器	Reference mixer
被测件	DUT
校准混频器	Calibration mixer
电子校准件	Electronic calibration kit

During the measurement process, only the reference mixer uses the LO source. The LO port

used in the previous calibration of the mixer needs to be well matched, so that the LO port of the reference mixer does not change after calibration. as shown in the following figure. The above considerations are not required when internal source 2 is used.

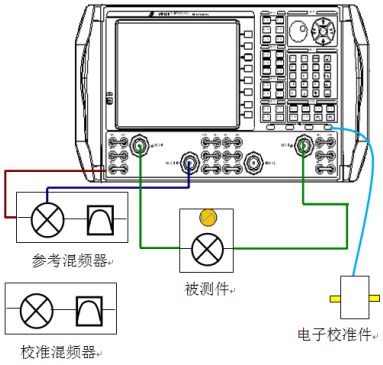


Figure 6.28 Embedded LO measurement

参考混频器	Reference mixer
被测件	DUT
校准混频器	Calibration mixer
电子校准件	Electronic calibration kit

For scalar mixer measurements, embedded LO measurements do not require specific settings.

c) How to start the Embedded LO Model dialog box

After establishing the mixer measurement, click [Measurement] → [Embedded LO] to enter the LO setting window



Figure 6.29 Embedded LO mode

d) Embedded LO Help dialog box

Open embedded LO mode: Click to enable embedded LO measurement

Set search frequency point: Select or specify the data point swept by the mixer, which is used to find the embedded LO frequency. If a marker has already been activated, this data point can be used. For broadband and accurate sweep, select a point with the lowest noise in the mixer sweep. It is usually the center of the sweep, if a filter is used to select the center of the filter.

LO frequency difference: Measure the absolute difference between the embedded LO frequency and the LO setting in the Mixer Settings dialog box. This value changes every time the local frequency is measured. This is a method of changing the frequency setting of the mixer and ensuring effective calibration.

Reset: Set the LO frequency to 0Hz

Start search: Use the current dialog settings to find and measure the actual LO frequency by the instrument. The data will be displayed in the search status box.

Set search criteria: These settings determine the ratio of time spent to the accuracy of local frequency measurement. You can see that when noise appears in the measurement trace, the accuracy becomes compromised.

Reset: All settings are returned to default values

Rough and accurate search: Perform all tuning processes for each background sweep

Accurate search only: No broadband tuning is performed for each sweep. Use this setting when the embedded LO is stable

Disable automatic search frequency: Simply apply the previously measured LO frequency difference to the reference mixer LO and instrument receiver

Search bandwidth: Reducing the sweep span limits the number of data points for broadband sweep measurement, making measurement faster

Maximum search times: The maximum number of sweep points for accurate sweep. When this point is reached, use the final measurement obtained

Error: When two consecutive precise measurements are within this range, the last measurement is used. If it is not obtained within this error value, use the last measurement

IF bandwidth during search: The intermediate frequency bandwidth used in broadband and accurate tuning sweep. The larger the IF bandwidth, the faster the sweep, but the signal may not be found

Number of sweeps per search interval: Set the interval time during which tuning is performed. ‘The number of sweeps per search interval is 3' refers to adding tuning sweeps before every three measurement sweeps. If the embedded LO drifts or if the DUT is frequently changed, use‘The number of sweeps per search interval is 1’

Search status: Allow LO measurement sweep to be displayed in text or icon form

Clear status: Remove the currently displayed text information

e) Connect the preamplifier

If the mixer measurement requires more source power than the input power provided by the instrument, a high-frequency preamplifier can be used to provide additional sources. The following describes how to use a high-frequency preamplifier to construct and perform a calibrated scalar mixer measurement.

Connect a high-frequency preamplifier between the source output and coupler connection ports on the front panel as shown in the following figure. The specific instrument diagram may not

look like this.

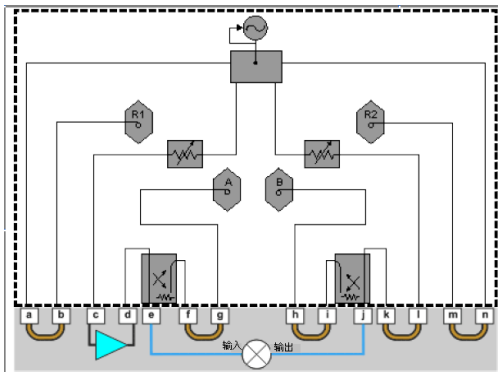


Figure 6.30 Schematic diagram of connecting the preamplifier

输入	Input
输出	Output

Measurement and calibration steps

In the following steps:

Test port power: Source output power level

Correction power: The power level required at the input and output ports of the mixer

The following program assumes that an stimulus source is applied to the mixer input port for S11 and SC21 measurements, and the mixer output port for S22 and SC12 measurements

1) Determine the gain of the high-frequency preamplifier. If the mixer spans the input and output ranges, there is a significant slope in the gain. Refer to a high-frequency preamplifier with gain ramp

2) Determine the corrected power of the mixer input and output ports.

3) Calculate the test port power of two ports by subtracting the gain of the amplifier from the input and output corrected power levels.

4) In the power dialog box of the instrument, remove the port power coupling column. It allows each port to have different power levels

5) Enter the calibration test port power for each port

6) Select the connection type and calibration kit of the DUT in the calibration wizard of the scalar mixer measurement, click VIEW/Change Source Calibration Settings to generate the Source Calibration Settings dialog box

7) At power deviation, input preamplifier gain

Scalar mixer measurement calibration is performed throughout the input and output range of the mixer. Therefore, the high-frequency preamplifier will be limited by the deviation of the entire input and output frequency range of the mixer. In order to compensate for the gain slope, it is necessary to add source attenuation settings, set power offset values, and initial power values to ensure that the calibration process does not cause uneven distribution of source power.

For example, suppose the gain of the preamplifier is 30dB at the low end and 20dB at the high end. If a 30dB power offset value is input, when the actual gain drops to 20 dB, it may exceed the

ALC range of the vector network analyzer. The vector network analyzer will try to increase its own source power to compensate for the loss of gain reduction. Therefore, add an offset power value in the middle of the amplifier gain bandwidth.

When conditions permit, an attenuator can be used to set the vector network analyzer ALC to operate in the middle of its power range, thereby obtaining appropriate gain correction power. This situation means that ALC can set the power high or low based on the gain ramp. However, when the gain of the preamplifier is too large and the power instability of the source cannot be solved by setting an attenuator, it is necessary to replace the preamplifier and use a more flat preamplifier.

Annex 7 Intermodulation Distortion Measurement

This chapter includes the following contents:

- Concept of Intermodulation Distortion.....514
- Intermodulation Distortion Parameters.....515
- Amplifier Sweep Frequency Intermodulation Distortion Measurement.....520
- Amplifiers Sweep Frequency Intermodulation Spectrum Measurement.....530

7.1 Concept of Intermodulation Distortion

When multiple signals of different frequencies are input to a device or system at the same time, the output port of the device under test will generate multiple signals of other frequencies due to its nonlinear characteristics. A typical measurement method is to input two single tone signals with different frequencies and same power at the input end of the DUT, so as to test the spectrum of their output signals. There are many measurement methods to determine the intermodulation distortion parameters of the DUT.

The intermodulation distortion generates multiple signals with predictable frequencies. The intermodulation frequencies generated by a dual frequency system are as follows:

$$f_{IM} = mf_1 \pm nf_2$$

Where m and n are natural numbers, and $m+n$ is the order of intermodulation distortion. Wherein, odd order products (such as 3rd, 5th, etc.) are very close to the frequency of the input dual tone signal, which has potential impact on adjacent communication channels. In addition to second-order products, higher even order products are usually far from the frequency of the dual tone input signal and are usually not considered.

Figure 7.1 shows the adjacent distortion products generated by two dual tone input signals with the same power and their intermodulation. From Table 7.1, it can be seen that the frequency of odd order products is relatively close to the frequency of the main signal. For most devices, the distortion product caused by intermodulation will be more obvious when the system generates compression artifact.

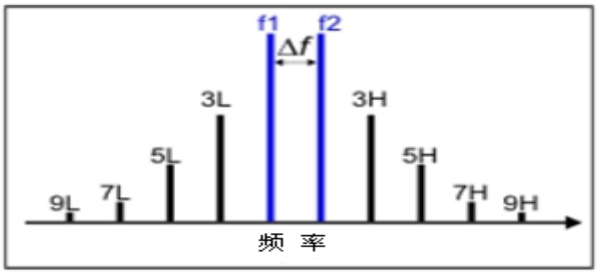


Figure 7.1 Dual tone signals (f1 and f2) and odd order intermodulation products

频率	Frequency

Table 7.1 Frequency product close to intermodulation distortion of main signal

Intermodulation product	Low frequency	High frequency
Main tone signal	**f1**	**f2**
	100MHz	120MHz
3rd order intermodulation	**2f1~f2**	**2f2~f1**
	80MHz	140MHz
5th order intermodulation	**3f1~2f2**	**3f2~2f1**
	60MHz	160MHz
7th order intermodulation	**4f1~3f2**	**4f2~3f1**
	40MHz	180MHz
9th order intermodulation	**5f1~4f2**	**5f2~4f1**
	20MHz	200MHz
2nd order intermodulation	**f2~f1**	**f1+f2**
	20MHz	220MHz

7.2 Intermodulation Distortion Parameters

This section will mainly introduce several basic parameters that can be used for amplifier and mixer measurement. By changing the intermodulation scale, intermodulation position (high scale, low scale, average value), and intermodulation measurement port, the basic parameters can be extended to over 150 intermodulation parameters.

- Basic Parameters.....515
- Intermodulation Distortion Application 519

7.2.1 Basic Parameters

7.2.1.1 Tone Power Parameters

The tone power parameters are used to measure the absolute power of the main tone signal, odd order products (up to 9 times), and second-order products. Both the input and output ends of the DUT can be used for tone power measurement. Due to the fact that intermodulation tone signals appear in pairs, it is possible to measure the high, low, or average values of the tone signal.

The formula for calculating the average tone power is as follows:

$$Avg = \frac{Hightone(dBm) + Lowtone(dBm)}{2}$$

Note that when measuring the second-order product, if the frequency difference of the dual tone signal is less than 10MHz, the low scale (f2-f1) of the second-order product will exceed the lower frequency limit of the analyzer. Therefore, in this case, the low scale of the second-order product cannot be measured.

7.2.1.2 Tone Gain Parameters

The tone gain (in dB) is the ratio of the output tone power to the input tone power.

Due to the pairwise nature of tone signals, it is possible to choose a low scale, high scale, or the average value of the two when the audio gain is calculated.

It should be noted that when intermodulation of frequency converter is measured, the audio frequency of input end and output end is usually different.

7.2.1.3 Intermodulation Distortion Parameters

The intermodulation distortion parameters are used to describe the power difference between the distortion product and the main tone signal, and is calculated according to the voice power. The intermodulation distortion parameters can also be used to measure the 2nd order and 3rd-5th order intermodulation products. The input or output port can be selected as measurement port, as well as low scale, low scale, or average value may be selected for measurement. The calculation formula of intermodulation distortion parameters is as follows:

IMxLo = PwrxLo-PwrMainLo

IMxHi = PwrxHi-PwrMainHi

IMx = Pwrx-PwrMain

IMxLoIn = PwrxLoIn-PwrMainLoIn

IMxHiIn = PwrxHiIn-PwrMainHiIn

IMxIn = PwrxIn-PwrMainIn

Where:

·X is the order of intermodulation distortion (2, 3, 5, 7, 9)

·The suffix Hi represents the high scale, Lo represents the low scale, and none represents the average value

·The suffix In represents the measurement input port, and without it, it represents the output port

7.2.1.4 Intersection Point Parameters

As shown in Figure 7.2, the black and green curves represent the main tone signal power and the intermodulation tone output power, respectively. From the figure, it can be seen that as the main tone signal output power increases, the intermodulation tone output power increases with a predictable and steeper slope. At a point, the main tone output power will be equal to the intermodulation tone output power, which is usually defined as the intersection point. The intersection point cannot be directly measured by a tester, but needs to be calculated by measuring the main tone power and intermodulation tone power.

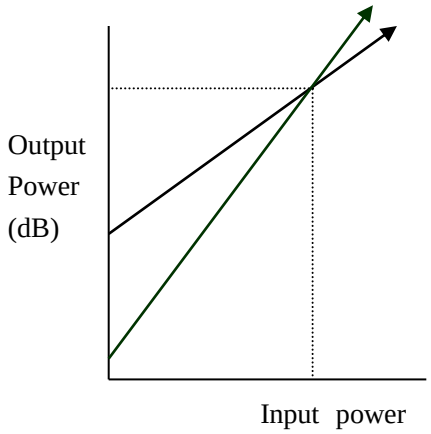


Figure 7.2 Schematic diagram of intersection point parameters

The intermodulation distortion measurement function can calculate and display the theoretical intersection point power values at the input and output ends of the DUT, namely, the input intersection point (IIP) and the output intersection point (OIP).

Intersection point measurement can also be set with the measurement scales: 2, 3, 5, 7, and 9 scales; the measurement position: high scale, low scale and average value; and the measurement port: input and output. For second-order intersection point, only the low or high scales can be measured, and the average value is not supported.

The calculation formula of intermodulation distortion intersection point parameters is as follows:

OIPxHi = PwrMain - IMxHi/(x-1)
OIPxHiIn = PwrMain - IMxHiIn/(x-1)
OIPxLo = PwrMain - IMxLo/(x-1)
OIPxLoIn = PwrMain - IMxLoIn/(x-1)
OIPx = PwrMain - IMx/(x-1)
OIPxIn = PwrMain - IMxIn/(x-1)
IIPxHi = PwrMainIn - IMxHi/(x-1)
IIPxHiIn = PwrMainIn - IMxHiIn/(x-1)
IIPxLo = PwrMainIn - IMxLo/(x-1)
IIPxLoIn = PwrMainIn - IMxLoIn/(x-1)
IIPx = PwrMainIn - IMx/(x-1)
IIPxIn = PwrMainIn - IMxIn/(x-1)

Where:

·OIP represents the output intersection point

·IIP represents the input intersection point

·x=scale number (2, 3, 5, 7, 9)

·PwrMain=The main tone signal power at the input or output end of the DUT

·IMx=main tone signal power – intermodulation tone power (from intermodulation distortion parameters)

7.2.1.5 Composite Triple Beat (CTB)

According to the NCTA (National Cable Television Association) standard, the composite triple beat is defined as the modulation beat of the target channel signal due to the triple beat generated by the nonlinear characteristics of the DUT. The composite triple beat represents the ratio of the signal level of the target channel to the maximum average level of the triple beat component around the carrier frequency of the target channel.

Intermodulation distortion supports two parameter types of composite triple beat:

·CTB: Approximate value based on the number of beats in the middle of the band

·CTBE: Approximate value based on the number of beats at the edge of the band

The calculation formula is as follows:

CTB(dB)=-2(Pi-Ps)+6+10Log(3N2/8)+CTB OFFset

CTBE(dB)=-2(Pi-Ps)+6+10Log(N2/4)+CTB OFFset

Where:

·Pi=power value of the third-order intersection point at the output end (dBm): OIP3 (Lo | Hi)

·Ps=Selection based on composite normalization mode:

  ·PDBM or PDBMV mode: Ps = CompositeNormalizedCTBPower (composite normalized CTB power)

  ·Number of Carriers mode, Ps = PwrMain (AVG) – 10Log(N/2)

  ·None mode, Ps = PwrMain (AVG)

·CTB OFFset=Offset value calculated by CTB

·N=number of carriers

Note: The values of CTB OFFset and N can only be set using SCPI or COM commands.

7.2.1.6 Composite Second Order Beat (CSO)

According to the NCTA standard, the composite second order beat is defined as the modulation beat of the target channel signal due to the second order generated by the nonlinear characteristics of the DUT. It represents the ratio of the target channel signal level to the maximum average level of the beat component within the range of 0.75 MHz and 1.25 MHz of the target channel carrier frequency.

Intermodulation distortion supports CSO parameter test, and the calculation formula is as follows:

CSO(dB)=(Pi-Ps)+10Log(N)+CSO OFFset

Where:

·Pi=power value of the second-order intermodulation at the output end: OIP (Lo | Hi)

·Ps=Based on one of the following composite normalization pattern types:

  ·PDBM or PDBMV mode, Ps = CompositeNormalizedCSOPower (composite normalized CSO power)

  ·Number of Carriers mode, Ps = PwrMain (AVG) – 10Log(N/2)

  ·None mode, Ps = PwrMain (AVG)

·CTB OFFset=Offset value calculated by CSO

·N=number of distortion products

Note: The values of CSO OFFset and N can only be set using SCPI or COM commands.

7.2.1.7 Intermodulation Distortion

According to the NCTA standard, the signal from the parasitic channel is modulated to the target channel, which can cause cross modulation distortion due to the nonlinear characteristics of the DUT. The measurement of intermodulation distortion is expressed as the power ratio of the self modulated signal to the parasitic interference modulated signal in the target channel.

Intermodulation distortion supports XMOD parameter measurement, measured as follows:

XMOD=-2(Pi-Ps)+6dB+20Log(N)

Where:

·Pi=power of the third-order intersection point at the output end: OIP3 (Lo | Hi)

·Ps=carrier power: PwrMain (AVG)

·N=number of carriers

7.2.2 Intermodulation Distortion Application

The intermodulation distortion function requires the analyzer to generate two main tone signals internally. Figure 7.3 shows the schematic diagram of the four port vector network analyzer with dual sources and internal combiner.

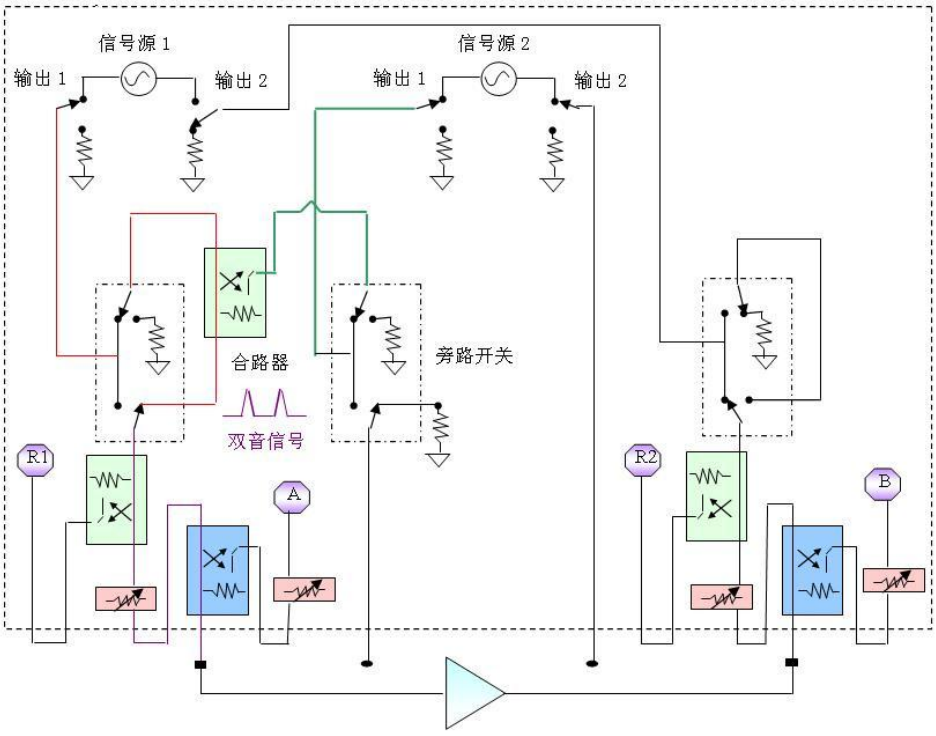


Figure 7.3 Internal configuration of intermodulation distortion

输出 1	Output 1
信号源 1	Signal source 1
输出 2	Output 2
输出 1	Output 1
信号源 2	Signal source 2
输出 2	Output 2
合路器	Combiner
双音信号	Dual tone signal
旁路开关	Bypass switch

According to the parameters to be measured and the sweep type, the intermodulation distortion program automatically adjusts and sets the source and receiver frequencies of the analyzer to cover all the main tone signal and intermodulation tone signal power required for measurement. For example, when the third-order intermodulation parameters are measured, it is necessary to measure the dual tone signal power and the high scale and low scale power of the third-order intermodulation tones.

7.2.2.1 Stimulus Setting Restrictions and Out-of-Range Intermodulation Tones

The intermodulation distortion function generates two dual tone signals from the network analyzer, so the dual tone signal frequency is required to be within the analyzer frequency range. To ensure that the dual tone f1 and f2 frequencies do not exceed the upper and lower limits of the analyzer frequency, the program will automatically adjust the parameter settings if necessary.

However, the network analyzer allows the setting of intermodulation products lower than the frequency range of the network analyzer. For example, the dual tone signal frequency is 10MHz and 15MHz. Although the network analyzer allows the measurement of parameter IM3Lo (third-order low scale product), the analyzer measures the lowest frequency (10MHz).

7.2.2.2 Measurement Point Limit

A maximum of 32,001 points can be measured in one sweep, but depending on the measurement parameters, the analyzer may need to perform multiple background measurements. Therefore, when there are many points, the analyzer may need a long time for background measurements to start refreshing the data.

7.3 Amplifier Sweep Frequency Intermodulation Distortion Measurement

This section includes the following contents:

- Characteristics and Limitations.....521
- Intermodulation Distortion Measurement Interface Calling.....521
- Intermodulation Distortion Measurement Parameter Setting.....526

- Simple Intermodulation Calibration.....528

7.3.1 Characteristics and Limitations

7.3.1.1 Characteristics

·The measurement setting of distortion parameters is fast and simple, and up to 9th order products can be measured;

·Both the input and output of the DUT can be measured;

·Support multiple dual tone sweep modes: Linear and list sweep frequency fc (dual tone center frequency) mode, sweep pitch mode, power sweep, and point frequency (fixed dual tone frequency) mode;

·The network analyzer can provide high power, high linearity, and low harmonic signal sources to achieve fast and accurate measurement;

·Support calibration and correction;

·The intermediate frequency bandwidth when measuring the main tone signal and the intermodulation tone signal can be independently set.

7.3.1.2 Limitations

·The two ports do not support frequency conversion testing, and the measurement of frequency conversion devices will be introduced in the future;

·There are restrictions on the port connection method between the DUT and the vector network analyzer. The port connection method can be changed in the [Power] setting list.

7.3.2 Intermodulation Distortion Measurement Interface Calling

Menu path: [Measurement] → [Measurement Class] → [Intermodulation Distortion] → [OK] → [Intermodulation Setting].

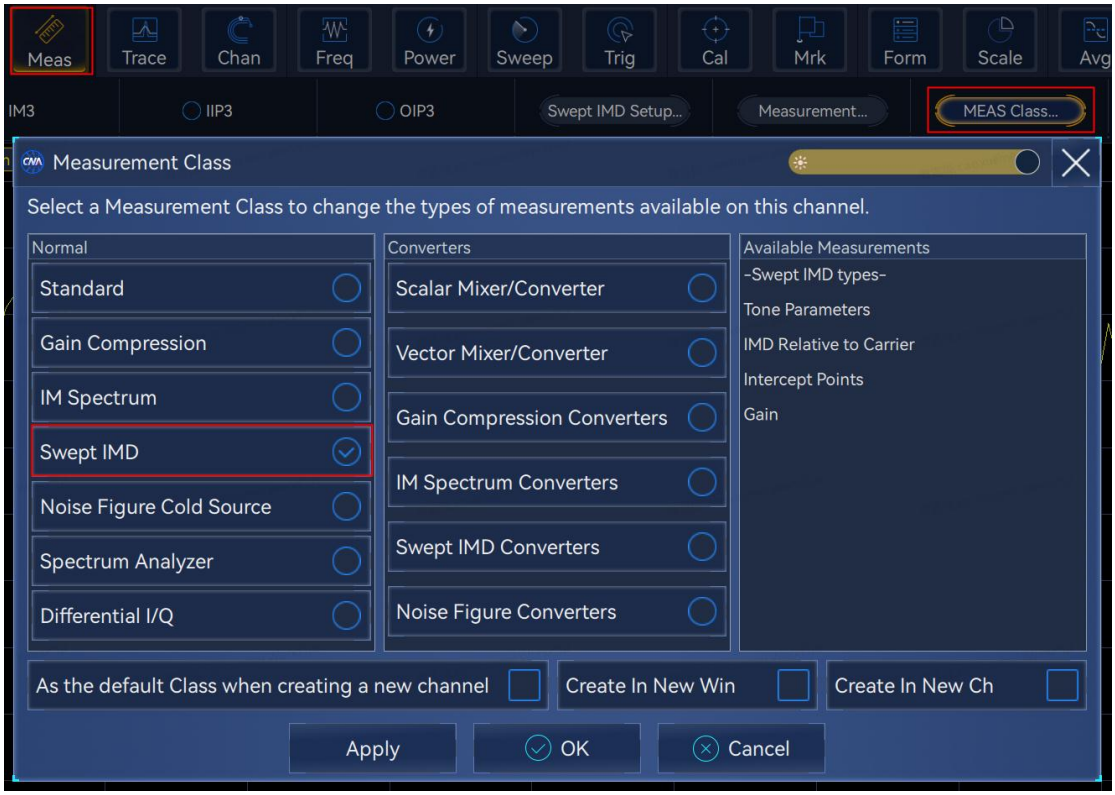


Figure 7.4 Enter Sweep Frequency Intermodulation Settings

7.3.2.1 Sweep Frequency Intermodulation Settings

1) Frequency list

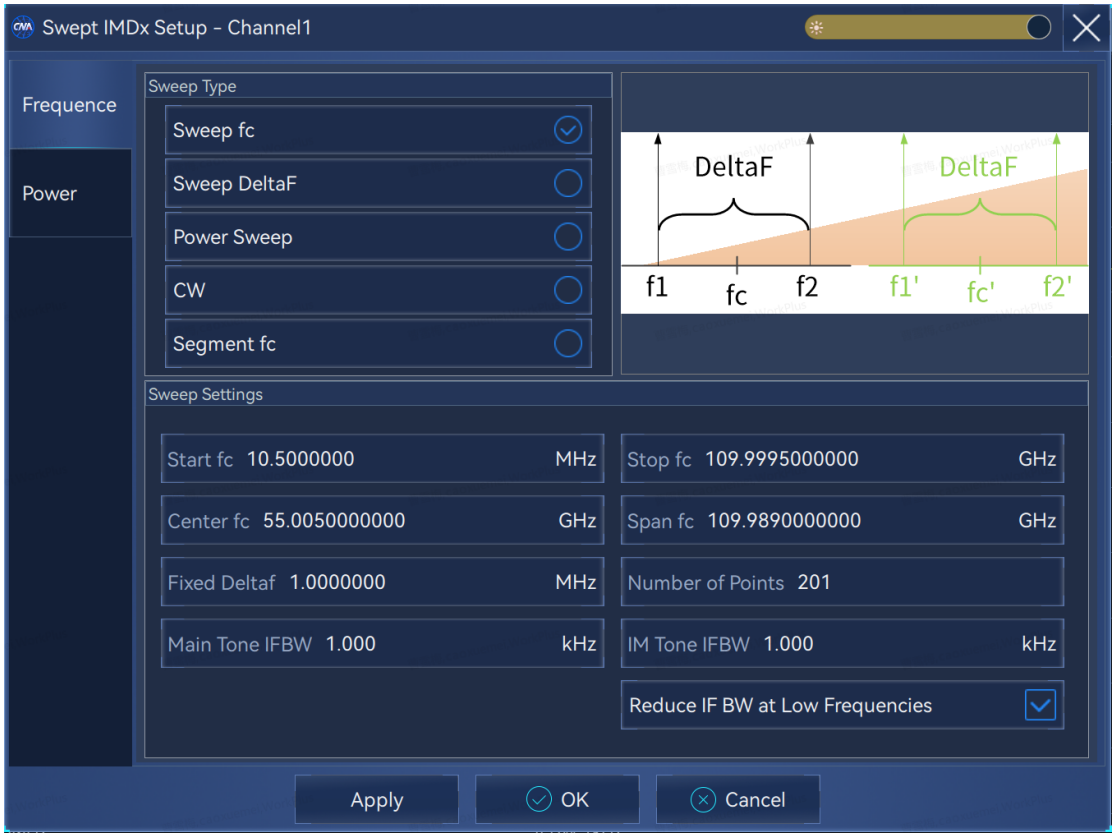
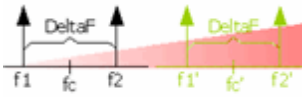


Figure 7.5 Intermodulation Distortion Frequency Settings dialog box

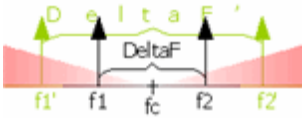
·Linear sweep fc (center frequency)



The pitch and power of the dual tone signal are fixed. Sweep the center frequency (fc) of the dual tone signal, set the starting and ending values of the signal center frequency, and also specify the center frequency point and sweep width of fc

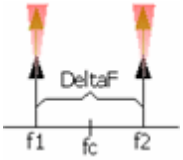
For each center frequency fc, the analyzer adjusts the receiver to measure all the required tone power.

·Linear sweep DeltaF (pitch)



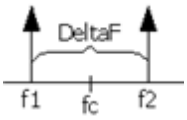
The center frequency and power of the dual tone signal are fixed, and the frequency difference (pitch) of the dual tone signal is swept from the starting difference to the ending difference.

·Power sweep



The frequency of the main tone signal is fixed (can be set through f1 and f2 or fc and DeltaF), and the power of the main tone signal is swept from the starting power to the ending power. The power settings of f1 and f2 can be set independently (in the [Power] list) or coupled.

·Point Ferq



The frequency and power of the main tone signal are fixed. The analyzer measures data for a specified number of points, with the X-axis representing the number of points.

·IFBW

The intermediate frequency bandwidth for the measurement of the main tone signal and the intermodulation tone signal can be specified separately. The main tone signal usually has a higher power, so a wider intermediate frequency bandwidth can be utilized for fast measurement; However, the power of intermodulation tone signals is often relatively low, and small bandwidth can be used for accurate measurement. [Reduce mid frequency bandwidth in low frequency range]: When the frequency is below 748MHz, the noise of the network analyzer becomes relatively poor, especially between 10MHz and 45MHz. When this option is selected, the analyzer will use a lower intermediate frequency bandwidth than the bandwidth set on the interface for measuring frequencies below 748MHz.

2) Power list



Figure 7.6 Intermodulation Distortion Power Settings dialog box

It is used to set main tone power during intermodulation distortion measurement RF source power and power sweep.

·**Power ON/off (all channels)**: Turns on or off the RF source power for all channels.

·**DUT input end**

·**Input Port**: Select Port 1 or Port 3, wherein Port 3 requires an external combiner.

·**Source attenuation**: Set the power attenuation value of Port 1, which will affect the signal power range input to the DUT.

·**Receiver attenuation**: Used to protect receiver A from burning out.

·**DUT output end**

·**Output Port**: Select Port 2 or Port 4.

·**Source attenuation**: Improve the load matching at the output end of the DUT. When the output signal power reaches 10dBm, the attenuator can be selected at 0dB. For every 10dBm increase in output signal power, the attenuation value should correspondingly increase by 10dB.

·**Receiver attenuation**: Set the receiver attenuation for Port 2. When the power input to the receiver test port is around 10dBm, the network analyzer receiver may generate compression. However, increasing receiver attenuation may result in lower power during analyzer measurement, resulting in

higher measurement noise. In this case, reducing the intermediate frequency bandwidth of intermodulation tone can reduce the noise impact of the intermodulation signal.

·**Tone power setting**

·**Coupling dual tone power**: Configure the power of the dual tone signal by setting f1, that is, f2 and f1 have the same power.

·**f1 Power**: Set the power of the input or output signal f1 of the DUT, with a power value range of -25dBm to 20dBm. When selecting coupling power, the power of f1 and f2 are set simultaneously.

·**f2 Power**: When coupling power is not selected,- 25dBm power of the input or output signal f2 of the DUT can be set, with a power value range of -25dBm to 20dBm.

·**Start power, end power, f1 step power, f2 step power**: When selecting the power sweep type, the start power and end power of f1 and f2 can be set, or both signals can be set simultaneously through coupling power.

7.3.3 Intermodulation Distortion Measurement Parameter Setting

7.3.3.1 Adding Intermodulation Distortion Parameters

Menu path: [Trace] → [New Trace].

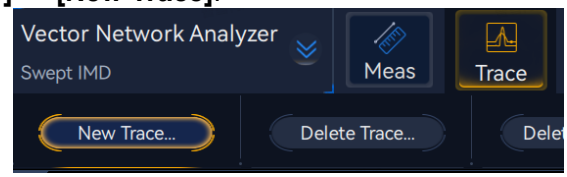


Figure 7.7 New intermodulation distortion parameter trace

A new trace for intermodulation setting



Figure 7.8 Intermodulation Distortion Measurement Parameter Setting

Use the dialog box shown in Figure 7.8 to set intermodulation distortion parameters. You can select up to 5 parameters at a time and click Apply to generate a trace. Without using the dialog box, more traces can be selected, and the number of traces and windows that can be added needs to meet the limitations of the analyzer.

7.3.3.2 Intermodulation Distortion Parameter Setting

1) Type

·**Tone power**: Measure the absolute power of a specified scale signal;

· **IMD relative power**: Measure the relative values of intermodulation tone and main tone signal f1 and f2 power;

·**IIP power**: Calculate the power value of the cut-off point at the input end of the DUT in theory, that is, the theoretical power value when the intermodulation power of the order to be calculated is equal to the main tone power.

·**OIP power**: Calculate the power value of the cut-off point at the output end of the DUT in theory, that is, the theoretical power value when the intermodulation power of the order to be calculated is equal to the main tone power.

·**CTBE**: Based on edge band composite third order beat distortion - only third order and output end can be selected

·**CTB**: Based on intermediate band composite third order beat distortion - only third order and output end can be selected;

·**CSO**: Composite quadratic beat distortion - only second-order and output end can be selected;

·**XMOD**: Output third-order intermodulation distortion;

·**Tone gain**: Measure the power of the tone signal and calculate the power ratio of the output signal to the input signal. The input and output signals have different frequencies;

2) Tone signal selection

Select the tone signal to be measured and displayed

·**High scale**: Measure and display the intermodulation signal above the main tone frequency;

·**Low scale**: Measure and display the intermodulation signal below the main tone frequency;

·**Average value**: Measure and display the average value of the intermodulation tone signal between the high and low ends of the tonic signal.

·**Maximum value**: Measure and display the maximum value of the intermodulation tone signal between the high and low ends of the main tone signal.

·**Minimum value**: Measure and display the minimum value of the intermodulation tone signal between the high and low ends of the main tone signal.

Notice

Notes on second-order intermodulation

·The second-order intermodulation tone is likely to be located outside the frequency range of the network analyzer, where the measured values are the upper and lower limits of the analyzer's frequency;

·When the high-frequency signal or low-frequency signal is outside the frequency range of the network analyzer, the average value measurement cannot be carried out;

·The average value of second order intermodulation tone cannot be measured because there is a significant difference in the high and low intermodulation audio rates of second order tones;

·If you need to calibrate second-order tones, you need to select the option‘Include second-order intermodulation tone' in the calibration dialog box.

3) Order and measurement port selection

·During the measurement process, you can select the order of the intermodulation product to be measured in the dialog box, including 1, 2, 3, 5, 7, and 9. Selecting order 1 means measuring the main tone signal, with a Main identifier attached after the parameter name. Otherwise, the order is added after the parameter name.

·Measurement can be carried out at the input or output end. When the input end is selected, an In identifier should be added after the parameter name, such as (PwrMainIn); When the output end is selected, there will be no additional identification after the parameter name, such as (PwrMain). A reference receiver is used for testing at the input end, and a measurement receiver is used for testing at the output end.

7.3.4 Simple Intermodulation Calibration

7.3.4.1 General

1. Simple Intermodulation calibration uses the source power calibration and receiver calibration interface for calibration;

2. Option to calibrate intermodulation scales is available;

3. When calibrating, connect port 1 to the power meter, calibrate the reference receiver and measurement receiver;

·The vector network analyzer can calculate a list of source and receiver frequencies, including the main signal and all intermodulation signals;

·Port 1 is connected to the power meter, and the source and receiver calibration can be used to calibrate the reference receiver R1;

·Reference receiver R1 is used to perform calibration of the main

signal source, first calibrating the low-frequency power of source 1, and then calibrating the high-frequency power of source 2. When measuring source 1 and source 2, it is necessary to disconnect them to ensure impedance matching;

·Pass through ports 1 and 2 to correct receiver B through receiver calibration.

Notice

Changing the frequency of the main tone signal will result in calibration being unavailable

If the frequency of the main tone signal changes, calibration will not be available. State changes may occur at the start/end/center frequency, frequency span, number of sweep points, or sweep type;

7.3.4.2 Selecting Product Signals

·**Maximum calibration scale**: Select the maximum order intermodulation product that can be measured. Low frequency and high frequency signals, as well as lower odd intermodulation products, are calibrated. For example, when measuring 5th order intermodulation, the main signal, high-order and low-order products such as 3rd and 5th order products are calibrated;

·**Including 2nd order**: Calibrate second-order products, where the frequency of these intermodulation products typically deviates significantly from the main signal frequency.

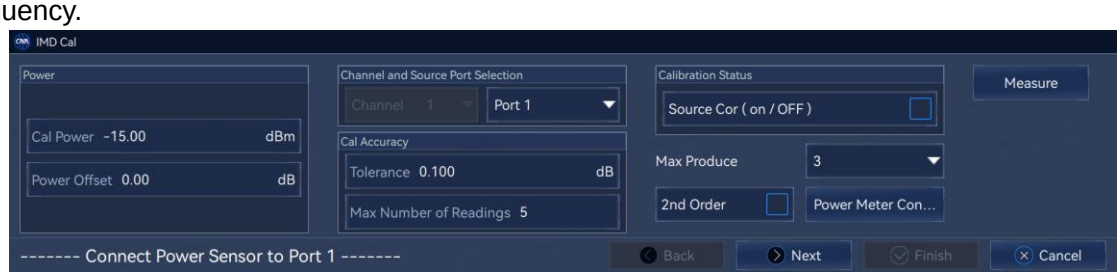


Figure 7.9 Intermodulation calibration process

·**Power Level**: Execute the power calibration value for Port 1. Given that the calibration standard of the power meter is 0dBm, the source power calibration power value is usually also chosen as 0dBm. If a component is added between the source end of the network analyzer and the calibration reference plane, the calibration power value should be adjusted so that it is still 0dBm at the calibration port of the power meter.

The remaining calibration steps page is the same as the source calibration page and receiver calibration page, and will not be introduced one by one here.

7.3.4.3 Sweep Frequency Intermodulation Distortion Power Data Storage

In the sweep frequency intermodulation distortion measurement data, only the logarithmic power amplitude can be saved as a *. csv file. This file can be read by the spreadsheet program Microsoft Excel, and the last swept data saved in the specified *. csv file.

Menu path: [System] → [File] → [Save Data As], then click and **select the extension as. csv**.

Notice

Data storage considerations

·Save the power values of the current established parameters, including input and output power values. If it is an average, maximum, or minimum type parameter, store all high and low scales: UT Avg | OUT Lo | OUT Hi | IN Avg | IN Lo | IN Hi;

·All displayed intermodulation products will be saved;

·The data is only saved as the power value of intermodulation products, and the calculated ratio is not saved;

·If calibration is turned on when saving data, all data will be calibrated and the original data will be saved.

7.4 Amplifier Sweep Frequency Intermodulation Spectrum Measurement

This section will focus on the intermodulation spectrum measurement, which can be performed simultaneously with the sweep frequency intermodulation distortion measurement. Due to the significant differences between these two measurements, the intermodulation spectrum measurement is usually conducted in separate channels. This section introduces the intermodulation spectrum measurement through the following sections:

- Concept of Intermodulation Spectrum.....530
- Creating Intermodulation Spectrum Channel.....532
- Intermodulation Spectrum Trace Parameters.....532
- Intermodulation Spectrum Settings Dialog Box.....533

7.4.1 Concept of Intermodulation Spectrum

The intermodulation spectrum channel of the vector network analyzer provides a traditional spectrum analyzer interface to measure the intermodulation distortion of the DUT. The interface of the intermodulation spectrum channel is different from that of the sweep intermodulation channel. Because the main signals f1 and f2 are fixed during the intermodulation spectrum measurement, the receiver receives the output signal of the DUT through the sweep mode, and its display interface is shown in Figure 7.10.

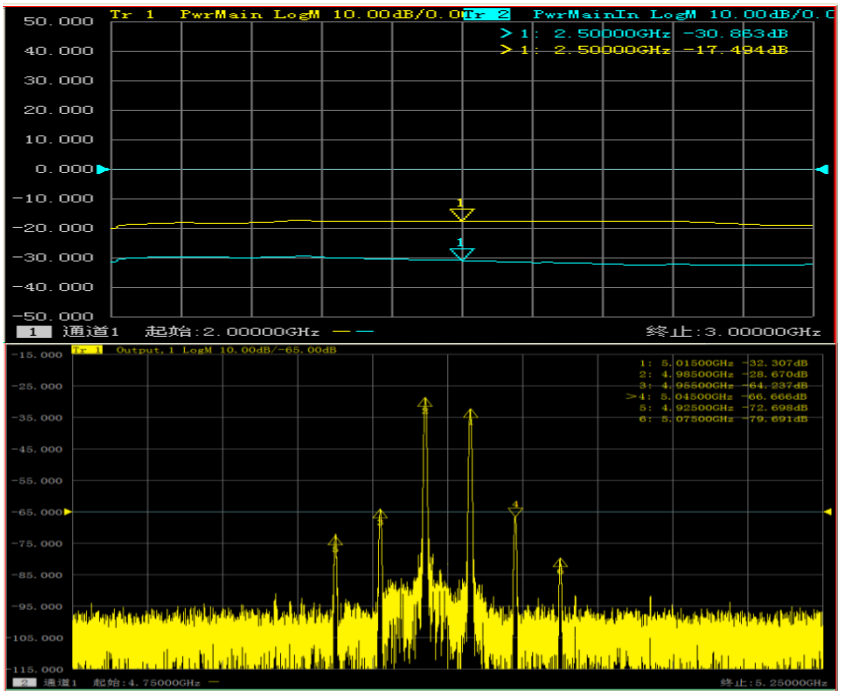


Figure 7.10 Intermodulation spectrum trace (below) Sweep frequency intermodulation distortion trace (above)

Figure 7.10 (above) shows the intermodulation distortion measurement window, where the sweep range is 2GHz-3GHz, the tone spacing is fixed, and the two-tone center frequency is 2.5GHz;

Figure 7.10 (below) shows the intermodulation spectrum measurement window, which is mainly used to measure the main tone signal and the third, fifth, and seventh order intermodulation products.

The vector network analyzer differs from a typical spectrum analyzer in that the vector network has an internal signal source, while the spectrum analyzer does not. Therefore, the vector network can generate the main signals f1 and f2 at different frequencies by setting the signal source in the operating mode of either the intermodulation spectrum channel or the sweep frequency intermodulation distortion channel. Among them, the settings mainly include the frequency and power of the main signal.

Receiver settings

The intermodulation spectrum receiver can be set to operate only on the intermodulation spectrum channel. The settings include how many orders of intermodulation products (which determine the frequency range and center frequency) and bandwidth are displayed.

The number of measurement channels is changed by selecting multiple different bandwidths. The wider the bandwidth, the fewer points. The corresponding relationship is as follows:

N = (Span/ResBW) * 3

The intermodulation spectrum channel performs multiple measurements on each data point, filtering out unwanted mirror interference (generated internally by the vector

network analyzer). It provides a highly pure signal acquisition path and ensures that there is no false response during the measurement process.

7.4.2 Creating Intermodulation Spectrum Channel

The intermodulation spectrum channel can be created independently or together with the existing and set sweep frequency intermodulation measurement channel.

Create an independent spectrum measurement channel

1. Press the auxiliary button of the network analyzer, click **[Measurement]**, and then click **[Measurement Category]**;

2. Select the **Intermodulation Spectrum** and click **OK** (delete existing measurement channels), or create a new channel (create a new measurement channel)

7.4.3 Intermodulation Spectrum Trace Parameters

Add intermodulation spectrum trace

Menu path: [Trace] → [New Trace].

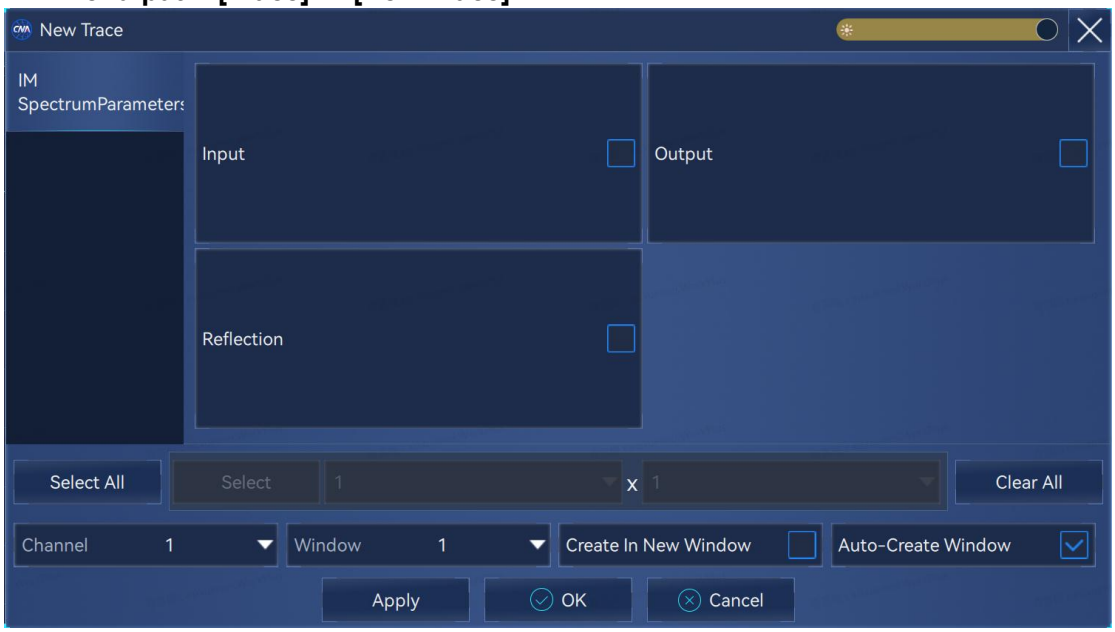


Figure 7.11 Intermodulation Spectrum Trace Parameter Selection dialog box

Output spectrum: Observe the output signal of the DUT (receiver B);

Input spectrum: Observe the input signal of the DUT (receiver R1), use this function to measure the frequency of intermodulation products, and then determine the power value of the false signal entering the DUT;

Reflection spectrum: Observe the reflection signal at the input end of the DUT, and measure it through port 1 of the network analyzer (receiver A).

7.4.4 Intermodulation Spectrum Settings dialog box

How to open the Intermodulation Spectrum Settings dialog box, the vector network analyzer provides a quick way to access it.

1. Click on **Measure**;
2. Then click on **Intermodulation Spectrum Settings**;

·Frequency bar

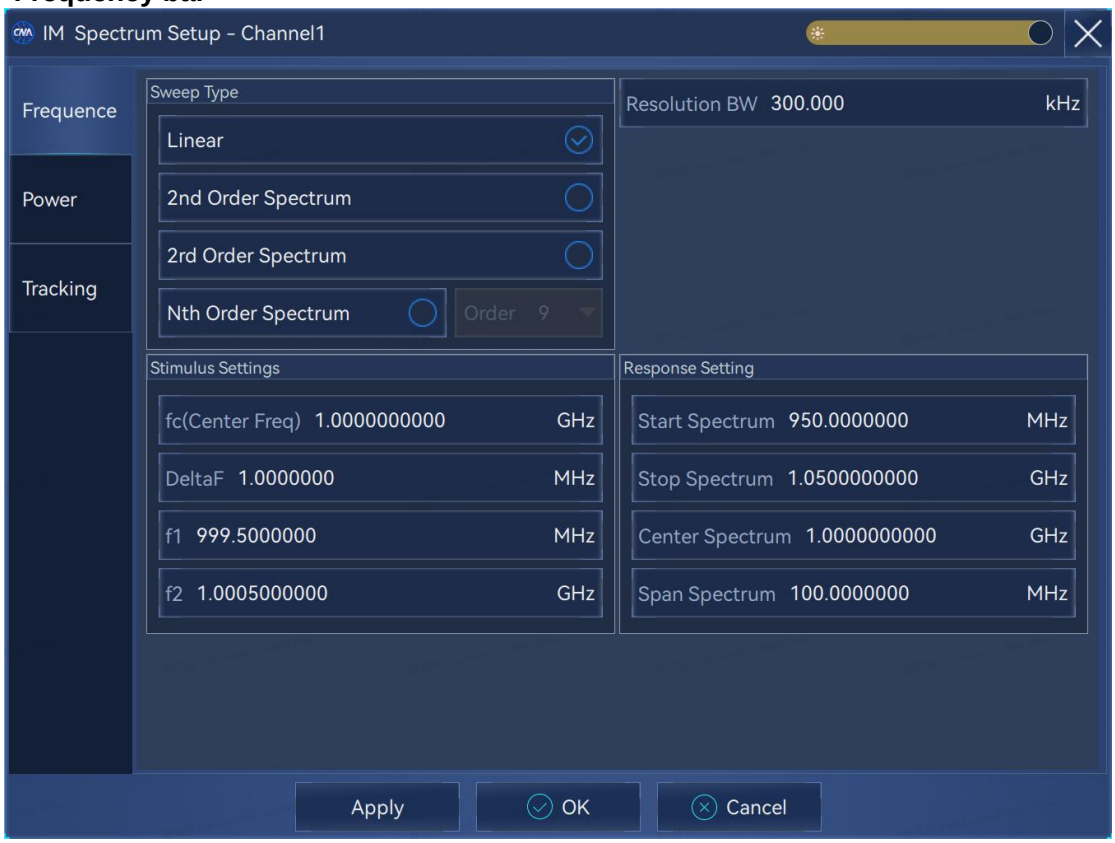


Figure 7.12 Spectrum Settings Frequency Bar menu

Notice

Intermodulation spectrum channel points = 3*spectrum span/resolution bandwidth, not directly transformable.

1. **Sweep Type:** The intermodulation spectrum receiver provides several methods to display the spectral power of any frequency range and different order distortion products.

   ·**Linear**: Under the conditions of set values f1 and f2, the start/end frequency, center frequency, and span of the receiver can be arbitrarily set;

   · **2nd order spectrum:** For the convenience of observing high-end 2nd order intermodulation and surrounding spectrum, in this mode, the receiver frequency range is set to be coupled to the stimulus values of f1 and f2, with center frequency=f1+f2

and span=3 * (f2-f1);

· **3rd order spectrum:** For the convenience of observing 3rd order intermodulation and surrounding spectrum, in this mode, the receiver frequency range is set to be coupled to the stimulus values of f1 and f2, with center frequency=(f1+f2)/2 and span=4 * (f2-f1);

· **N-th order spectrum**: For the convenience of observing N-th order main products, in this mode, the receiver frequency range is set to be coupled to the stimulus values f1 and f2, with center frequency=(f1+f2)/2 and span=N * (f2-f1);

2. **Resolution Bandwidth:** Gaussian filter is used in the intermodulation spectrum channel instead of the standardized network instrument IF filter, so as to provide functions similar to the traditional spectrum analyzer. The provided bandwidth types are: 3 MHz, 1 MHz, 600 kHz, 300 kHz, 150 kHz, 100 kHz, 60 kHz.

3. **Stimulus Setting**: When tracking is turned off, the frequency of the main tone signal can be set through this column.

·**fc (center frequency)**=(f1+f2)/2;

·**Delta F (frequency difference)**=(f2 – f1);

·**f1**=Low frequency main tone signal frequency;

·**f2**=High frequency main tone signal frequency

4. **Response setting**: In linear sweep mode, the sweep range of the receiver can be set through two methods: start/end frequency, and center frequency/span.

·**Spectrum start**=the first frequency point of power spectrum sweep;

·**Spectrum end**=the last frequency point of power spectrum sweep;

·**Spectrum center**=(spectrum start+spectrum end)/2;

·**Spectrum span**=(spectrum end - spectrum start)

·**Power bar**



Figure 7.13 Spectrum Settings Power Bar menu

·**Tracking bar**

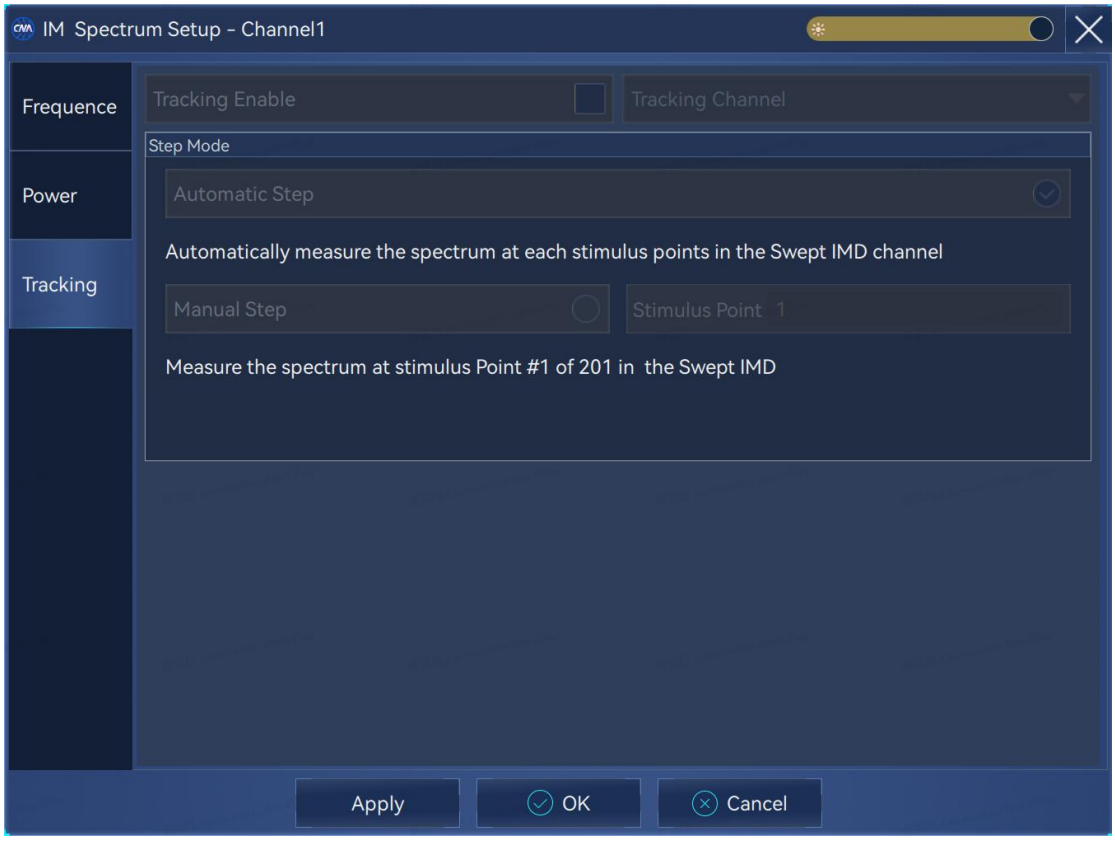


Figure 7.14 Spectrum Settings Tracking Bar menu

This dialog box is used when there are intermodulation measurement channels, that is, two channels coexist.

1. **Tracking Enable**: If this option is selected, the frequency setting of the stimulus will not be effective, and all stimulus settings, such as frequency, power, attenuator and calibration, will copy the settings of the sweep intermodulation distortion channel to the intermodulation spectrum channel.

2. **Step Mode**: When Tracking Enable is selected, intermodulation spectrum measures the way to track intermodulation channels.

·**Automatic Step**: The intermodulation spectrum measurement automatically tracks the current data point frequency of the IMD channel for the dual tone signal frequency. After completing a spectrum measurement, the dual tone signal frequency is automatically set to the frequency value of the next data point in the IMD channel.

·**Manual Step**: Select this option to align the intermodulation spectrum measurement dual-tone frequency with the frequency of the specified data point (number of points can be specified by the user) in the IMD channel.

Annex 8 Noise Figure Measurement

This chapter includes the following contents:

- Overview of Noise Figure Measurement.....514
- Instructions for Using the Noise Figure Option.....515
- Using the Noise Figure Option Application.....520
- Converter Noise Figure Measurement.....530

8.1 Overview of Noise Figure Measurement

8.1.1 Basic Concept of Noise Figure

The noise performance of microwave electronic systems that need to receive signals requires special attention. System designers always make every effort to achieve the optimal signal-to-noise ratio (SNR) of the entire system. To achieve this goal, either increasing the signal or reducing the noise can be used. Because it is difficult to obtain high signal power in the microwave frequency band, and the circuit losses are quite severe, the received signal is very weak. The presence of noise hinders the detection of weak signals. Measurement errors caused by noise inevitably cause harm to various systems (such as radar, communication, etc.). Compared to using methods to increase transmitter power, methods to reduce receiver noise are easier and more economical.

In order to measure the noise performance of a linear circuit or system, scientists have proposed the concept of noise figure through research and conducted meticulous research on its definition, so that it can reflect the actual network noise performance. In fact, all electronic devices will increase some noise, the most common form of which is thermal noise, which increases with the temperature of the device. The device with the lowest noise is the best, and these devices have the lowest noise figure.

The following is the IEEE definition of noise figure: the ratio of the input signal to noise power ratio of a linear four port network to the output signal to noise power ratio when the temperature is at 290K (standard noise temperature).

$$\mathrm{NF(dB)} = \frac{\text{输入信噪比}}{\text{输出信噪比}} = \frac{S_i/N_i}{S_o/N_o}$$

输入信噪比	Input signal-to-noise ratio
输出信噪比	Output signal-to-noise ratio

For an ideal noise free linear network, the signal and noise entering the input end will get the same amplification through the network, that is, the signal-to-noise ratio at the output end is equal to the signal-to-noise ratio at the input end, that is, $\mathrm{NF} = 0\mathrm{dB}$. However, actual active or passive networks inevitably have internal noise, so the output noise power will be the sum of the amplified input noise and the internal noise of the network. Therefore, the signal-to-noise ratio of the output end will be smaller than the signal-to-noise ratio of the input end, i.e. $\mathrm{NF} > 0\mathrm{dB}$.

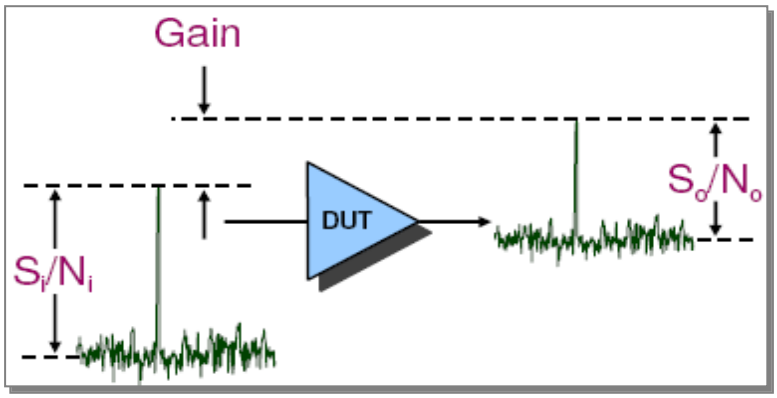


Figure 8.1 Definition of noise figure

An alternative definition of the noise figure is introduced as the ratio of the total noise power N_O at the output end of the network to the available noise power at the input end multiplied by the gain G of the network.

$$\mathrm{NF} = \frac{\mathrm{N}_O}{G_n KTB} = \frac{\mathrm{N}_O}{\mathrm{N}_{OS}}$$

Where, $K = 1.38 \times 10^{-23} j/k$ is the Boltzmann constant;

T- Noise temperature (in k);

B- Passband;

N_{OS} represents the portion contributed by input noise included in the total noise power N_O at the output end, and represents the output noise power after the input noise passes through the ideal network.The relationship with gain is:

$G_n = \frac{N_{OS}}{N_i}$ G_n- Available power gain of noise

The formula indicates that the noise figure NF is only related to the total noise power N_O and N_{OS} at the output end, and is independent of the size of the input signal. The physical meaning of the noise figure indicated by the formula is to represent the multiple increase in the output noise power of the actual network compared to the output noise power of the ideal network due to the influence of internal network noise. The formula can also be written as:

$$\mathrm{NF}_1 = \frac{S_i/N_i}{S_o/N_o} = \frac{\mathrm{N}_O}{\mathrm{G}_S \mathrm{N}_i}$$

$$\mathrm{NF}_2 = \frac{N_O}{N_{OS}} = \frac{\mathrm{N}_O}{\mathrm{G}_n \mathrm{N}_i}$$

Where, $G_S = S_o/S_i$ is the available power gain of the signal, and $G_S = G_n$ is for the linear network, so the definition of noise figure by the two methods is consistent.The advantage of the first method is that the physical concept is clear; while the advantage of the second method is that it is easy to calculate because it indicates that the noise figure is independent of the signal and only related to the noise, which simplifies the calculation of the noise figure.

The following equation calculates the noise power at 290K. To measure lower noise power, a lower temperature is necessary.

$$\mathrm{P} = 10LOG(4.0 \times 10 - 21watts/.001watt)$$
$$\mathrm{P} = -174dBm/Hz$$

8.1.2 Basic Principle of Noise Figure Testing based on 3674 Vector Network Analyzer

The two main noise figure measurement techniques currently commonly used are the Y-factor technique (hot/cold source technique) and the cold source technique. The Y-factor technology provides two known output noise powers to the input port of the DUT through a precisely calibrated solid-state noise source, and the method measures the noise figure and scalar gain of the DUT. This technology is used in both the dedicated noise figure analyzer and the noise figure test option in the spectrum meter/signal analyzer. The Y-factor measurement technology is relatively easy to implement and provides high measurement accuracy when the noise source is well matched to the source and can be directly connected to the input port of the device under test.

Another commonly used measurement technology is the cold source technology (also known as the direct noise method). This method requires only a known load (typically 50Ω) instead of the solid-state noise source in the Y-factor method. The cold-source technique requires an independent measurement of the gain of the DUT. Vector network analyzers can measure the gain of the DUT very accurately based on advanced calibration techniques, so this cold-source technique is well suited for network analyzers. This technology combines vector error correction with a unique source calibration method to provide the highest accuracy noise figure measurements available in the industry today. Another advantage of the cold source method based on the network analyzer is that a single connection can measure both the noise figure and S-parameters of the device under test, where a solid-state noise source or power meter is required for system calibration.

The cold source technology based on the 3674 vector network analyzer is relatively simple in concept. The port of the DUT (device under test) is always at room temperature (i.e., a cold source), and it is sufficient to measure the output noise power in this state. The measured noise is the sum of the noise generated by the DUT and the receiver. If the gain of the receiver and the DUT can be accurately obtained, then the receiver noise can be removed from the total noise power. In this way, the noise figure of the DUT can be obtained. The vector network analyzer is particularly suitable for accurate gain measurement of the DUT, and it can be used to calibrate various error terms. Therefore, the implementation of the cold source method in a vector network analyzer is particularly suitable.

Similar to the Y-factor method, the cold source method also requires a calibration step for the noise figure and gain of the receiver itself. A noise source can be used for calibration similar to the Y-factor method, or a power meter can be used to calibrate the Gain–bandwidth product of the receiver. It is important to note that the noise source or power meter is only used during the calibration process and is not required during the measurement process.

Figure 8.2 shows a graphical representation of input and output noise, where the slope can be measured separately. Only one measurement of the noise power in the cold source state is required to fully determine this straight line and determine the increased power of the

network under test.

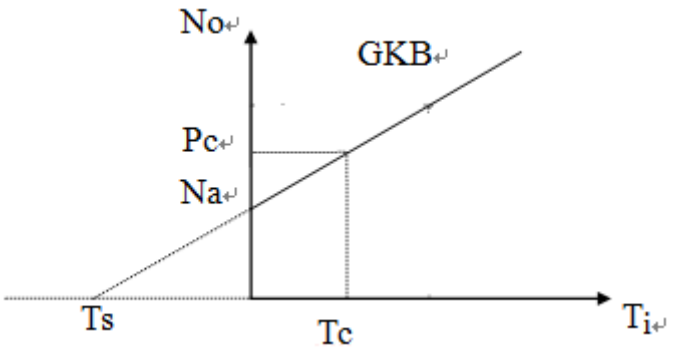


Figure 8.2 Relationship curve between noise power and source temperature

The gain of the DUT can be measured more accurately with a vector network analyzer than with the Y-factor method. Vector error correction requires obtaining the full S-parameters of the DUT, i.e., two sweeps, forward and backward, are required.

Although the gain of the DUT was accurately measured using a vector network analyzer, the effect of noise parameters was not considered here. According to the active two-port network equivalent model, it can be concluded that the noise figure is a function of the source reflection coefficient, that is, the noise figure varies with the source reflection coefficient. The definition of the noise figure is based on the premise of a standard 50Ω source impedance, and it is impossible to provide a standard 50Ω test environment in reality, so there is bound to be an error introduced by the source impedance matching problem, called the noise parameter effect error.

The relationship between the noise parameter effect and the source impedance can be expressed by the following equation.

$$F = F_{min} + \frac{4R_n}{Z_0}\frac{|\Gamma_{opt} - \Gamma_S|2}{|1 + \Gamma_{opt}|^2(1 - |\Gamma_S|^2)}$$

F_{min}- Minimum noise figure

R_n- Noise resistance

Γ_{opt}- Optimal reflection coefficient

Γ_S- Source reflection coefficient

Z_0——System impedance

This equation shows that the noise factor F varies with the source impedance Γ_S. In addition to Z_0(50 Ω system reference impedance), there are three constants (two scalar constants and one vector constant) known as noise parameters. These four noise parameters are: F_{min} (minimum noise figure), Γ_{opt}-magnitude, Γ_{opt}-phase (optimal source reflection coefficient, corresponding to the optimal source impedance of F_{min}) and R_n (noise resistance, which is a sensitivity parameter that controls the degradation rate of noise figure when the source impedance deviates from Γ_{opt}).

In the 3674 series vector network analyzer, we refer to the option that provides the noise figure and noise parameter measurement results of the DUT as the noise figure option (003 option).

8.2 Instructions of Noise Figure Option

003-noise figure option uses high-sensitivity noise receiver and needs an electronic calibration part. Adopting the advanced source matching error and vector error correction calibration technology, it can get the industry's highest noise figure measurement accuracy, and measure noise figure, noise parameters, noise power parameters and noise correlation matrix.

8.2.1 How the Noise Figure Option Works

The goal of the noise figure option is to accurately measure the noise of the DUT. This can be accomplished by the low-noise receiver specially provided to 003 option.

The low-noise receiver is calibrated by a calibrated noise source.

Matching errors in the DUT input can cause errors in the noise figure measurement. In each measurement, the noise figure option uses an electronic calibration kit module to provide at least four different impedance states at the input of the DUT. This noise tuner is connected to the internal source jumper of port 1 of the 3674 series vector network analyzer. Although the port resistance of the 3674 is 50Ω, the 3674 obtains the noise output of the DUT through multiple measurements and calculations of different impedance states. Here there is no preset condition for the input impedance of the DUT.

The following describes how the noise figure measurement of the 003 option is performed.

a) The noise tuner is in the through state and the exact gain of the DUT is obtained using S-parameter measurements. This requires forward and reverse sweeps.

b) The source power of the 3674 vector network analyzer is turned off and the noise tuner is switched to the first state.

c) At each frequency point, the noise receiver collects a large amount of data in order to get a valid measurement. If averaging is selected in the noise settings, the measured noise power is also averaged again at each frequency point based on the average times. All frequency points are measured in this way.

d) The noise tuner is switched to the next state and the steps in step 3 are repeated until all the tuner states are performed. A complete measurement requires at least 4 tuner states.

e) By calibrating the error term, the calculation formula is applied to obtain the noise figure and noise parameter 50Ω ohms. The measurement results are displayed on the 3674 vector network analyzer.

f) Return to step 1.

8.2.2 Noise Figure Option Measurement

The noise figure calibration method is divided into three methods: noise parameter, noise figure, and S-parameter. Among them, the noise parameter calibration requires noise tuner (electronic calibration kit) to give noise figure and noise parameter measurement results, while the noise figure calibration method does not require electronic

calibration kit and only gives noise figure measurement results. The S-parameter calibration only gives the S-parameter measurement results.

As described in the previous section, the noise tuner switches between at least four states in a single sweep. These states are not needed in the noise figure calibration option, so that the measurement is faster. Of course, the noise figure calibration is not as accurate as noise parameter calibration because it assumes that the input and output impedances of the DUT are both 50 ohms. When adding an attenuator close to the input of the DUT, the closer the attenuator is to the place, the higher the measurement accuracy can be obtained, because this improves the effective source matching of the system. The effect of the attenuator's attenuation amount can be removed during the calibration process.

Due to the use of the noise figure calibration method, it is not necessary to connect a noise tuner. If the noise tuner is still connected, it will be switched to a through state when the noise figure calibration is applied. However, the accuracy will be affected by the insertion loss of results. To improve the measurement accuracy, the noise tuner switch can be manually turned to the internal position.

In the noise figure calibration, the port and DUT are considered to be matched, so there is no need to consider the effect of noise parameters. The receiver calibration is first performed to get the noise figure and gain of the receiver. Then the DUT is connected and the noise figure of the DUT is obtained after measuring the noise power in the cold state.

8.2.3 Block Diagram of Noise Figure Option

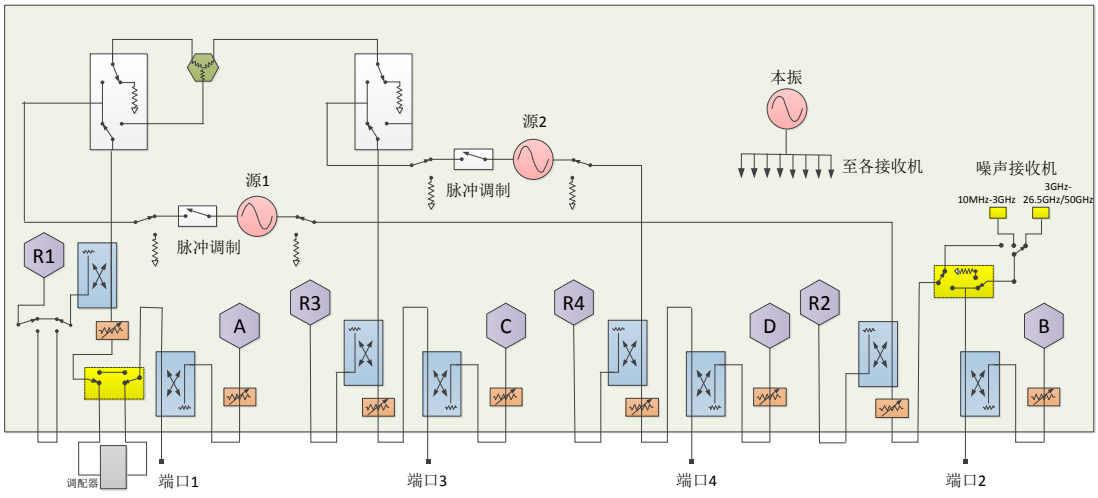


Figure 8.3 Block Diagram of Noise Figure Option

调配器	Tuner
端口 1	Port 1
端口 2	Port 2
端口 3	Port 3
端口 4	Port 4

脉冲调制	Pulse modulation
源 1	Source 1
源 2	Source 2
本振	LO
至各接收机	To each receiver
噪声接收机	Noise receiver

The noise figure component is shown in yellow.

The front panel jumper of test port 1 and the switch of noise tuner are connected to the source 1 output channel to provide multiple source input impedances.

In test port 2, a switch sends the RF signal from the output of the DUT to two noise receivers. The test frequency program will automatically switch the RF signal to the appropriate receiver.

The port 1 noise tuner switch is turned to the outside by default, as shown in Figure 8.3. At this setting, input power is provided through the front panel jumper and noise tuner. When the mixer is connected, the noise tuner can be in the through state and used for accurate S-parameter measurements.

The switch state can be changed in two ways:

1) Use the channel configuration dialog box to set the switch to internal to the S-parameter channel

2) Use the remote control command:

Set the switch to internal (S-parameter channel) by using the following SCPI command:

"Port1NoiseTuner", "Internal"

Set the noise tuner (electronic calibration kit module) to a through state by using the following SCPI command:

Notice

003 option switch selection:

If the switch state is switched to internal, the switch must be manually switched to external.

8.3 Use the Noise Figure Option Application

Use the following steps for the 003 option measurement.

a) Connect the tuner and the noise source.

b) Create a noise figure measurement class.

c) Set the noise figure option.

d) Copy the ENR file to a suitable folder or create a new ENR file.

e)Perform calibration

f) Connect the DUT and measure the noise figure.

g) Optionally, click "File" to store the noise figure measurement data.

8.3.1 Connect the Noise Tuner to the Noise Source

Connect the noise source to the 28V connector on the rear panel of the 3674. The noise source will automatically turn on or off when calibrated. When prompted during calibration, connect the noise source to the reference side of port 2.

Connect the noise tuner (electronic calibration kit).

A. On the 3674 front panel, remove the jumper "Source Out to Source In" from the port 1.

B. Connect the noise tuner to the jumper between "Source Out" and "Source In".



Figure 8.4 Connecting the tuner

8.3.2 Create a Noise Figure Measurement

1. At the 3674 auxiliary button, click [Measurement] and then [Measurement Class].

2. Select [Noise Figure Cold Source] and then [OK] to delete an existing measurement, or check [Create in New Channel] to create a measurement channel in a new channel.

3. A noise figure measurement curve is displayed on the screen. Then, it will display show how to select and change each parameter.

1) Noise parameters

All four noise parameters, as well as the standard S parameter, can be measured in the noise measurement channel.

Add noise parameters

a) Create a noise figure measurement channel

b) Select [Trace] of the auxiliary button or use the mouse to select [Trace]

c) Select [New Trace].

d) Select a parameter

Change the noise parameter

a) Create a noise figure measurement channel

b) Select a parameter to be changed

c) Select [Measure] of the auxiliary button or use the mouse to select [Response]

d) Select a parameter

2) The noise measurement that can be provided

The 3674 vector network analyzer can provide the following 3 types of noise measurement options.

Noise figure NF - the DUT increases the noise metric 50Ω ohms.

Noise power parameter SYSNPD - the noise power output of the DUT at 50 ohms. These power parameters are used to calculate the noise figure through the power measurement of the DUT.

Noise parameter - the model generating the noise inside the DUT, which is similar to the S-parameter model that characterizes the effects on the RF signal flowing through the DUT.

Noise Figure (NF) - refer to the previous section 8.1

Excess Noise Ratio-selected when measuring noise sources. The ENR table is compared to determine the accuracy of the system. ENR is calculated by the following equation:

$$ENR(in\ dB) = 10log10(\frac{Thot-Tcold}{T0}),\text{T0=290K}$$

T-Effective-Effective temperature in Kelvin, or the measured noise level. As an example:

290K=-174dBm/Hz

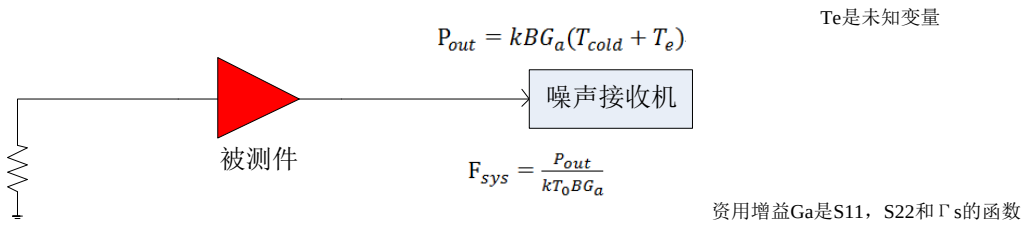


Figure 8.5 Noise receiver power formula

被测件	DUT
噪声接收机	Noise receiver
Te 是未知变量	Te as a variable
资用增益 Ga 是 S11，S22 和Γs 的函数	Available gain Ga is a function of S11, S22 and Γs

3) Noise power parameters

Noise power parameters are given in the following two forms:

Available Noise Power: the power calculated under the condition of ideal matching at

the output of the DUT. These parameters are always available in the 3674 noise option.

Incident Noise Power: An "I" is added to the Available Noise Power. It is the power calculated with an ideal 50Ω receiver input, regardless of the output power of the DUT.

SYSNPD / SYSNPDI: System Power Spectral Density: The total power measured at the ADC, including the noise power caused by the DUT and the internal noise receiver. It is usually expressed in absolute power, namely in dBm or Watts or Kelvin.

$$\mathrm{dBm} = 10\log_{10}(\mathrm{K} * \mathrm{T} * \mathrm{B} * 1000)$$

SYSRNP / SYSRNPI: System Relative Noise Power: The value of the noise temperature of the DUT and the receiver relative to 290K, which is usually a ratio shown in dB. Thus a perfect and quiet device would show a trace at 0dB.

$$\mathrm{dB} = 10\log_{10}(\mathrm{T}/290)$$

DUTNPD / DUTNPDI: Noise power spectral density of the DUT: When Calibration is turned on, this trace shows the available noise power, which is the maximum available power output from the DUT when the impedance of the noise receiver port and the output impedance of the DUT are conjugately matched. More precisely, this occurs when the noise receiver port match is equal to the conjugate of the DUT output port match. The receiver's contribution to the noise power is removed.

When Calibration is turned off, this curve shows a transmitted power. The transmitted power is the true power as seen on the ADC. The effect from the mismatch between the receiver and the DUT is ignored. The noise power of the receiver is removed.

Measurements are expressed in dBm, which is normalized to a 1 Hz bandwidth. For convenience, the marker and curve readings are displayed in dBm.

The power can be displayed to different bandwidths using the formula calculator.

$$\mathrm{dBm/Hz} = 10\log_{10}((\text{DUT Temperature} - \text{Receiver Temperature}) * \mathrm{B} * 1000)$$

In the above equation, B is the bandwidth and 1000 is changed from W to mW.

DUTRNP / DUTRNPI DUT: relative noise power of DUT: measure the noise temperature of the DUT relative to 290K. It is usually expressed as dB. The specific meaning is the same as DUTNPD above.

$$\mathrm{dB} = 10\log_{10}(\text{DUT Temperature} - \text{Receiver Temperature})$$

4) Noise model, noise parameters, noise correlation matrix

Noise parameter is the model generating the noise inside the DUT, which is similar S-parameter model that describes how the RF signal flows through the DUT.

Noise model

The arbitrary linear two-port noise wave model can be represented by the following figure:



Figure 8.6 Linear two-port network noise wave model

无噪声两端口网络[T]	Noise-free two-port network [T]

The above figure shows a two-port noise-free network with input noise waves ofc_{t1} and c_{t2}. a1, a2 and b1, b2 are standard S-parameter waves. The noise correlation matrix of the noise waves is shown as follows:

$$C_t = \begin{bmatrix} \overline{|a_{n1}|^2} & \overline{a_{n1}b_{n1}^*} \\ \overline{b_{n1}a_{n1}^*} & \overline{|b_{n1}|^2} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} \\ C_{21} & C_{22} \end{bmatrix}$$

Where:

$\overline{|a_{n1}|^2}$ and $\overline{|b_{n1}|^2}$ is the time-averaged noise power in the 1 Hz bandwidth;

$\overline{a_{n1}b_{n1}^*}$ and $\overline{b_{n1}a_{n1}^*}$ is the time-averaged inter-correlation term;

The horizontal line above represents the time-average;

The asterisks represent the complex conjugates;

Noise parameters

GammaOpt (optimal complex reflection coefficient) - the optimal impedance in the noise figure measurement. The following data display formats can be selected:

Gamma-Opt-Magnitude and Gamma-Opt-Phase (most commonly used), or Gamma-Opt-Real and Gamma-Opt-Imaginary.

NFmin - The minimum noise figure, which appears at the optimal reflection coefficient.

Rn (Noise Resistance) - Specify the rate of change of noise magnitude when changing the source impedance.

NCorr_11, NCorr_21, NCorr_12, NCorr_22 - NCorr_11 and NCorr_22 are effective noise temperature terms, which are normalized to 290K. The both terms are time-averaged noise wave power at the input of the DUT. NCorr_11 is the forward input wave (noise passing through the device to the output) and NCorr_22 is the reverse noise wave (noise output from port 1 of the DUT, reflected to the source).

In order to change to the available noise power, it is multiplied by 290*k*B to the terms in the matrix:

k is the Boltzmann constant;

B is the system bandwidth.

Standard parameters (amplifier only)

S-parameters: S11, S21, S22, S12

Non-proportional parameters, using the following labeled order (receiver, source port). These parameters replace the active measurement. To select from the auxiliary buttons, click [Measurement] and then [Raw Receiver Parameters]. The following parameters are available:

(R1_1), (R2_2), (A_1), (A_2), (B_1), (B_2)

8.3.3 Noise Figure Measurement Settings

·How to open the noise figure setting window

1. [Measurement]-> [Measurement Class]-> [Noise Figure Cold Source]
2. [Measurement] → [Noise Setting]

The following figure shows the noise setting window

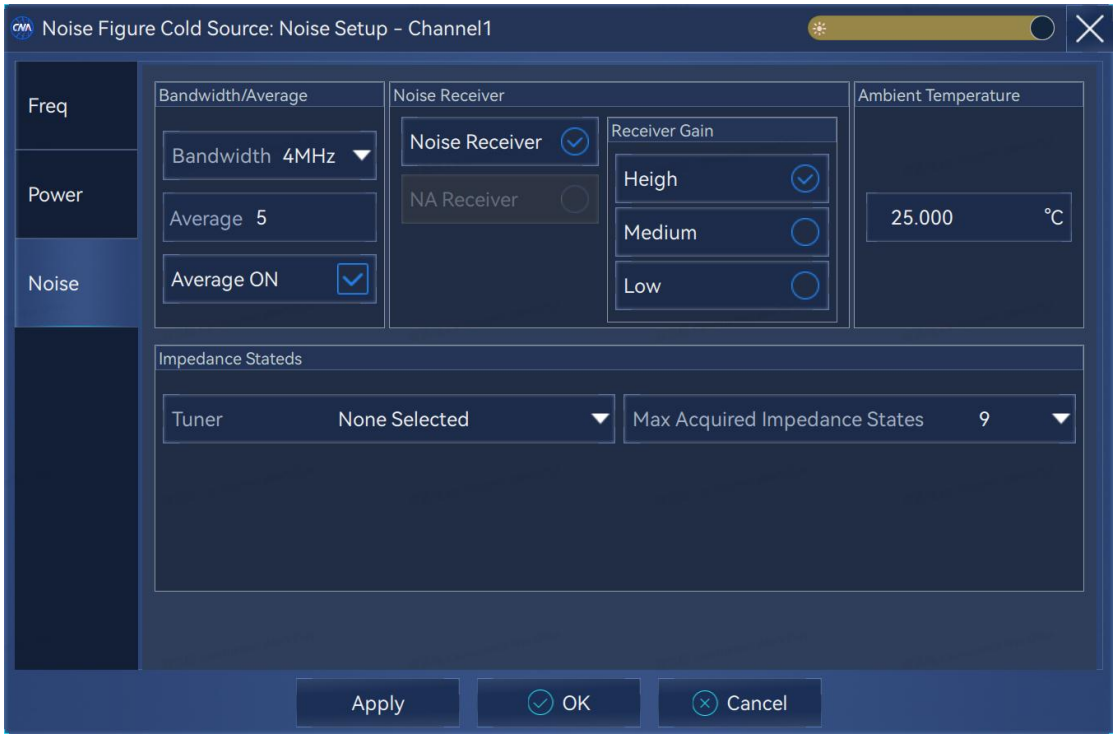


Figure 8.7 Noise measurement settings

Notice

Here, jitter is used to describe the curve-to-curve fluctuations in the measurement. In other places, it is also referred to as trace noise.

BW/Avg

The next settings get the optimal measurement accuracy relative to a certain measurement speed.

Noise bandwidth: Increasing the bandwidth reduces trace noise in noise power and noise figure measurements. However, a larger bandwidth reduces the frequency accuracy of the measurement. The noise bandwidth setting should be narrower than the bandwidth of the DUT. The noise bandwidth setting is only used for noise power measurements and is independent from the IF bandwidth setting for S-parameter measurements. The noise figure is obtained from the noise power measurement and the S-parameter measurement.

Calibration and measurement should be performed at the same noise bandwidth. If the noise bandwidth is changed after calibration, the noise figure measurement will change by 0.5dB or more, depending on the frequency, gain and noise figure of the DUT.

Notice

The noise bandwidth may be automatically adjusted at lower frequency bands, as shown in the table below. At each frequency point, if the specified noise bandwidth is higher than in the table below, then the noise bandwidth is automatically set to the maximum bandwidth in the table.

Table 8.1 Noise bandwidth vs. input frequency

RF input frequency	10-25MHz	25-60MHz	60-150MHz	150MHz以上
Allowable noise bandwidth	0.8MHz	0.8MHz	0.8MHz	0.8MHz
	2MHz	2MHz	2MHz	2MHz
		4MHz	4MHz	4MHz
			8MHz	8MHz
				24MHz

Notice

In the case of noise bandwidth 8MHz and 24MHz, the power meter cannot be used for calibration. The power meter calibration function is not currently released.

Average times: Increasing the average times reduces jitter. However, this also reduces the measurement speed. To have better measurement accuracy, use the recommended average times for noise calibration below. When using a noise receiver, 5 averages can be used.

In the measurement, the gain of the DUT will make the noise power exceed the vector network receiver significantly. As a result, in order to improve the speed, the average times can be reduced without much loss of measurement accuracy.

Noise receiver

Noise receiver - Noise figure is measured using an internal low-noise receiver.

1. The measured noise figure ranges from 0 to 50dB and the device gain ranges from -40 to 60dB.

2. The noise figure of the high gain amplifiers can be measured by adding an attenuator to the output of the DUT and then removing the value of the attenuator using a fixture de-embedding method.

Receiver gain

With your prior knowledge of the DUT gain, set the appropriate receiver gain to optimize the noise power level received by the noise receiver.

The following values reflect the gain of the DUT and the noise figure NF and SUM.

For example, if gain=10dB and NF=6dB, then SUM=16dB.

If SUM<30dB, select high gain;

If 20dB<SUM<45dB, select medium gain;

SUM> 35dB, select low gain;

The above choices have overlapping parts that can be considered. Because the above three noise gains are calibrated each time the noise is calibrated, the gain setting is selectable after calibration to avoid noise receiver compression and to select a better gain for the minimal jitter.

Only one gain can be used in a noise measurement in the full frequency band. Therefore, two noise channels can be used to set different gains at different frequencies to achieve the best noise figure measurement accuracy.

Ambient temperature

Enter the room temperature in degrees Celsius at the time of the measurement. For better results, use a thermometer to measure the temperature at port 2 (noise receiver input) or at the output of the DUT.

Notice

This setting is only used for calibrated noise figure measurements, but has no effect on uncalibrated noise figure channels. **The default value is used for measurements that are not calibrated.**

There is an inverse relationship between the spatial temperature and the noise figure. When using the effective noise input temperature Te format, a 3 degree increase in spatial temperature will reduce the measurement by 3 degrees and will have an impact on the subsequent noise figure measurements.

Impedance Status

Tuner： Display the name of the electronic calibration kit used as the impedance tuner. You can select the electronic calibration kit to be used during calibration.

Maximum Captured Impedance State: Select the number of impedance states to be used for noise measurements. A minimum of four impedance states are required.

Frequency Settings - Noise settings dialog box



Figure 8.8 Noise settings dialog box

The settings here are the same as those under the standard category of vector network analyzer.

Swp Type

Select a sweep type.

Sweep settings

Set points.

Set IFBW. IF bandwidth is important to improve measurement accuracy.

Start/stop frequency, center/span frequency.

Frequency settings - Noise settings dialog box

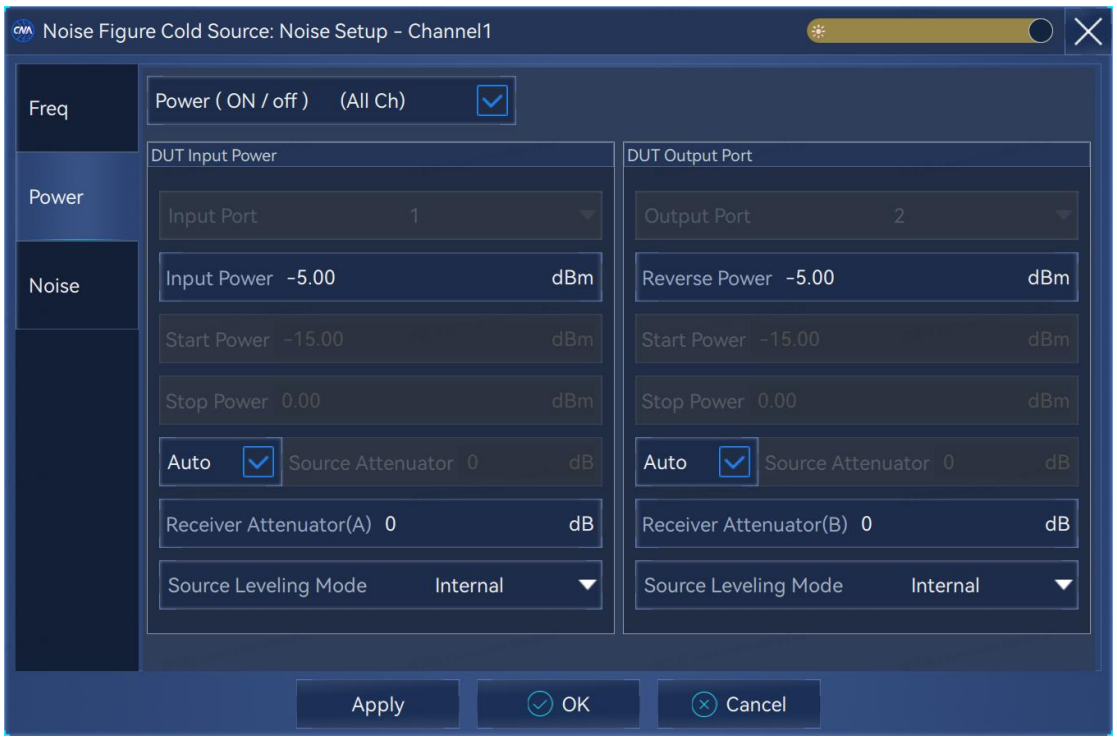


Figure 8.9 Noise power setting

Notice

S-parameter power settings are critical for accurate noise measurements.

Configure the RF power needed for S-parameter measurements before doing noise measurements. The input power of the DUT is turned off during noise measurements.

These settings can also be made in the place of the normal power settings.

[Turn Power On] (all channels): Check √ to turn RF power on for all channels.

DUT input port

003 Option - The DUT input can be set to any port except port 2 (currently only port 1 can be selected). However, without the noise tuner bypass switch, other measurement channels using the same source port will always pass through the noise tuner. The noise tuner must be connected to the source path of the selected port.

Notice

Input power is critical for accurate noise measurements.

Input power: The power at the time of the S-parameter measurement of the DUT.

Source attenuation: If a source attenuator is available, manually select the attenuation value required for the input power. All vector network channels have the same

attenuation value in a continuous sweep.

Receiver attenuation (A): designate the receiver attenuation value for Port 1.

Source fixed amplitude mode: Specify the fixed amplitude mode Select internal fixed amplitude. Open loop is only available for broadband pulse measurements (not available in noise measurement mode).

DUT output port

Connect the DUT output port to port 2.

Output Power: Set the power of the reverse sweep input and output ports. Port power is automatically uncoupled. There is always a reverse sweep of the DUT for full dual port sweep. Enhanced response calibration cannot be used for noise measurements.

Source attenuation: Specify the source attenuator setting for reverse power.

Receiver attenuation (B): Specify the receiver attenuator setting for the output port.

Source fixed amplitude mode: Specify the fixed amplitude mode Select internal fixed amplitude.

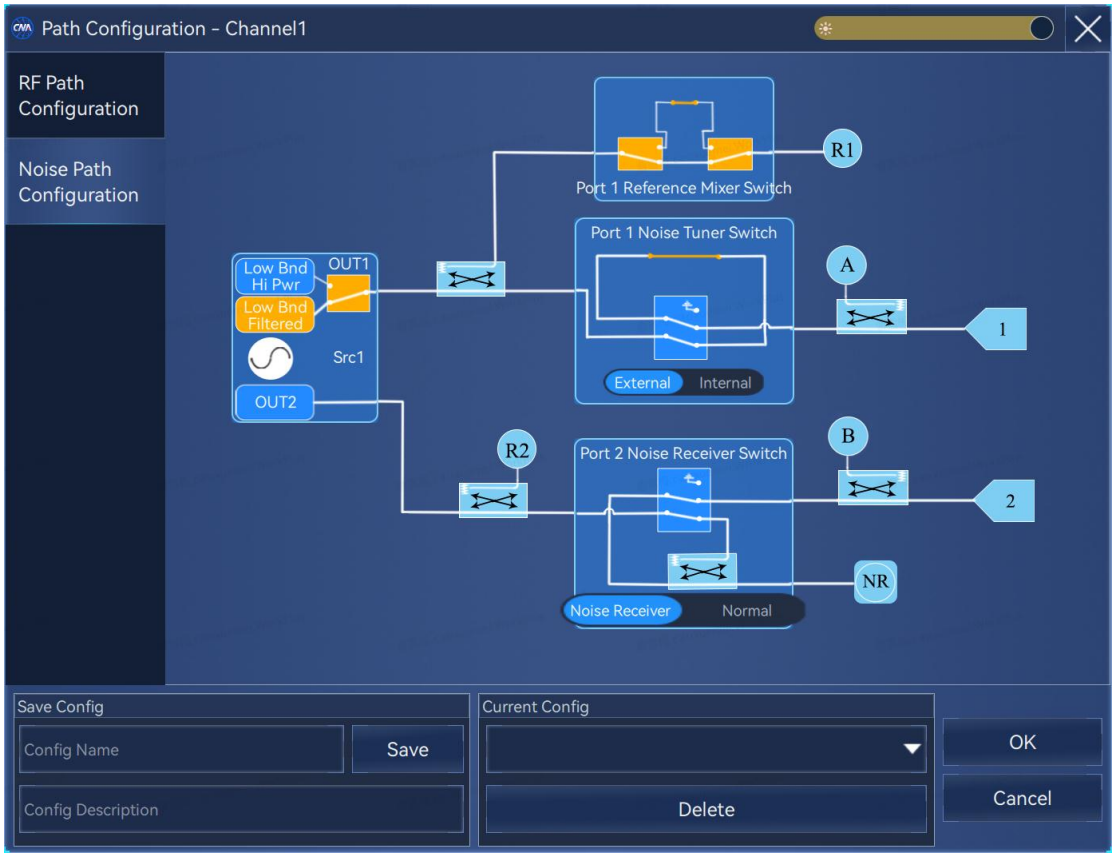


Figure 8.10 Noise channel settings

Port 1 Noise tuner switch

The orange line between the source input and source output represents the noise tuner.

External: Select an external mixer for noise measurements.

Internal: Bypass noise tuner.

Port 2: Noise Receiver: Perform noise measurements.

To avoid premature damage to the switch, the vector network analyzer will not be turned on when switching between the noise and non-noise channels. In order to have a continuous trigger for noise and non-noise measurements on different channels, the switch should be set to the same state as the noise channel:

In order to make the non-noise measurement channel effective, go to the Noise Channel Settings dialog box for settings.

Set the noise tuner switch to external so that the source signal is transmitted to the front panel jumper and passes through the connected noise tuner. Use the command to set the noise tuner to through (THRU) state internally, with a little extra attenuation for noise tuner.

Set the ON/OFF of the noise receiver.

8.3.4 Noise Figure Option Calibration

This section discusses amplifier noise figure and converter noise figure calibration.

·Overview of noise figure measurement calibration

Notice

Without the measurement calibration of the noise figure option, the noise figure result is inaccurate.

Converter noise figure calibration

The noise figure option calibration for amplifiers and converters is very similar.

The converter noise figure calibration does not provide an "S-parameter only" calibration.

The converter noise figure calibration provides an optional calibration of LO power.

The converter noise figure calibration may produce an error message for the calibration kit frequency.

Calibration noise receivers

The process of performing noise figure calibration using a noise source or a vector network analyzer source is different from that using a power meter.

Use a low-noise receiver

Step 3 will definitely be performed (measuring the frequency response of the noise filter) with the frequency range specified by the set noise channel.

The gain bandwidth product information is included in the noise figure calibration set.

Notice

In the case of noise bandwidth of 8MHz and 24MHz, the “power meter calibration” cannot be used. Noise averaging is not automatically turned on.

Use noise source calibration

A noise source is a device that produces two constant noise powers over a specified frequency range.

Hot (ON) noise sources plus bias provide a high noise power.

Cold (OFF) noise sources without bias provide an ambient noise power.

Noise source power levels are measured by the manufacturer and provided in both tabular and electronic manners, with each noise source having a serial number and being provided separately to the user. The electronic file is the Excess Noise Ratio (ENR) file.

The noise source is connected to the noise receiver via port 2.

At each measurement frequency, the noise source is measured by the noise receiver. The difference between the known ENR noise power level and the measured noise power level is the noise error term. The error term is removed in the next measurements.

In the noise source measurement, the noise average and noise bandwidth are automatically turned on and set to the specified values.

Notice

For the highest accuracy, the noise source should be connected as close to port 2 as possible (minimal electrical differential loss). This will make the biggest difference between the hot (ON) and cold (OFF) state of the noise source.

Noise receiver calibration

The two port S parameters are calibrated in the noise figure measurement channel. This is because the S-parameters are measured prior to each frequency noise measurement. In the S-parameter calibration, at least four different impedance states are to be present at port 2 in order to characterize the noise of the noise receiver. The calibration can be SOLT or TRL.

After the calibration, the correction is automatically switched to ON. The vector network analyzer status bar shows Calibration to be On.

Method to perform noise figure calibration

1) Activate the noise figure measurement class.

2) Connect the noise tuner to the vector network analyzer (for 003 option noise calibration).

3) Click on [Calibration] → [Calibration]

The Noise Calibration Wizard is made specifically for noise measurements. The steps shown below are the same as in the calibration wizard.

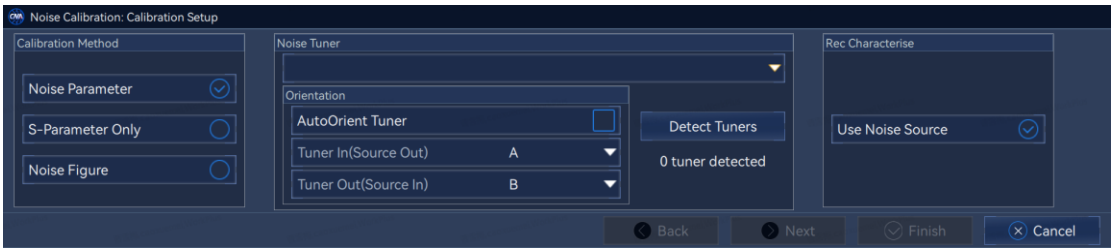


Figure 8.11 Noise calibration wizard

Calibration method

Noise parameters: Noise figure and noise parameter calibration

S-parameters: Calibrate only the S-parameters, not the noise receiver.

Noise figure: Contain only noise figure measurements.

Notice

When calibrating the LO, there is no correction if a connector is connected from the power probe to the LO port.

Noise tuner

Noise figure calibration method is not available

Select the electronic calibration kit module connected to the USB

Direction

Adaptive: Automatically select the direction of the mixer when selected. If not selected, the following two items are used to provide a manual method of selecting the mixer direction

Tuner Input (Source Output)/Tuner Output (Source Input): Specify the direction of the electronic calibration kit connected to the front panel jumper of vector network analyzer.

Search for Tuner: Click the button to re-search for a noise tuner (electronic calibration kit) connected to the USB. If the calibration kit module does not detect it, check the USB connection and click this button again. The number below this button represents the number of calibration kits connected to USB.

Receiver characterization

Use Noise Source - The Noise Source is used to characterize the noise receiver.

Configure the noise source

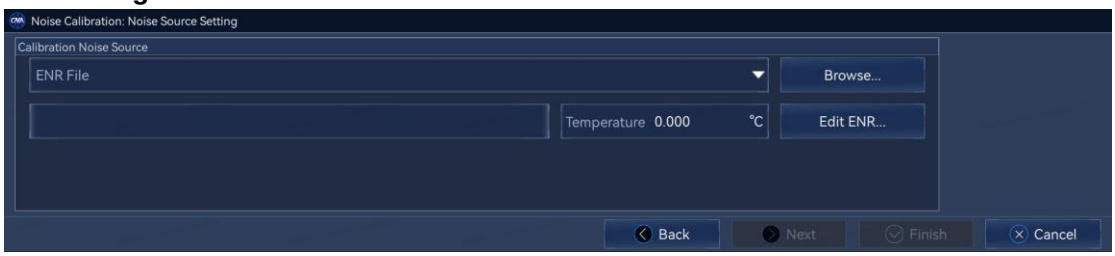


Figure 8.12 Configure noise sources

ENR file: Select the noise source ENR file. Copy your ENR file to the specified directory if you do not already have it.

Edit ENR: Click to load the ENR Edit dialog box to change and create the ENR file. This is not often used.

Temperature: Specify the temperature of the noise source connector. The noise source is always on at the time of measurement. This will make the noise source 4 to 5

degrees warmer than its surroundings, and setting it this way will make the calibration more accurate.

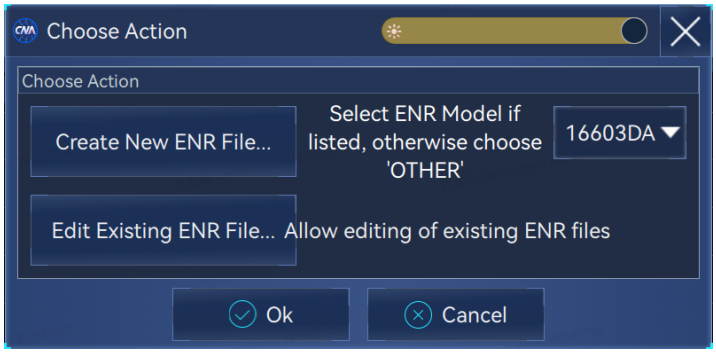


Figure 8.13 Create or edit an ENR file

Clicking on Create or Edit invokes the same dialog box, as shown below.

Edit: Already filled with all fields that can be edited.

Create: All fields are empty except for the frequency.



Figure 8.14 ENR Editor

ENR data

Use the forward-backward buttons to go to entry # for editing.

Done: Click **Done** to edit all values. Then click to save the ENR file.

Identify Information

Model#: The model number of the noise source. This cannot be changed.

Serial number#: The serial number of the noise source.

Temperature and humidity: The temperature and humidity at the noise source. This is used here for identification information only. ENR data is always normalized to 290K…

Save ENR file: Click on the Call dialog box to select a new or in-edit ENR file.

Select DUT connector and calibration kit dialog box

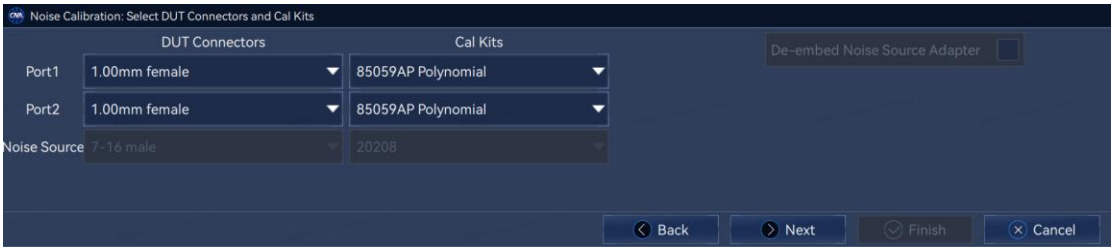


Figure 8.15 Calibration kit and DUT type settings

Port 1 and Port 2

Connection type of DUT: Specify the type of connector for the DUT.

Calibration kit: Select the calibration kit to be used to calibrate each port. The list shows the same type of DUT port as the calibration kit port type. Using an incorrect calibration standard will reduce the measurement accuracy.

Noise Source: calibrate noise receivers.

Notice

For the highest accuracy, the noise source should be connected as close to port 2 as possible (minimal electrical differential loss). This will make the biggest difference between the hot (ON) and cold (OFF) state of the noise source.

For the calibration device (noise source), specify the connector type. When the calibration device connector and the DUT port connector type and polarity (negative/positive) are different, it is necessary to add an adapter between the calibration device and the reference surface for optimal accuracy. At this moment, it is necessary to add additional steps to measure and calibrate the error caused by the adapter.

Ignore (at the bottom of the DUT connector): there is no compensation for the DUT.

Select the calibration toolbox used for the calibration process.

Noise Adapter De-Embedding: The vector network analyzer needs to determine whether adapter de-embedding is required based on the type and negative/positive of the DUT and the type and negative/positive of the calibration kit, and whether de-embedding requires an additional calibration step.

However, if the DUT Type is used, the required additional step will be hidden in some special cases. In these cases, select "Noise adapter de-embedding" to inform the vector network analyzer of the additional steps required.

In this case, as shown in the figure below, the noise source is connected directly to

port 2 to get a higher measurement accuracy. If not selected, the vector network analyzer will assume that the noise source in this case is connected to the input reference plane of the DUT in port 1 of the through standard.

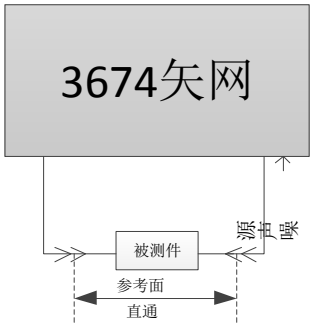


Figure 8.16 Noise adapter de-embedding

3674 矢网	3674 vector network analyzer
被测件	DUT
参考面	Reference plane
直通	Through
噪声源	Noise source

Modify Calibration: Click “**Modify Calibration”** and then “**Next”** to change the calibration method (Standard and Thru methods).

Notice

Enhanced response calibration is not supported inside the noise figure measurement.

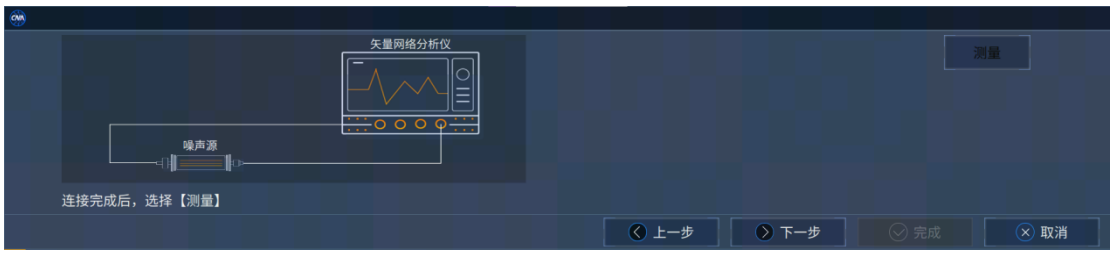


Figure 8.17 Calibrate the noise source

Connect the noise source to the port 2 measurement (reference) plane

When the "Noise Adapter" is de-embedded, an additional calibration step is required.

The following steps

Connect Port 1 to Port 2 - Connect the port 1 reference plane to the port 2 reference plane using the desired through standard or adapter.

Connect Ecal between Port 1 and Port 2 - Connect the Ecal module from the port 1

reference plane to the port 2 reference plane.

Verify noise source calibration

To verify the noise source calibration, connect the noise source to port 2 and measure the ENR.

Compare the values in ENR table with the measured ENR values.

Manually switch the noise source on and off

[Power] → [Noise Source OF|OFF]

8.3.5 Measure the Noise Figure

Notice

Jitter is used to describe the curve-to-curve fluctuations in the measurement. In other places, it is also referred to as trace noise.

Noise figure measurement tips

The measured noise figure of the device ranges from 0 to 50dB and the device gain ranges from -40 to 60dB.

The highest noise figure measurement accuracy can be obtained when the sum of the noise figure and gain of the DUT is between 0 and 55 dB.

In order to obtain the highest noise figure measurement accuracy and stability, the attenuation between the DUT and the vector network analyzer port 2 should be kept to a minimum.

DUT output power level

The S-parameter is used to measure the gain of the DUT prior to the noise measurement. The jitter of the S-parameter measurement is directly reflected in the noise measurement.

In the noise measurement, in order to get the best noise figure measurement accuracy, the output power of the DUT in the S-parameter measurement section should be 15 to 20 dB lower than the output compression point of the DUT.

To reduce jitter, the power of the B receiver (port 2) should be greater than- 20dBm. If it is lower than -20dBm, the S-parameter measurement will have a large jitter. Input power must be less than 10 dBm because the B receiver is compressed. However, the S-parameter measurements is not prompted at the beginning of the compression.

The best way to control this power is to display a (B_1) measurement. When a DUT is being measured and the power is turned on, change the input power and note the power of the B receiver.

For low gain DUTs, select 5dB of source attenuation to improve the uncalibrated match of port 1.

For high gain DUTs, both source and receiver attenuation are required. Use the smallest attenuation value.

Because there is no DUT at calibration, it is also challenging to get the optimal power

level at calibration. Therefore, it is necessary to set a higher source power during calibration than during measurement. As long as the programmable step attenuator and individual switch positions are constant and the receiver is in the linear region (below +10dBm), the accuracy of the measurement results is guaranteed. Before calibration, the best power and attenuation value settings for calibration and noise measurements should be found.

IFBW

Reducing the IF bandwidth will reduce jitter accordingly. If the DUT has a large gain, calibration needs to be performed at a low source power or large receiver attenuation. Then, the IF bandwidth must be reduced during calibration to reduce jitter. An increase in IF bandwidth can increase the measurement speed.

Noise measurement settings

Refer to the Noise Settings dialog box for to get details on these important settings.

Temperature

Noise figure measurements are very sensitive to temperature.

Interference

When measuring noise figure in an unshielded space, such as an amplifier on a printed circuit board, it is susceptible to interference from external signals such as cell phone signals, wireless LAN or broadcast signals. This signal appears as a spike in an unrepeated measurement.



Figure 8.18 Potential spatial interference in noise figure measurements

通道 1	Channel 1
起始	Start
终止	Stop

Usually, interference affects the noise figure measurement at the frequency where it occurs. However, if this interference is large enough to exist for a long enough time, it can cause compression of the receiver, causing problems at many measurement frequencies. When this occurs, noise figure measurements should be made in a shielded darkroom.

Using the formula editor in the noise figure traces

In the noise power traces, the units of the following equation are the noise temperature

$$10 * log_{10}(Temperature * 1000mw/w * 1.38 * e^{-23})$$

$1.38 * e^{-23}$ is the Boltzmann constant.

When using the formula editor in a noise power trace, the above formula will use logarithmic format. Therefore, select the real part format first, and then generate an equation.

RF frequency electromagnetic fields

Test performance may be degraded if there are surrounding EMFs interfering with the vector network analyzer. When the surrounding electromagnetic field happens to match the frequency at which the noise figure and gain are measured, the measured value will not match the desired value. This phenomenon only affects specific frequencies, and the analyzer can still make normal measurements for other frequency points.

8.3.6 Saving Noise Figure Option Measurement Data

In order to save the noise measurement data, click **[File]**, then **[Save Data As]**, and then select the storage type:

(*.prn), (*.cti), (*.csv), (*.mdf) - Noise figure data can be stored in these formats only. PRN stores the running curve. CITI format, CSV format, and MDF can store all displayed curves.

(*.s2p) - S-parameter data is stored after the noise calibration is done. This data will be saved not matter whether the noise measurement is being displayed or not.

Trace and Noise parameter (*.s2p) - save the S-parameter data and then save the noise parameter data. This data will be saved not matter whether the noise measurement is being displayed or not. If the vector calibration is not on or the noise parameter is not recognized, then the noise parameter has no calculated value. In this case, the following values are displayed:

$$\textbf{Gamma}_{\textbf{Opt}} = 0$$

$$\textbf{NF}_{\textbf{min}} = \text{raw noise figure}$$

$\textbf{R}_{\textbf{n}}$= Z0 / 4 * (F - 1). This equation shows how Rn calculates the problematic data. F is the noise factor, and F is related to the noise correlation matrix value Ct11 and the normalized noise temperature Tn. F=1+ Ct_{11}=1+ Tn, so $\textbf{R}_{\textbf{n}}$= (Z0 / 4)×Ct_{11}.

NoiseCorr (*.nco) - the noise correlation data will be saved no matter whether the noise measurement is being displayed. *. The nco file is the noise correlation matrix expressed in T-parameter form (Ct11 , Ct21 , Ct12 , Ct22). These parameters are consistent with NCorr_11, NCorr_21, NCorr_12, NCorr_22. When the vector calibration is off, the data is set to -200dBm.

8.4 Converter Noise Figure Measurement

This section discusses converter noise figure measurements:

Requirements and Limitations

If the converter noise figure function should be turned on, both the noise figure option (003) and the converter (S82 or S83) should be turned on in advance.

Limit

- Only the down-converter or up-converter can be used at one time (not both);
- Mirror suppression is not available in the NFX;
- The following vector network analyzer functions are not available in the NFX;
- Analog sweep (only step sweep is provided);
- Independent IF bandwidth, power level, or sweep time is not supported in a segment table;
- Logarithmic sweep;
- Electronic calibration kit user definitions;
- Time domain;
- Balanced device testing;
- Save to citfile format;
- Interface control;
- Some fixture features;
- Control of external test devices;
- External DC devices;
- Pulse measurement;
- Frequency restrictions;

How to configure your hardware

The 3674 vector network analyzer is versatile. The converter noise coefficient tests can be configured in a variety of ways. Not all of the configuration ways are summarized herein. A few of the most common examples are listed below to show the basics.

DUT configuration

The DUT's LO can be connected to an external source, or to a second source of the vector network analyzer (if available).

Select the LO source in the Converter Noise Figure Settings tab.

Notice

The noise at the LO source will be transmitted directly to the output of the DUT. This noise will not be calibrated in the measurement. Therefore, compared with a source inside the vector network analyzer, it is better to choose a low-noise source as the LO, such as a dedicated signal source.

LO source inside the vector network analyzer

Port 3 or port 4 of a four-port vector network analyzer

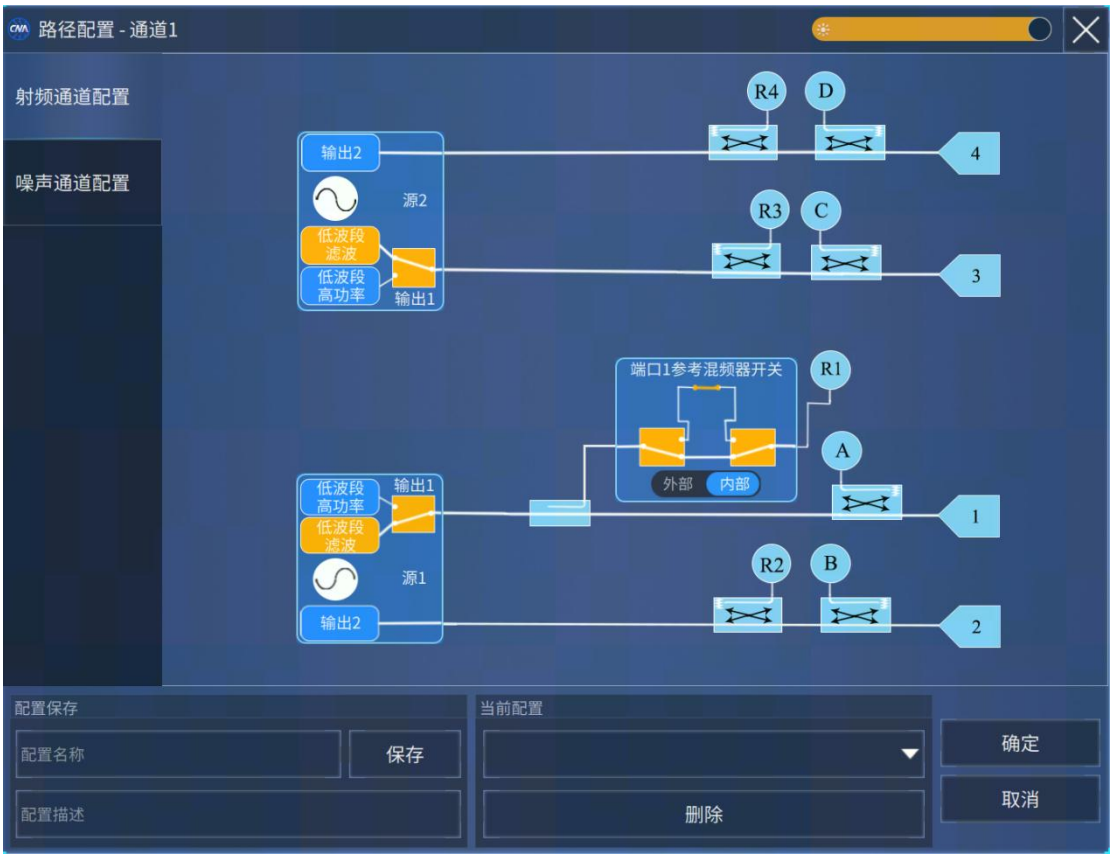


Figure 8.19 RF channel configuration

Pass configuration - 3674 default configuration Option 400

By default, source 2 is provided to port 3 and port 4.

Use the noise figure option application

Use the following procedure for noise figure measurements:

1. Connect the mixer and noise source.
2. Create a noise figure measurement.
3. Make the noise figure measurement settings.
4. Copy the ENR file to the appropriate folder.
5. Perform calibration.
6. Connect the device under test.
7. Measure the noise figure.
8. Click on [File] and save the noise figure data in the following format (ON when noise calibration is turned on): *.CTI Citfile, *.PRN, noise correlation matrix data.

Connect the noise tuner to the noise source

1. Connect the noise source to the 28V connector on the rear panel of the 3674 vector network analyzer. The noise source is automatically turned on or off as needed during calibration. Connect the noise tuner to the reference plane of port 2 when prompted during calibration.
2. Connect the noise tuner (Ecal module). No

connections are required for scalar measurements.

a. At the 3674 front panel, remove the jumper source output /source input from port 1.

b. Connect the M-F mixer using the cable and adapter provided.

c. If you use the F-F Ecal module, you will need to provide an M-M adapter.



Figure 8.20 Connecting the tuner

Create a noise figure measurement

1. In the 3674 front panel, click **[Measurement]**, and then select **[Measurement Class]**.

2. Select **[Noise Figure (Downconverter)]**, and then: Click **[OK]** to delete the existing measurement, or select **[Create in New Channel]**.

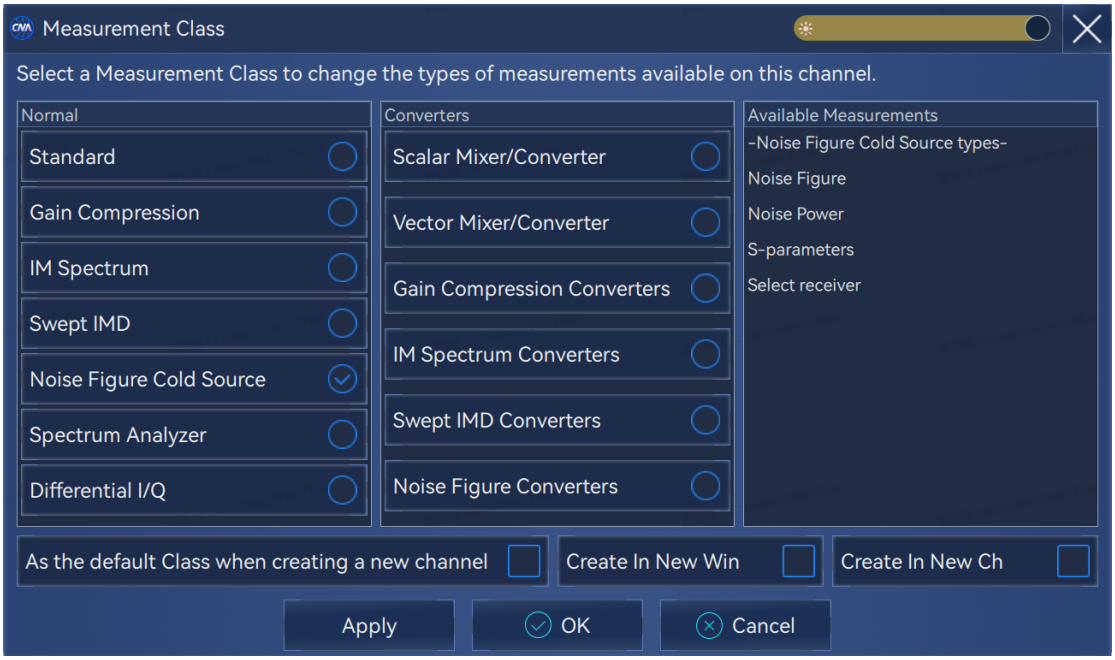


Figure 8.21 Create a noise figure measurement

3. A noise figure measurement is displayed. Add or change the parameters below.

Parameters of converter noise figure measurement

How to add NFX parameters

Create an NFX channel.

Then do the following:

Click **[Trace]**, then select **[New Trace]**, then select a parameter.

How to change NFX parameters

Create an NFX channel.

Select the noise parameter you want to change.

Then do the following:

[Measurement] → **[Measurement]**, and select a parameter.

The noise parameters in the general noise figure measurement have the same noise figure available for selection in the converter noise figure measurement NFX.

In addition, the following mixer and raw receiver parameters are provided in the NFX channel:

Mixer parameters

- SC21 - Frequency conversion loss
- SC12-Reverse frequency conversion loss
- S11-Input matching
- S22-Output matching
- IPWR- Input power to the mixer/converter. Same as R1 (source 1).
- OPWR-Output power to mixer/converter. Same as B (source 1).
- RevIPWR-The power supplied to the output of the mixer/converter. Same as R2 (source 2).
- RevOPWR-The power measured at the mixer/converter input. Same as A (source 2).

Raw receiver parameters

Specify a receiver with label to measure LO1 frequency:

<receiver> LO1

Example: ALO1 or R1LO1

Specify a receiver with label to measure the source port:

<receiver>_<source port〉

Example: A_3 or R1_1

How to open the Noise Figure Measurement dialog box

To provide quick access, use the Settings softkeypad.

Using the auxiliary soft keypad, press **[Frequency]**, then **[Noise Figure Setting]**.

Using the menu, click **[Response]**, then **[Measurement]**, then **[Noise Setting]**.

Verify mixer configuration/sweep type combinations

Table 8.2 Mixer sweep types

Swp Type	Input	LO	Output
Linear	Sweep	Sweep	Fix
	Sweep	Fix	Sweep
	Fix	Sweep	Sweep
Fixed	Fix	Fix	Fix

To determine the effective mixer configuration using two LO, a fixed LO and a swept LO are equal to a single-order swept LO.

If you create an invalid sweep type/mixer configuration, there will be a red error message as follows:

Notice

All frequencies must be set to fixed mode in point frequency or power sweep mode.

If this happens, change the sweep type in the Frequency tab.

Refer to the Mixer Test Configuration section for additional rules.

The following tabs can be set in the NFX dialog box:

Sweep

Power

Noise figure settings

Mixer frequency

Mixer power

Mixer settings

Sweep tab



Figure 8.22 Frequency tab for frequency converter noise figure measurement

Swp Type

Linear sweep - use linear sweep parameters

Point frequency - use fixed input parameters

Select X-axis

Select the frequency range displayed on the X-axis.

Sweep settings

You can set the number of points and IF bandwidth. IF Bandwidth is the IF bandwidth of the vector network receiver. This setting is useful to improve the accuracy of noise measurements.

Power tab

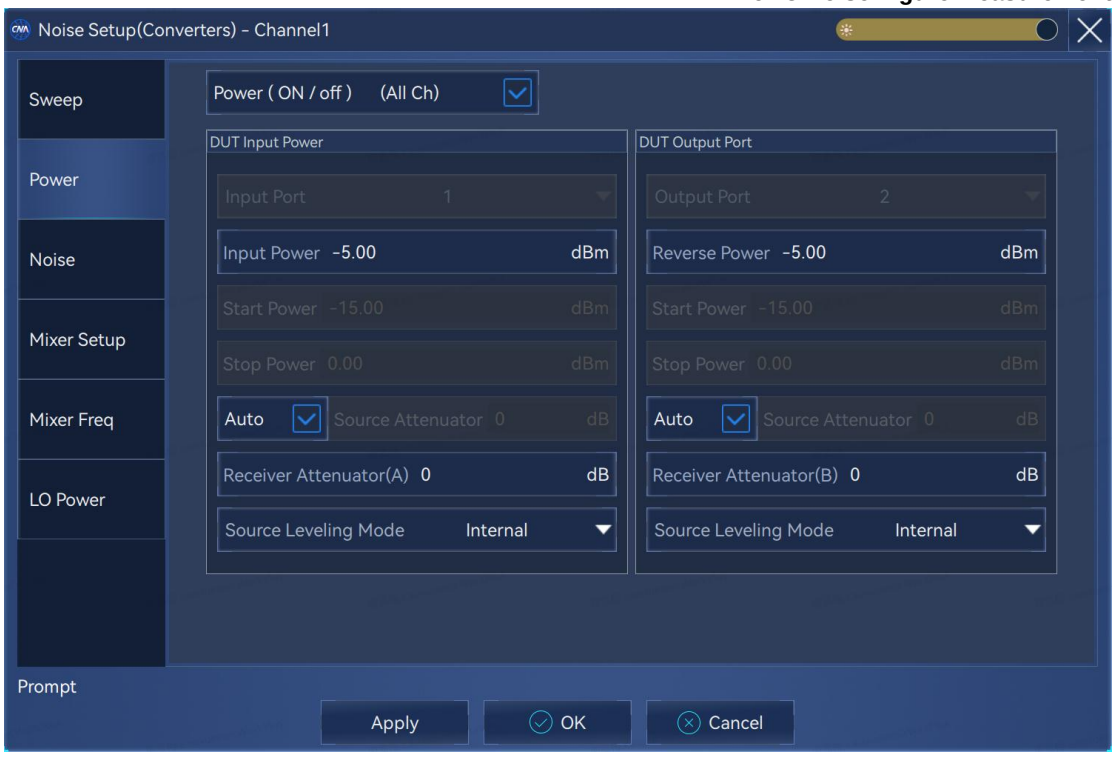


Figure 8.23 Converter noise figure measurement power tab

Noise channel configuration

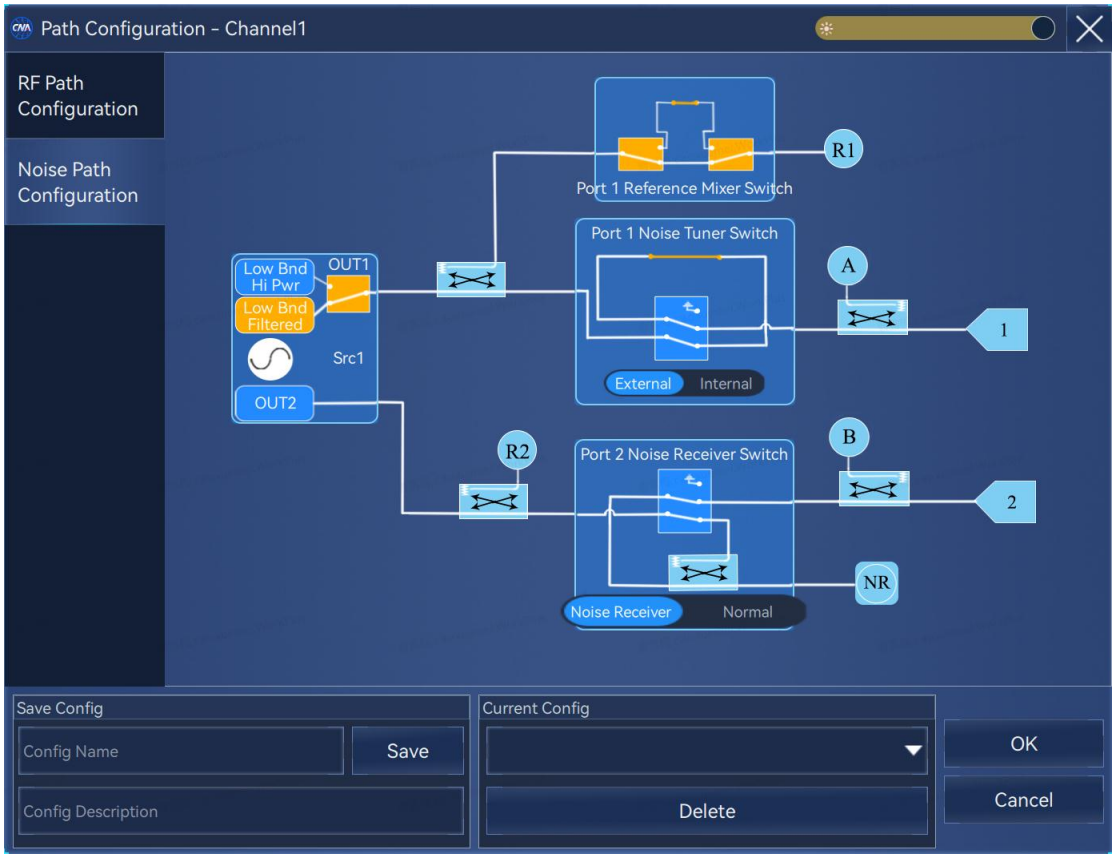


Figure 8.24 Noise channel configuration

Noise Figure Settings tab



Figure 8.25 Converter Noise Figure Noise Settings tab

This dialog box is the same as that in amplifier noise settings.

NFX Frequency tab

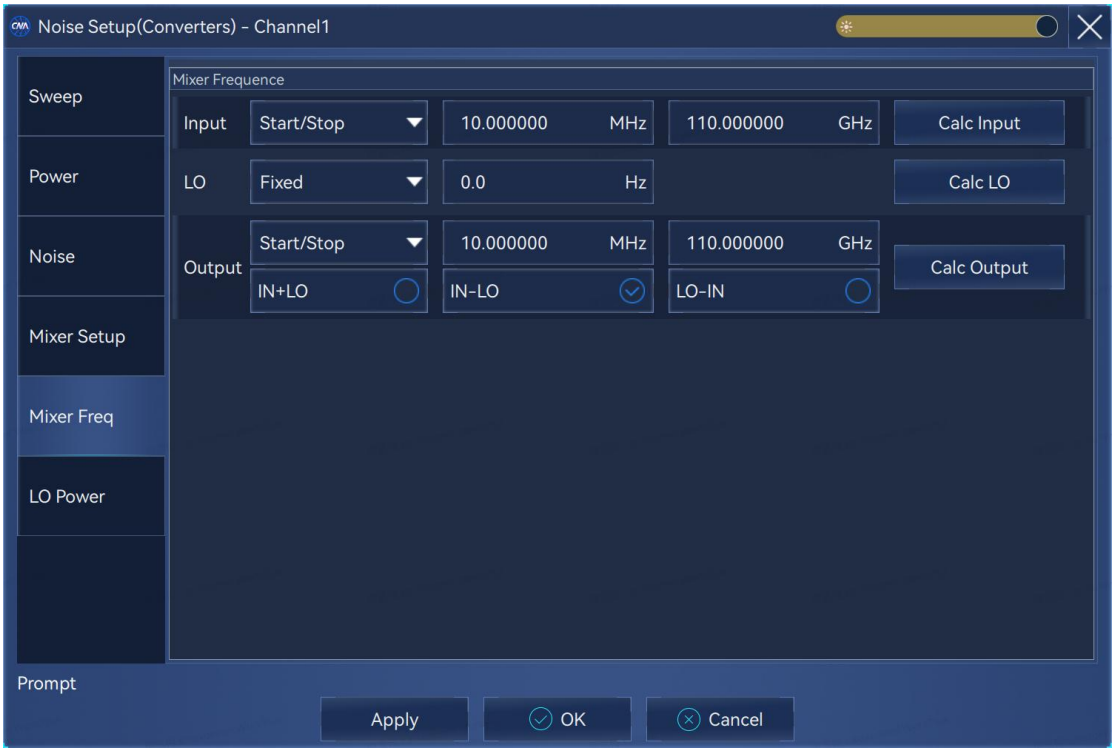


Figure 8.26 Mixer Frequency tab

NFX Mixer Settings tab

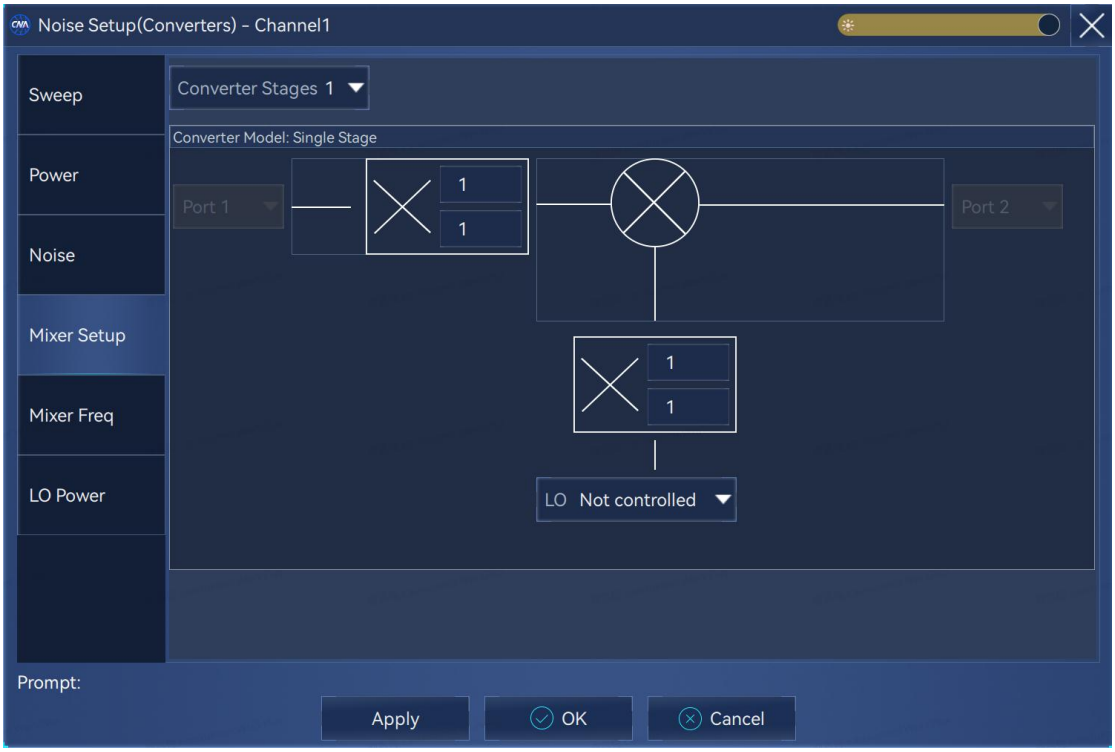


Figure 8.27 Mixer Settings tab

LO Power tab

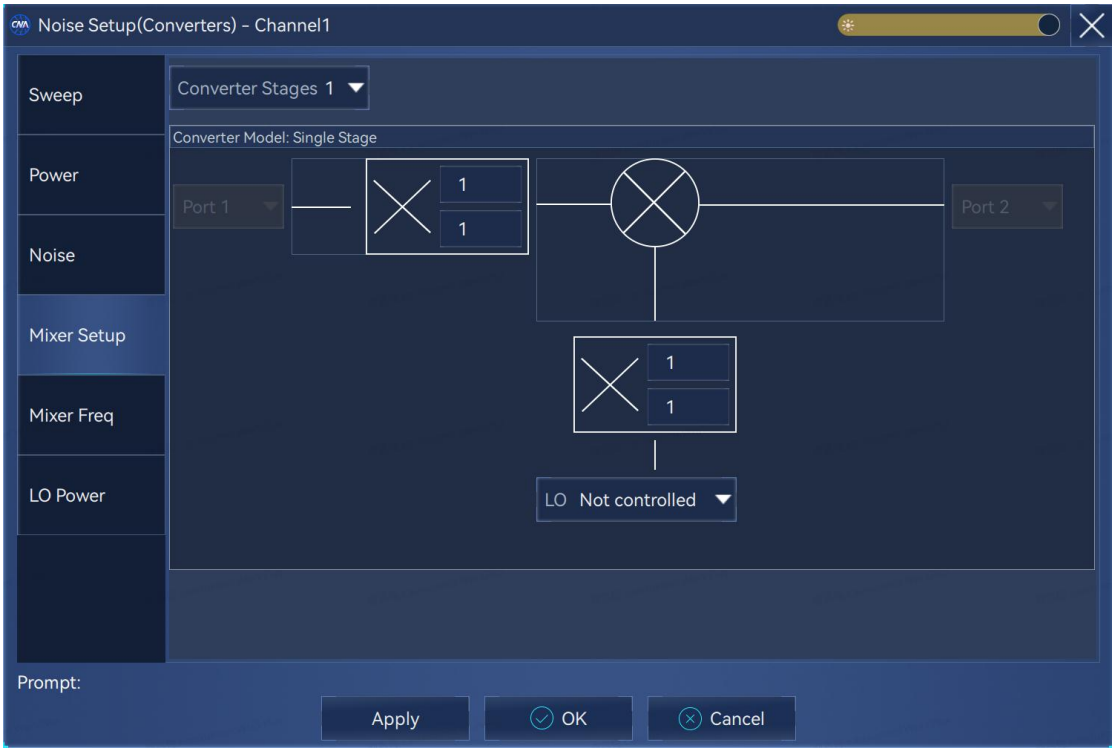


Figure 8.28 Mixer Settings tab

Annex 9 Advanced Time Domain Analysis

This chapter includes the following contents:

- Advanced Time Domain Analysis Overview 514
- Measurement Setting..... 515
- Measurement Process 520
- Eye Diagram and Eye Diagram Template Measurements 530
- Advanced Analysis of Eye Diagram Waveforms 530

9.1 Overview of Advanced Time Domain Analysis

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**

9.1.1 On-screen Display

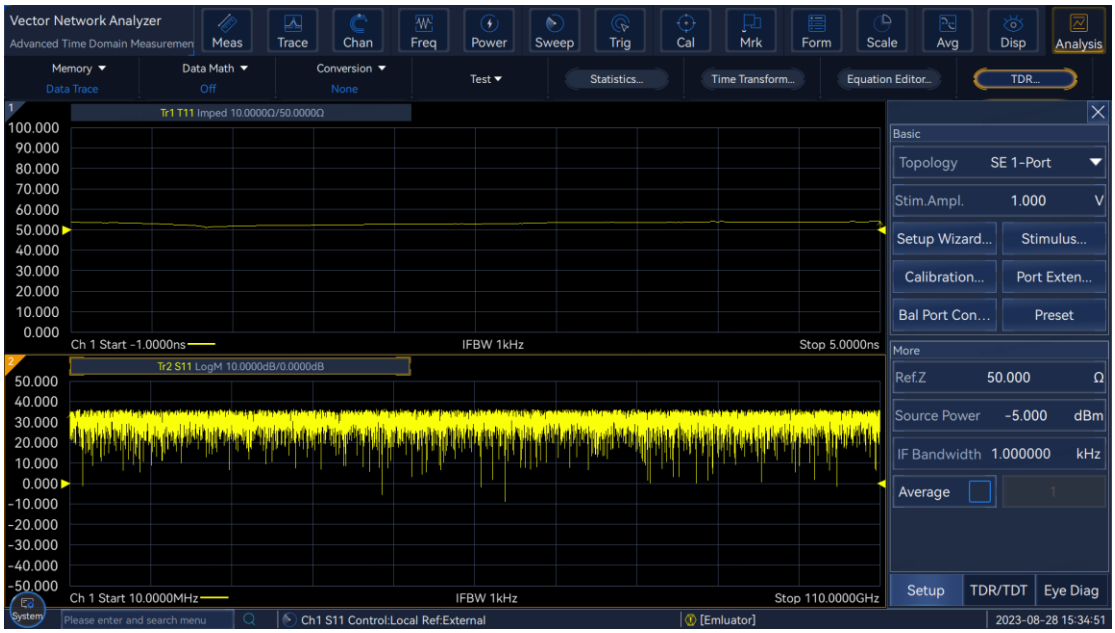


Figure 9.1 Advanced time domain analysis option display

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.

9.1.2 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 3674 Advanced Time Domain Analysis Option application screen to close it.

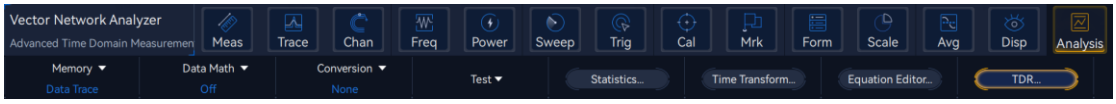


Figure 9.2 Advanced Time Domain Analysis Menu

9.2 Measurement Settings

9.2.1 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.

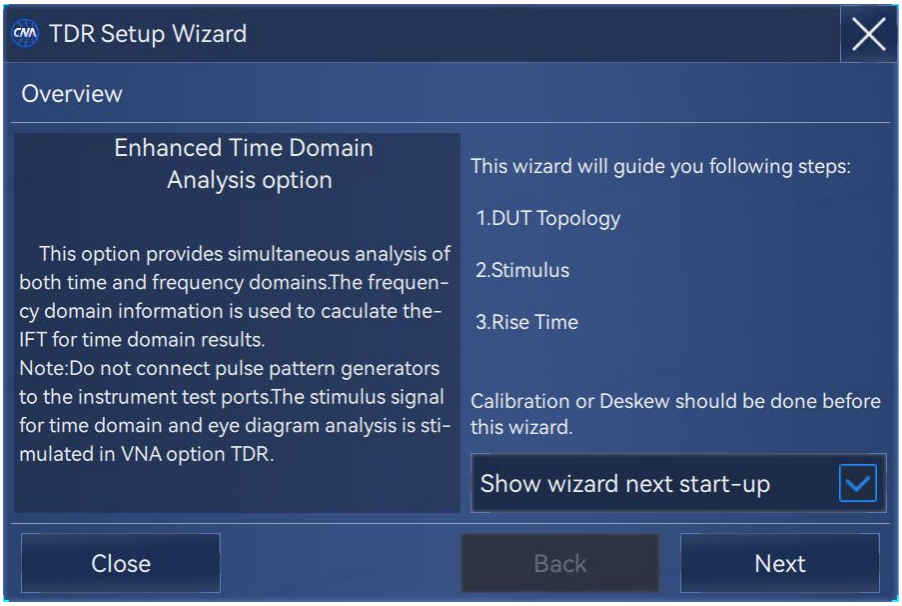


Figure 9.3 TDR configuration wizard

a) 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.
b) Click [Next] → Start the setting process.
c) Select the topology of the DUT.



Figure 9.4 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 9.5 Stimulus setting

e) Click **[Next]**.

f) After the Rise Time setting dialog box of the configuration wizard is displayed, click [Rise Time] and [Type].



Figure 9.6 Select rise time

g) Click [Finish] to finish the configuration wizard settings.

9.2.2 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.

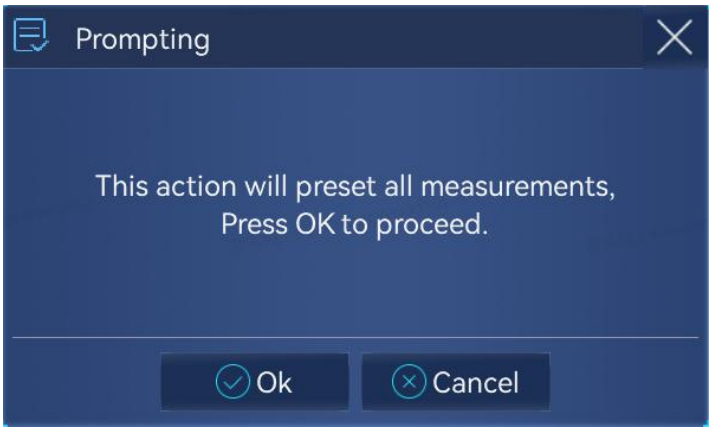


Figure 9.7 Change topology tip

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 9.8 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.

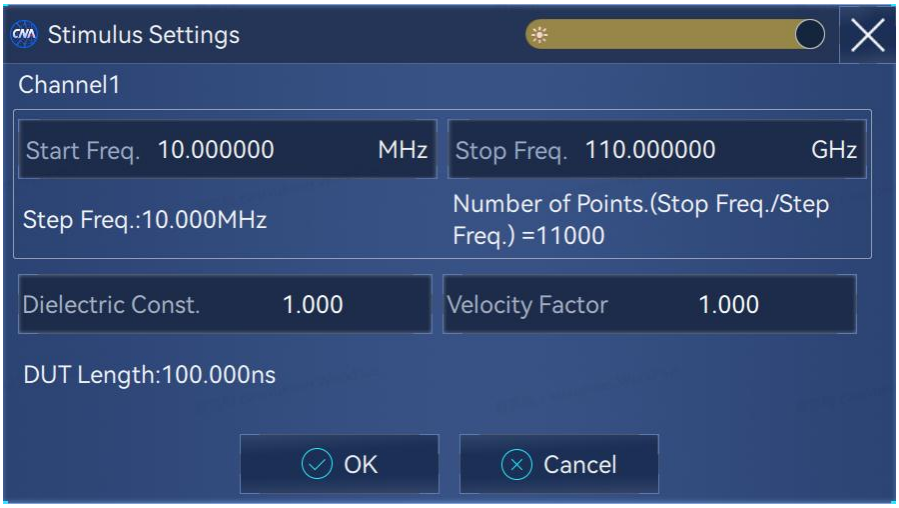


Figure 9.9 Measuring the length of the DUT

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 starting 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 (εr) are set via the velocity factor and electrical delay edit boxes. The relationship between the two is as follows: velocity factor =1/sqrt(εr)

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 setting.

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

Velocity factor = 1 /√(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) Averaging

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”.

9.2.3 Error Correction

There are many different ways to remove the effects of test fixtures and cables from a measurement. The 3674 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 9.10 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 9.11 Auto port extension

After clicking the [Calibrate] button, the "Calibration" dialog box will pop up, and refer to Chapter 7 for general calibration steps.

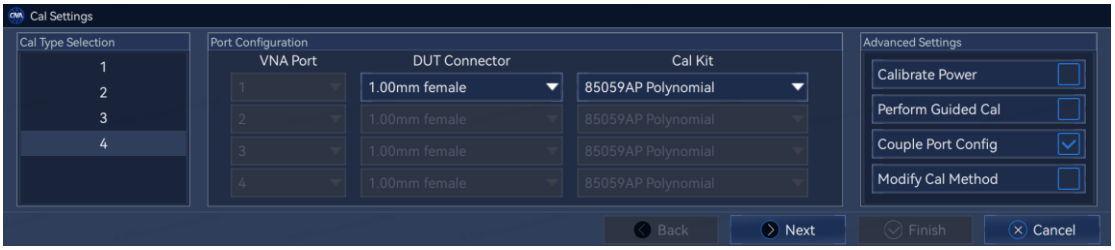


Figure 9.12 Calibration dialog box

9.3 Measurement Process

9.3.1 Trace Parameter Settings

Trace selection

There are two ways to select the displayed trace.

a) Using 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.

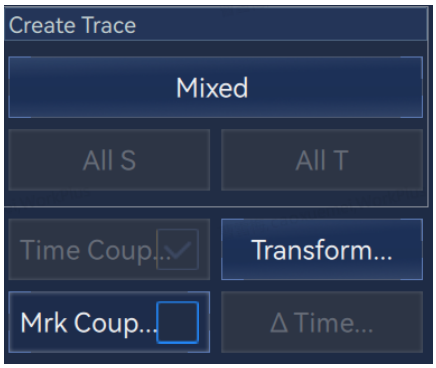


Figure 9.13 Trace control options

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.

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 9.1 Measurement formats

Measure	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

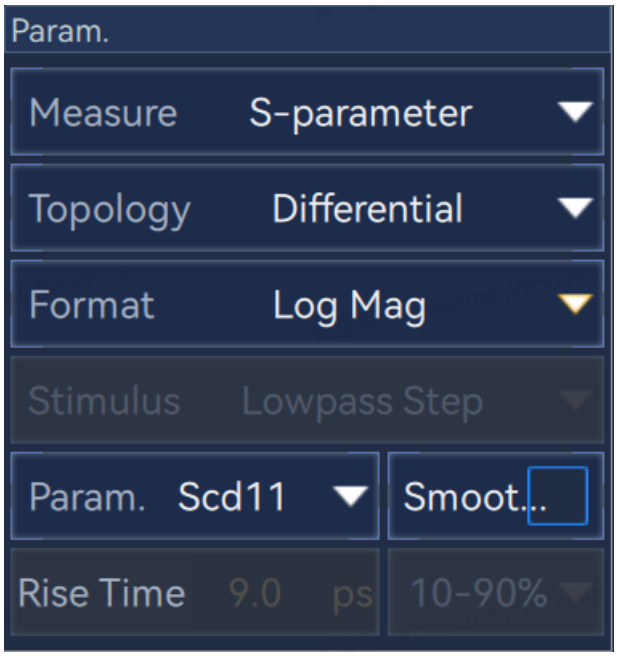


Figure 9.14 Parameter settings

The list boxes in the "Parameters" area change with the selection of the measurement, as shown below:

Exhibit 9.2 Measurement parameters

Measure		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.

9.3.2 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.

9.4 Eye Diagram and Eye Diagram Template Measurements

9.4.1 Eye Diagram Measurement

Eye diagrams are commonly used to analyze signal quality in the oscilloscope. You can determine problems such as effect of attenuation, noise, jitter and dispersion on the system, by using an eye diagram.

The 3674'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 the 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.

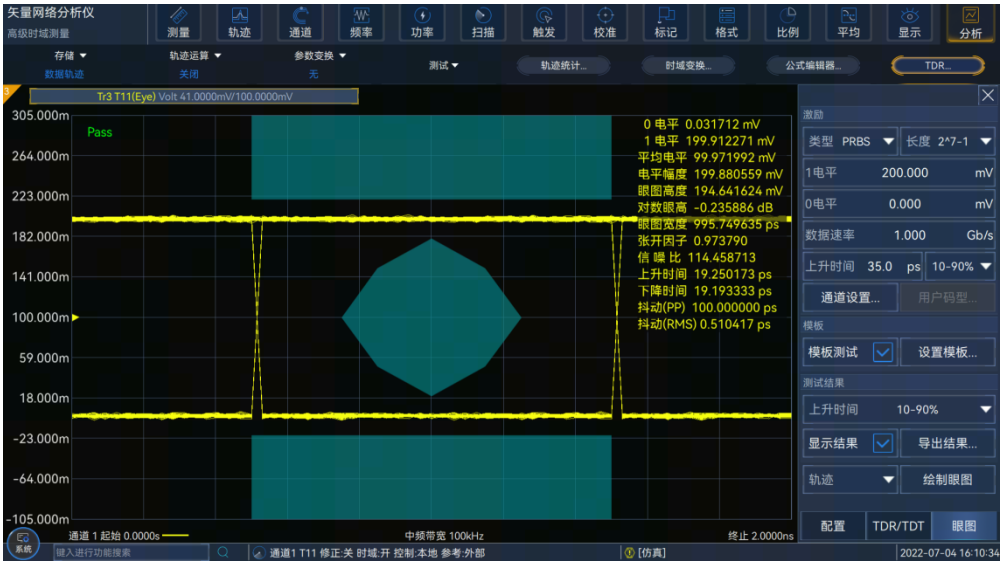


Figure 9.16 Eye diagram display

Figure 9.3 Eye diagram measurement parameters

Name	Unit	Description
Rising 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 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 × 100
Expansion	None	(1 level - σ1) - (0 level + σ0)/amplitude

factor		
Signal/noise ratio	None	(1 level-0 level)/(σ_1+σ_0)
Duty ratio distortion	Picosecond	$\lvert T_{\text{rise middle}} - T_{\text{fall middle}} \rvert$
Duty ratio distortion (%)	%	Duty ratio distortion (s)/ bit period × 100
0 level	Voltage	Histogram average of 0 level
1 level	Voltage	Histogram average of 1 level
Average level	Voltage	(0 level + 1 level)/2
Level amplitude	Voltage	1 level - 0 level
Eye diagram height	Voltage	(1 level - 3σ_1) - (0 level + 3σ_0)
Eye diagram width	Picosecond	Bit period - 2 × 3 × jitter RMS
(50%)		Bit period = 1/bit rate Input amplitude = set 1 level - set 0 level $T_{\text{rising middle}}$ = time for the rising edge to cross the intermediate threshold $T_{\text{falling middle}}$ = 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]**.

```
# Ceyear 3661 TDR Eye Results,
# 2019-10-16 15:43:02,
#,
Level Zero,       -0.000,
Level One,        0.200,
Level Mean,        0.100,
Amplitude,        0.200,
Eye Height,        0.190,
Eye Height,       -0.459,
Eye Width,        0.000,
Open Factor,         0.948,
SNR,       57.336,
Rise Time,        0.000,
Fall Time,        0.000,
Jitter PP,        0.000,
Jitter RMS,        0.000,
```

Figure 9.17 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.

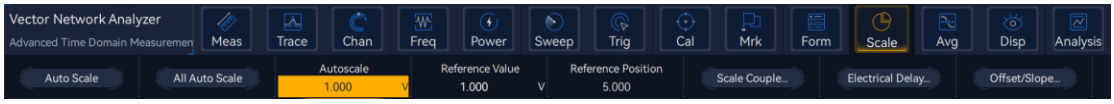


Figure 9.18 Eye diagram scaling settings

9.4.2 Simulation Code Type Selection

The 3674'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].

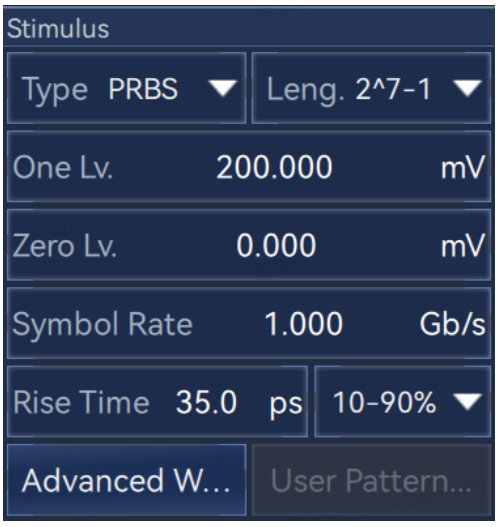


Figure 9.19 Stimulus setting

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 9.4 Description of simulation code type parameters

Label	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.
Rising 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].

9.4.3 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 standards, the input waveform must remain outside the shaded template area. The template test can be found under "Scale".



Figure 9.20 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".

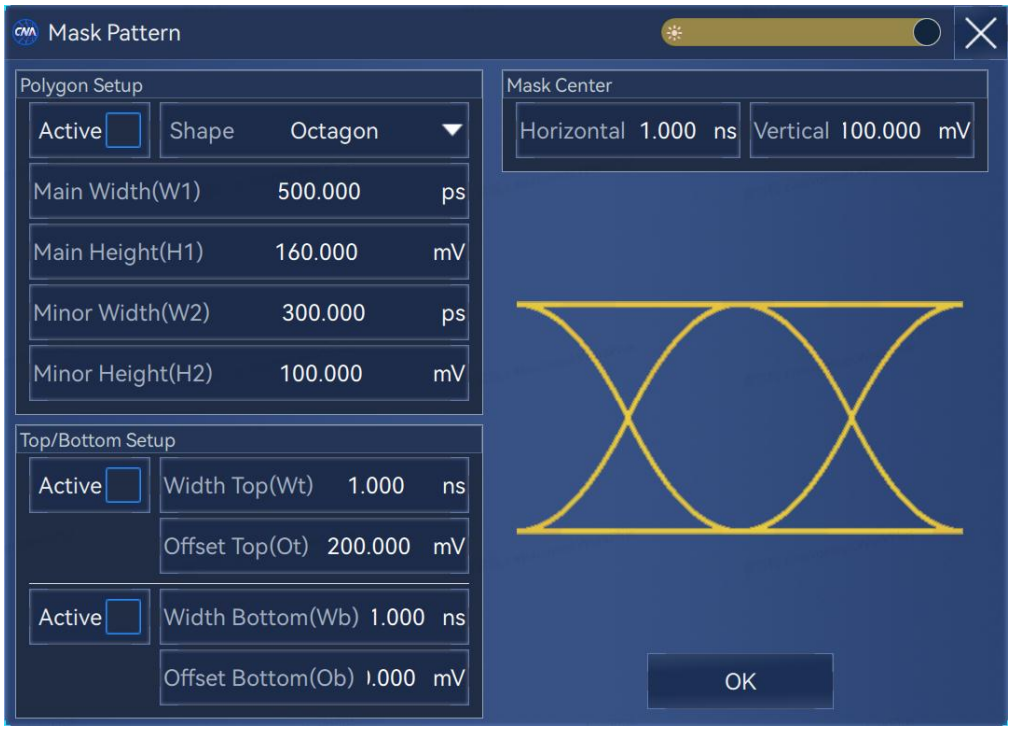


Figure 9.21 Customize templates

b) Execute the template test

1. Select the [Template Test] checkbox 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.

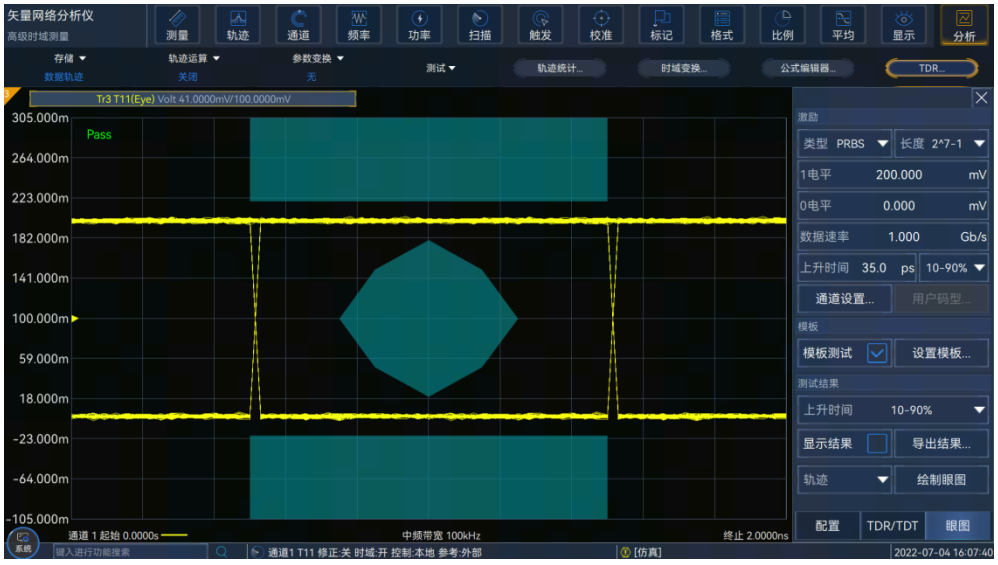


Figure 9.22 Template test

9.5 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 9.23 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.

9.5.1 Jitter Injection

a) Click the [Jitter] button and check [Enable] to perform jitter injection to the source.
b) When jitter injection is 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 9.24 Jitter injection settings

9.5.2 Noise Injection

a) Click the [Noise] button and check [Enable] to apply noise injection to the source.
b) When noise injection is on, a check mark will appear on the [Noise] button.

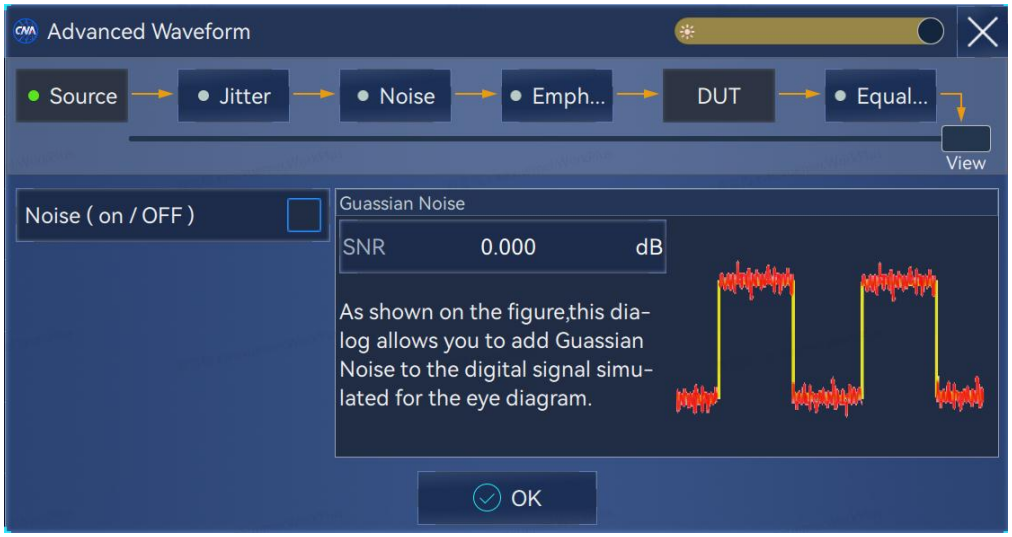


Figure 9.25 Noise injection settings

9.5.3 Pre-emphasis

a) To perform pre-emphasis, click the [Pre-emphasis] button and check [Enable].
b) When pre-emphasis is on, a check mark appears on the [Pre-emphasis] button.
•The pre-marker is the ratio of Vd and Vc:

Pre-marker = 20 Log10 (Vd/Vc)

•Post-1 marker is the ratio of Vb and Va:

Post 1 marker = 20 Log10 (Vb/Va)

•Post 2 marker is the ratio of Vc and Vb:

Post 2 marker = 20 Log10 (Vc/Vb)

Notice

When pre-emphasis is on, the sum of the absolute values of the pre-emphasis FIR filters is normalized.

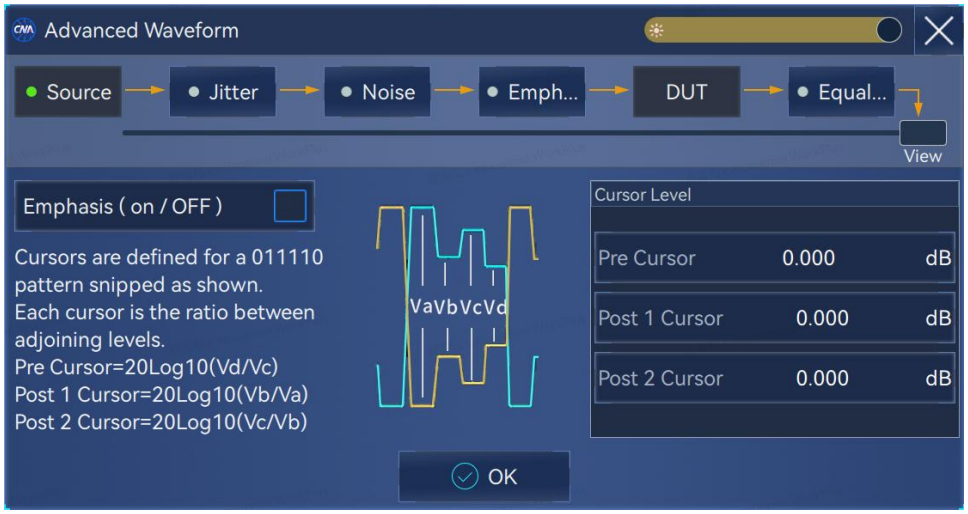


Figure 9.26 Pre-emphasis settings

9.5.3 Equalization

a) To perform equalization, click the [Equalize] button and check [Enable].
b) When pre-emphasis is on, a check mark will appear on the [Equalize] button.

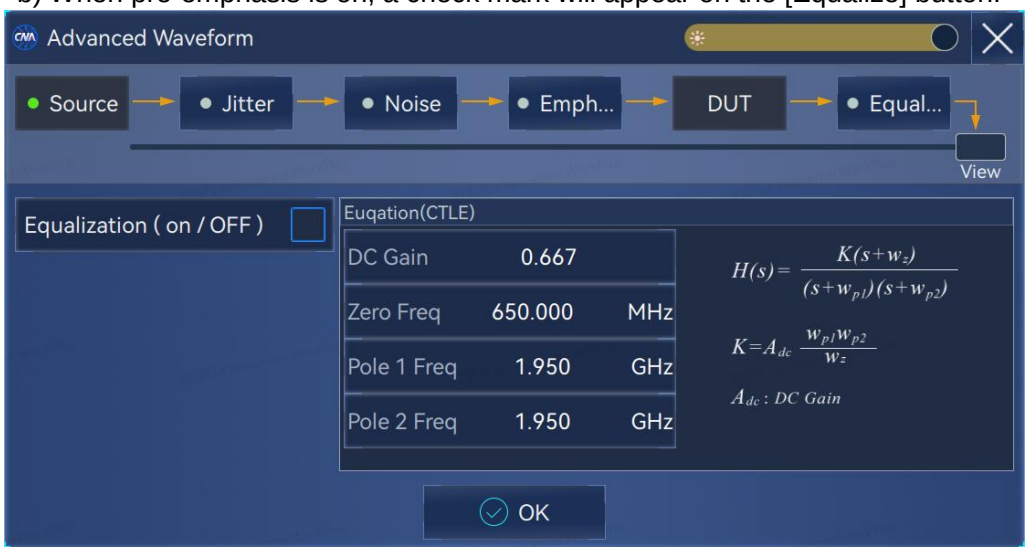


Figure 9.27 Equalization settings

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

Annex 10 Fast Continuous Point Measurement Mode

This chapter includes the following contents:

- Overview.....593
- Measurement Description 593
- Remote Control Command Descriptions 594

10.1 Overview

The fast continuous point measurement mode is a proprietary measurement mode provided by the vector network analyzer for antenna testing. In this mode, the network analyzer acquires measurement data with minimal delay and returns it to the programmer. For faster execution, the fast continuous point measurement mode does not support interface operation and can only be operated by remote control commands.

10.2 Measurement description

In the fast continuous point measurement mode, the network analyzer will maintain a 4GB FIFO that can store 1,000,000 data, which is the original data before calibration. The data in the FIFO is stored in the following format:

4-port

R , A , B , C , D ,
R/R , A/R , B/R , C/R , D/R
R/A , A/A , B/A , C/A , D/A
R/B , A/B , B/B , C/B , D/B
R/C , A/C , B/C , C/C , D/C
R/D, A/D , B/D, C/D , D/D

2-port

R1 , R2 , A , B ,
R1/R1, R2/R1, A/R1, B/R1
R1/R2, R2/R2, A/R2, B/R2
R1/A, R2/A, A/A, B/A
R1/B, R2/B, A/B, B/B

Figure 10.1 Data storage format

The original data stored in each measurement parameter includes two parts: real part and imaginary part, so one measurement point includes 2 data (I and Q). When the SYSTem:FIFO:DATA:COUNt? query command is sent, the number of actual stored data is returned, which is equal to the specified number of stored points*2.

When sending SYSTem:FIFO ON command, it will enable the network instrument fast continuous point measurement mode. At this time, no matter what sweep mode the current analyzer channel is currently in, the FIFO function (please make sure the other channels are in the hold state) will be enabled. When the FIFO is not turned off, the uncorrected original measurement data collected by the network analyzer are written into the FIFO in order, and the user can use the remote control command SYSTem:FIFO:DATA? dpoints to read dpoints of measurement data.

If the analyzer is in CW mode, the number of points to be stored in the FIFO can be set by the command SENSe<cnum>:SWEep:TYPE:FACW pts; the number of points can be larger than the number of points swept by the network analyzer (up to 200,001 points).

When the pts is greater than 0 and less than 2~31-1, the network analyzer loops and stores the data in the FIFO; when the pts equals -1, the analyzer stores data in the FIFO until the user sends the command SYSTem:FIFO OFF; when the pts equals 0, this function is turned off.

Send SYSTem:FIFO:DATA:COUNt? The command queries how much data has been written so far; when the command SYSTem:FIFO:DATA? dpoints is sent, the data with the length of dpoints is returned. When sending the command SYSTem:FIFO OFF, the FIFO will be emptied and the fast continuous point measurement mode will be turned off.

10.3 Description of Remote Control Commands

The remote control commands related to the fast continuous point measurement mode are described as follows:

1): SYSTem:FIFO:DATA? <dpoints> read-only

Read the specified length of data (real part/imaginary part format) sequentially from the FIFO, and clear this segment of data after reading.

The return data format, (<REAL,32>, <REAL,64> or <ASCii,0>), can be specified with FORMat:DATA.

The order of the returned data bytes can be changed with FORMat:BORDer.

Note: Because of the remote control limitation, the maximum number of points that can be read by FIFO at one time cannot exceed the maximum number of points swept by the network analyzer.

2)SYSTem:FIFO:DATA:BYTe:COUNt? Read only

Return the length of bytes of readable data in the FIFO buffer. Return type: Shaping.

3)SYSTem:FIFO:DATA:CLEar write only

Clear the FIFO buffer.

4):SYSTem:FIFO:DATA:COUNt? Read only

Return the number of data in the FIFO buffer.

5) SYSTem:FIFO[:STATe] <bool> Read/Write

FIFO buffer read/write status, shutdown operation, system reset or callback will clear the FIFO.